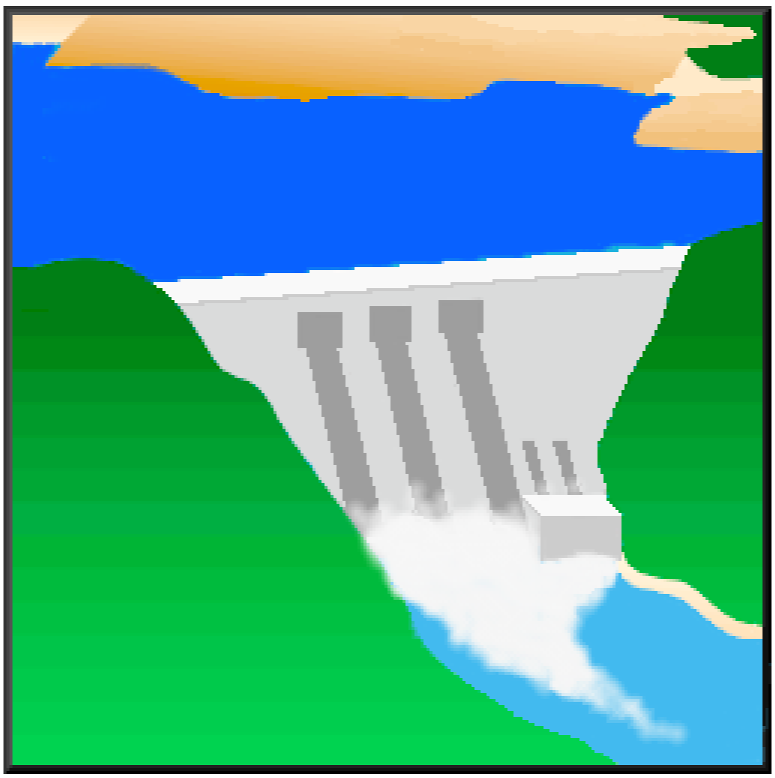
US Army Corps
of Engineers
Hydrologic Engineering Center

HEC-ResSim

Reservoir System Simulation



User's Manual

Version 3.3

February 2021

Approved for Public Release. Distribution Unlimited.

CPD-82

REPORT DOCUMENTATION PAGE						Form Approved OMB No. 0704-0188
The public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to the Department of Defense, Executive Services and Communications Directorate (0704-0188). Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number. PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ORGANIZATION.						
1. REPORT DATE *(DD-MM-YYYY)* February 2021			2. REPORT TYPE Computer Program Documentation			3. DATES COVERED *(From - To)*
4. TITLE AND SUBTITLE HEC-ResSim Reservoir System Simulation User's Manual Version 3.3						5a. CONTRACT NUMBER 5b. GRANT NUMBER 5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S) Joan D. Klipsch George C. Modini Sara M. O'Connell Marilyn B. Hurst Daniel L. Black						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-82
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 Quick Start Guide, CPD-82a						
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				
U	U	U	UU	670	19b. TELEPHONE NUMBER	

Standard Form 298 (Rev.8/98) Prescribed by ANSI Std. Z39-18

HEC-ResSim

Reservoir System Simulation

User's Manual

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-82

HEC-ResSim, Reservoir System Simulation

Software Distribution and Support Statement

2013. The HEC-ResSim software and documentation was developed by the US Army Corps of Engineers, Hydrologic Engineering Center (HEC) with U.S. Federal Government resources and is therefore in the public domain. It may be used, copied, distributed, or redistributed freely. However, it is requested that HEC be given appropriate acknowledgment in any subsequent use of this work.

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 for Use before the software can be installed or used. For reference, a copy of the Terms and Conditions for Use are included below so that they may be examined before obtaining the software. The software described by this document can be downloaded at no cost from our web 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.

This document contains references to product names that are trademarks or registered trademarks of their respective owners. Use of specific product names does not imply official or unofficial endorsement. Product names are used solely for the purpose of identifying products available in the public marketplace.

Microsoft and *Windows* are registered trademarks of Microsoft Corporation.

Solaris and *Java* are registered trademarks of Oracle Corporation and/or its affiliates.

Terms and Conditions for Use of HEC-ResSim

The United States Government, US Army Corps of Engineers, Hydrologic Engineering Center ("HEC") grants to the user the rights to install HEC-ResSim "the Software" (either from a disk copy obtained from HEC, a distributor or another user or by downloading it from a network) and to use, copy and/or distribute copies of the Software to other users, subject to the following Terms and Conditions of Use:

All copies of the Software received or reproduced by or for user pursuant to the authority of this Terms and Conditions of Use will be and remain the property of HEC.

User may reproduce and distribute the Software provided that the recipient agrees to the Terms and Conditions for Use noted herein.

HEC is solely responsible for the content of the Software. The Software may not be modified, abridged, decompiled, disassembled, un-obfuscated, or reverse engineered. The user is solely responsible for the content, interactions, and effects of any and all amendments, if present, whether they be extension modules, language resource bundles, scripts or any other amendment.

No part of this Terms and Conditions for Use statement may be modified, deleted or obliterated from the Software.

No part of the Software may be exported or re-exported in contravention of U.S. export laws or regulations.

Waiver of Warranty

THE UNITED STATES GOVERNMENT AND ITS AGENCIES, OFFICIALS, REPRESENTATIVES, AND EMPLOYEES, INCLUDING ITS CONTRACTORS AND SUPPLIERS PROVIDE HEC-RESSIM "AS IS," WITHOUT ANY WARRANTY OR CONDITION, EXPRESS, IMPLIED OR STATUTORY, AND SPECIFICALLY DISCLAIM ANY IMPLIED WARRANTIES OF TITLE, MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NON-INFRINGEMENT. Depending on state law, the foregoing disclaimer may not apply to you, and you may also have other legal rights that vary from state to state.

Limitation of Liability

IN NO EVENT SHALL THE UNITED STATES GOVERNMENT AND ITS AGENCIES, OFFICIALS, REPRESENTATIVES, AND EMPLOYEES, INCLUDING ITS CONTRACTORS AND SUPPLIERS, BE LIABLE FOR LOST PROFITS OR ANY SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES ARISING OUT OF OR IN CONNECTION WITH USE OF HEC-RESSIM REGARDLESS OF CAUSE, INCLUDING NEGLIGENCE.

THE UNITED STATES GOVERNMENT'S LIABILITY, AND THE LIABILITY OF ITS AGENCIES, OFFICIALS, REPRESENTATIVES, AND EMPLOYEES, INCLUDING ITS CONTRACTORS AND SUPPLIERS, TO YOU OR ANY THIRD PARTIES IN ANY CIRCUMSTANCE IS LIMITED TO THE REPLACEMENT OF CERTIFIED COPIES OF HEC-RESSIM WITH IDENTIFIED ERRORS CORRECTED.

Depending on state law, the above limitation or exclusion may not apply to you.

Indemnity

As a voluntary user of HEC-ResSim you agree to indemnify and hold the United States Government, and its agencies, officials, representatives, and employees, including its contractors and suppliers, harmless from any claim or demand, including reasonable attorneys' fees, made by any third party due to or arising out of your use of HEC-ResSim or breach of this Agreement or your violation of any law or the rights of a third party.

Assent

By using this program you voluntarily accept these terms and conditions. If you do not agree to these terms and conditions, uninstall the program and return any program materials to HEC (If you downloaded the program and do not have disk media, please delete all copies, and cease using the program).

Table of Contents

List of Chapters

List of Tables

List of Figures

List of Appendices

List of Appendix Tables

List of Appendix Figures

Chapter 1
Introduction

Welcome to the Hydrologic Engineering Center's *Reservoir Simulation* (HEC-ResSim) computer program. HEC-ResSim is the successor to HEC-5, *Simulation of Flood Control and Conservation Systems*" (HEC, 1998). HEC-ResSim is comprised of a graphical user interface (GUI), a computation engine (to simulate reservoir operations), data storage and management capabilities, and plotting and reporting facilities. HEC-DSS, the Hydrologic Engineering Center's *Data Storage System*, (HEC, 1995 and HEC, 2009) is used for storage and retrieval of input and output time-series data.

1.1 Starting ResSim

To begin modeling reservoirs, you must first start ResSim. Like most other Windows applications, you can launch ResSim by:

- Double-clicking the **HEC-ResSim** icon on your Windows desktop (Figure 1.1), or
- Selecting **HEC-ResSim** from the **HEC** menu in the Windows **Start** menu (Figure 1.2).

Figure 1.1
ResSim Desktop Icon

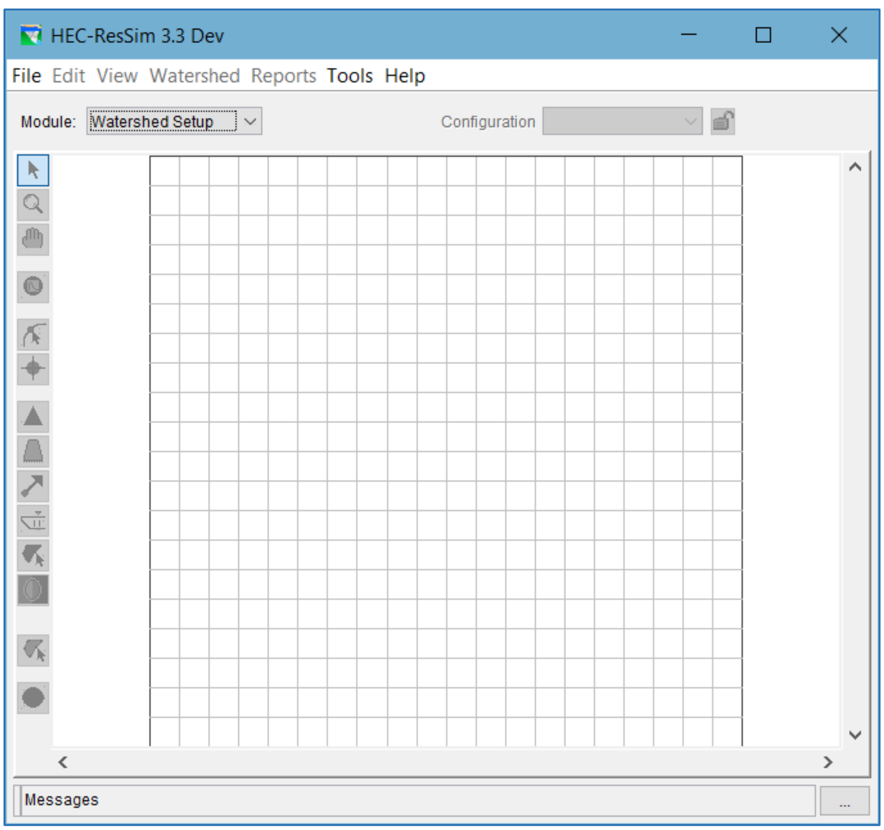


Figure 1.2 ResSim Main Window

After a moment or two, ResSim's main window will open (Figure 1.2). The short delay before ResSim displays its main window is because ResSim is a Java application and, as such, the Java Runtime Environment must be launched before ResSim can be loaded.

1.2 A ResSim Watershed

In the physical world, a watershed is a region of land that drains into a stream, river system, or other body of water. In the hydrologic modeling world, a watershed is a dataset associated with a geographic region that drains into a system of rivers and reservoirs for which you may develop one or more models in one or more modeling applications (e.g., HEC-HMS, HEC-ResSim, HEC-RAS, etc.). A watershed may include some or all of the streams, projects (reservoir, levees, etc.), gage locations, impact areas, time-series locations, and hydrologic and hydraulic data for the associated geographic area. All of these details together, once configured, form a watershed data set.

Since ResSim is used to model the system of rivers and reservoirs of a watershed, the term *watershed* is used to label the set of files and folder that represent a ResSim model. Thus, a ResSim project (dataset) is called a ***watershed***.

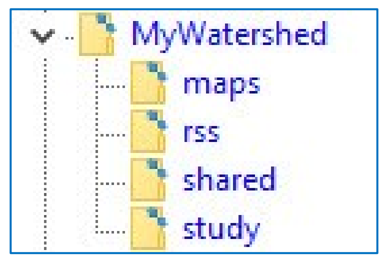


Figure 1.3 Watershed Folder Structure

When you create a new watershed, ResSim generates a folder structure named for the watershed in which it will store all files associated with the watershed. The watershed folder structure is illustrated in Figure 1.3.

Two files are stored in the root folder of the watershed:

- *watershed*.wksp - where *watershed* is the name you gave to the watershed when you created it and .wksp is appended to the watershed name to identify the file type. The .wksp file is your "project" file.
- stream.align - the file containing the stream alignment for your watershed.

Four subfolders are created in the watershed folder:

- maps - the maps folder is where you should put a copy of the background maps you will display with your watershed schematic
- rss - the rss folder is where ResSim will store the networks and alternatives that are created in the **Reservoir Network** module as well as the simulation folders that represent the simulations created in the **Simulation** module
- shared - the shared folder is where you should store the HEC-DSS files from which that your alternatives get their time series data

- study - the study folder is where ResSim stores most of the watershed configuration and its associated schematic elements defined in the **Watershed Setup** module

The modules mentioned above are introduced in the next section of this chapter and described in detail in subsequent chapters.

1.2.1 Understanding Watershed Directories

When you create a new watershed, ResSim creates a watershed tree in the directory you indicated. The watershed folder (the root of the watershed tree) is assigned the same name you gave to the watershed.

For example, in Figure 1.4 the watershed folder is *Hayes_Basin*. The full path to the folder (or directory) where the watershed *Hayes Basin* is stored is *D:\CurrentProjects\Example Watershed;* this is the path you would specify if you created a *shortcut* to your watersheds.

The watershed tree stores all of the data for the watershed, including maps, schematic elements, model configurations, simulation data, and results. The watershed tree is highlighted in yellow in Figure 1.4.

When you create a new simulation, ResSim generates a **simulation** directory named (by default) according to the date and time of the simulation. In Figure 1.4, the simulation directory name *1999.10.18-2400* corresponds to a simulation for October 18, 1999 at midnight. ResSim also copies all of the model data that define your selected alternatives into the simulation tree. This facilitates archiving of simulation information and ensures consistency in your model results.

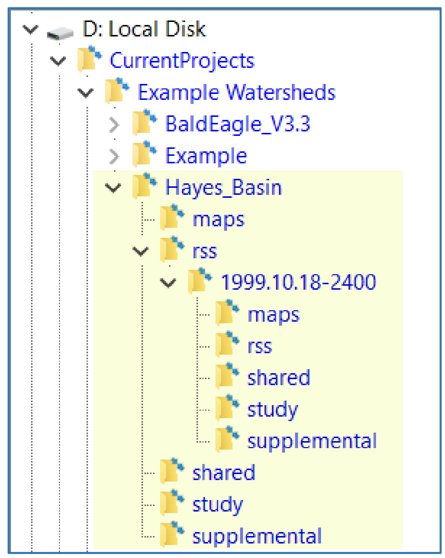


Figure 1.4 Example Watershed Tree

If you **Save to Base Directory** in the Simulation Module, ResSim copies your model alternative data from the simulation directory back to the base directory. If you **Replace From Base Directory** in the Simulation Module, ResSim copies the original model alternative data from your base directory into your simulation directory. See Section 15.8 for more information about these commands.

1.3 Opening an Existing Watershed

Once you have launched ResSim, you will need to open a watershed. Here's how:

- Select **Open Watershed** from the **File** menu (of any module). The **Open Watershed** browser will appear (Figure 1.5).

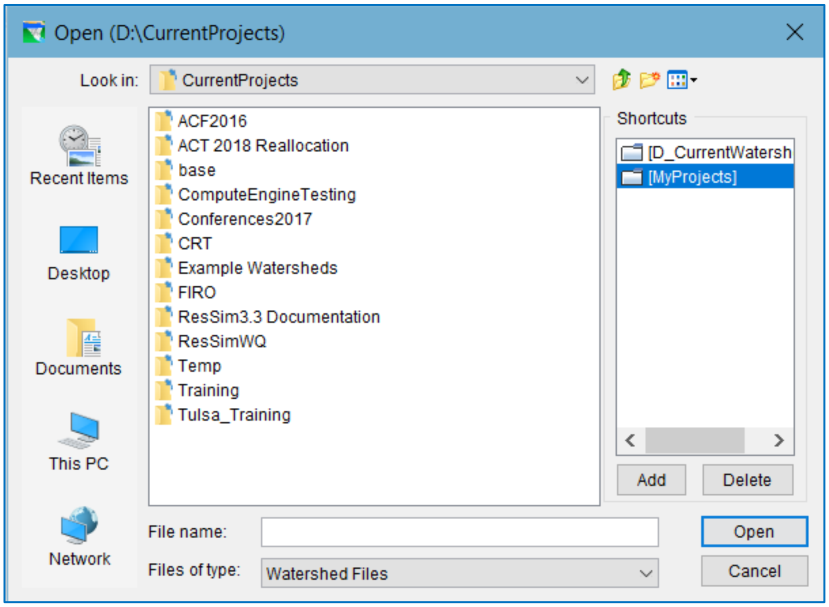


Figure 1.5 Open Watershed Browser

- Navigate (browse) to and select your watershed’s .wksp file. The *.wksp* file is the ResSim “project” file. This file is located in your watershed’s root or main folder and its filename is the name of your watershed with an extension of .wksp. The .wksp file identifies the watershed to ResSim.
- Click the **Open** button. The **Open Watershed** browser will close and the watershed you selected will load and appear in ResSim’s main window with the watershed name shown in the **Title Bar**.

The options for opening a watershed are described in more detail in section 2.4

1.4 About this Manual

This User’s Manual and its companion, the *HEC-ResSim Quick Start Guide,* are the primary references for providing ResSim users with instructions for creating ResSim models and analyzing simulation results. With the exception of Chapters 1 and 2 (which deal with introductory and core concepts), the chapters of this manual group tasks according to module. Each chapter provides an overview of key concepts and a detailed description of the user interface components and tools available in the module. Appendices cover topics and tasks that require levels of detail beyond the scope of an individual chapter. The organization of this manual is summarized in Table 1.1.

The *HEC-ResSim Quick Start Guide* provides a brief introduction and overview of the HEC-ResSim capabilities while describing the process for developing a reservoir model with ResSim. Throughout the *Guide* are references to this User’s Manual for more detailed information on topics and features that are only briefly described in the *Quick Start Guide*.

Table 1.1 Contents Summary of the HEC-ResSim User's Manual

If you want to learn about…	Refer to…
HEC-ResSim concepts	Chapter 2
How to create and manage watersheds	Chapter 2
How to work with map layers	Chapter 3, Appendix B
How to create watershed configurations	Chapter 3
How to establish a stream alignment	Chapter 4
How to create and define watershed elements and projects (e.g., reservoirs, levees, etc.)	Chapter 5
How to create computation points (hydrograph locations)	Chapter 5
How to develop reservoir networks	Chapter 6
How to edit element data (junctions, routing reaches and diversions)	Chapter 7
How to define and edit physical reservoir data	Chapter 9
How to develop reservoir operations…	
…operation sets	Chapter 10
…operational rules (downstream control, induced surcharge, hydropower, pumps, tandem)	Chapter 11
…Boolean statements for operations	Chapter 12
…scripted operations — state variables and scripted rules	Chapter 12
…special operations — outage schedules, storage credit, decision schedules, projected elevation	Chapter 12
How to import network properties from another watershed	Chapter 12
How to define reservoir systems	Chapter 13
How to define alternatives	Chapter 14
How to develop and execute simulations	Chapter 15
How to calibrate model	Chapter 15
How to manage simulations	Chapter 15
How to plot results	Chapter 16
How to develop reports	Chapter 17
Detailed application settings	Appendix A
How to define the coordinate system	Appendix B
How to edit maps	Appendix B
How to change colors of messages	Appendix A
How to use HEC-DSSVue	Appendix D
How to copy, export, and print tables	Appendix E
Yield Analysis	Appendix F
Ensemble computing	Appendix G
Monte Carlo computing	Appendix H
Operation Support Interface	Appendix I
References	Appendix J

1.4.1 Document Conventions

A variety of text formatting is used throughout this manual to emphasize the significance of a word or phrase. The text formatting follows these conventions:

"ResSim"	HEC-ResSim is the software's full name but it is easier to say and type just ResSim, so we do.
Bold Text	Words or phrases shown in **Bold Text in Title Case** indicate that the word or phrase refers to something you will see somewhere in the ResSim user interface. It may be the name of a menu, an option in a menu, the name of a dialog or editor, or the name of a data entry field or region.
Italic Text	*Italic text* is used to *emphasize* a word or phrase in a sentence. *Italic text* is also used identify *the name of a document* or file. A word or phrase shown in *ItalicText* in *camelCase* is a placeholder for something you should enter in its place. For example: *WatershedName.*wksp is a reference to a file that has the name of your watershed and the extension .wksp.
Underlined Text	Words or phrases that have been underlined represent the definition for a word or phrase in ResSim's terminology.
Menu → Menu Item	The → (arrow symbol) is used between menu selections in a sequence of selections or steps in sequence of steps. For example: **Edit→Reservoir** can be read as "from the **Edit** menu, select **Reservoirs**".
"…left mouse button…"	By default, in Windows (and most other windowed operating systems), the *left* button is the *primary* button on a *standard* mouse. We recognize that not everyone uses a standard mouse or uses their mouse with the default settings. However, for efficiency, we have written most instructions with the assumption that you, the user, are using default settings. If you are using a non-standard mouse or are not using default settings for your mouse, we leave it up to you to translate "left mouse button" to "primary mouse button" and act accordingly.
	**New Features** or program enhancements are marked with this image. These features were added to ResSim with the version number identified on the cover of this manual.
	This storm cloud icon is used to mark a disabled or malfunctioning feature. Like rain in California, this is an unusual occurrence so you won't see a lot of these in this manual.

Tip

Tips are added where we wanted to make an instruction or description standout. Sometimes, a Tip will provide a "how-to" for a "hidden feature" that we wanted to share where it seemed most logical. For example, see the Tip in the section on creating a New Simulation.

Warnings are used to tell you to be careful, or to watch out for something, or to just not do something.

Notes are used to share information that you should take special note of. They are intended to emphasize a point or to tell you something beyond the how-tos of the ResSim user interface.

Chapter 2
The ResSim User Interface

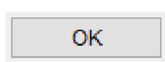
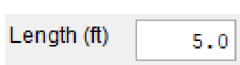
Before you begin working with ResSim, you should understand some ResSim concepts and user interface fundamentals to enable you to develop, access, and interact with watersheds in ResSim. Specifically, this chapter covers:

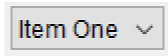
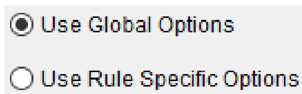
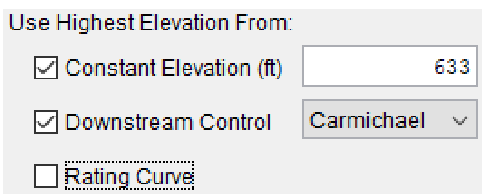
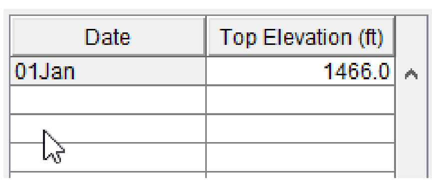
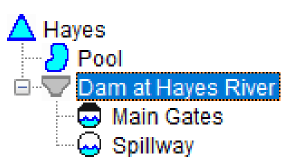
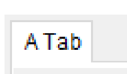
- The Widgets that make up the User Interface
- The Common ResSim Interface Components
- Layers and the Map Display Area

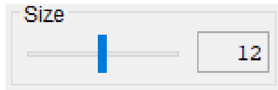
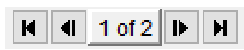
2.1 User Interface Components

By now, most computer users recognize the various components or *widgets* that make up a graphical user interface and how to interact with them. For those who are uncertain of the terminology or what to expect when using a graphical user interface, Table 2.1 lists the various widgets that are used in the ResSim user interface and describes how to interact with them.

Table 2.1 User Interface Widgets

Name	Description
Pointer 	The object that moves around on your monitor when you move your mouse.
Button 	By positioning your pointer on a button and pressing then releasing (a.k.a. "clicking") your left mouse button, the function indicated by the label or image on the button is executed.
Text Box 	Usually, a data entry field. Position your pointer in the box and click. A blinking vertical bar (cursor) will appear. Type in an appropriate value. Some text boxes are used to display information and are not editable.

Selection (or Drop-down) List 	A selection list looks like a text box with a down-pointing triangle or "V" in it. When you click in the text box, a list of options will appear below it. If the list of possible options is long, a scrollbar may appear to the right of the list. You can select one of the options by clicking on it in the list with your pointer. The list will disappear, and your selection will be displayed in the text box.
Radio Buttons 	A set of options that are mutually exclusive — pick one and only one. The buttons are open circles. The selected option appears with a black circle inside.
Check boxes 	A check box is used to identify an option that is or isn't activated, on, or selected. A check box is an open square. An "activated" checkbox has a checkmark inside the box. The checkmark is added or removed by simply positioning your pointer in the box and clicking.
Table, Function, Relationship 	A table is a set of columns and rows that allow input and/or output of related data. For data entry, almost all data entry tables must be entered in monotonically increasing order, especially for the independent variable — the first column.
Trees 	Trees are used to reflect a hierarchy or order of data objects or elements. The tree structure should be familiar to a Windows user since the same basic structure is used in the standard file explorer.
Tabs 	Tabs are used to organize editors/windows so that they can be less "busy" — to collect similar data into subpanels of the editor. Click on the tab to make that tab "current" or bring it "forward"; to view it.

Slider Bar 	Slider bars are used to allow quick selection of a value from an allowable range. The bar usually appears as a horizontal bar representing the range with a small vertical bar as the marker on the range. The selected value appears in a textbox to the side of the slider.
VCR Buttons 	These buttons are a set that look like the buttons on your VCR/DVR remote. They are typically used in the ResSim editors to move quickly through the various elements that can be edited by the editor. The four buttons shown from left to right are go to the first element, go to the previous element, go to the next element, and go to the last element. The box in the middle indicates which element of the full set is the current element shown in the editor.

2.2 ResSim's Modules

The design of the ResSim user interface separates the model development process into three major stages and provides the functionality and activities associated with each stage as *Modules*. These modules are **Watershed Setup**, **Reservoir Network**, and **Simulation**. Each module has a unique purpose, one or more unique datasets, and a set of functions accessible through menus, toolbars, and schematic elements for creating and editing that module's dataset(s). Figure 2.1 illustrates the basic modeling features and data available in each module.

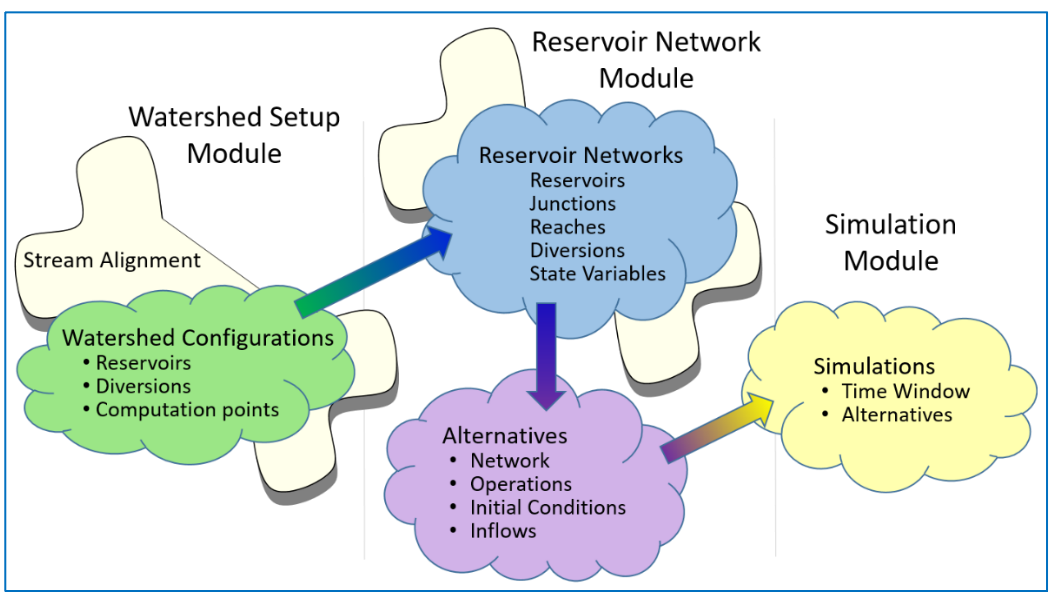


Figure 2.1 ResSim Module Concepts

2.2.1 The Watershed Setup Module

The purpose of the **Watershed Setup** module is to provide a common framework for defining the features of a watershed for modeling with one or more HEC modeling applications. Aspects of this module can be found in HEC-ResSim, HEC-FIA, HEC-WAT, and the CWMS CAVI.

Activities associated exclusively with the **Watershed Setup** module include creating new watersheds, adding background maps, establishing and editing a stream alignment, defining watershed configurations, assembling configuration elements to describe a watershed's physical features, and viewing lists of schematic elements that you have created.

Chapters 3 through 7 describe the **Watershed Setup** module and its activities in detail.

2.2.2 The Reservoir Network Module

The purpose of the **Reservoir Network** module is to isolate the development of the reservoir model and alternatives from the analysis of results. The activities associated with the **Reservoir Network** module include creating a reservoir network and schematic, describe the physical and operational features of your network, assembling the alternatives that you want to analyze, and viewing reports that list your network elements by type.

Chapters 8 through 13 describe the **Reservoir Network** module and its activities in detail.

2.2.3 The Simulation Module

The purpose of the **Simulation** module is to isolate the analysis of results from the model development process. Once you have created your reservoir model (network) and defined at least one alternative, you can use the **Simulation** module to create simulations, compute your alternatives, view output, and make changes to your model as appropriate to achieve your desired operations.

Chapter 15 describes the **Simulation** module and its activities in detail.

2.3 Recognizing Common Screen Components

Although the main ResSim interface has three faces or modules—each with its own functions and module-specific tools, menus, and output—the modules share common screen components, as illustrated in Figure 2.2. These common components include:

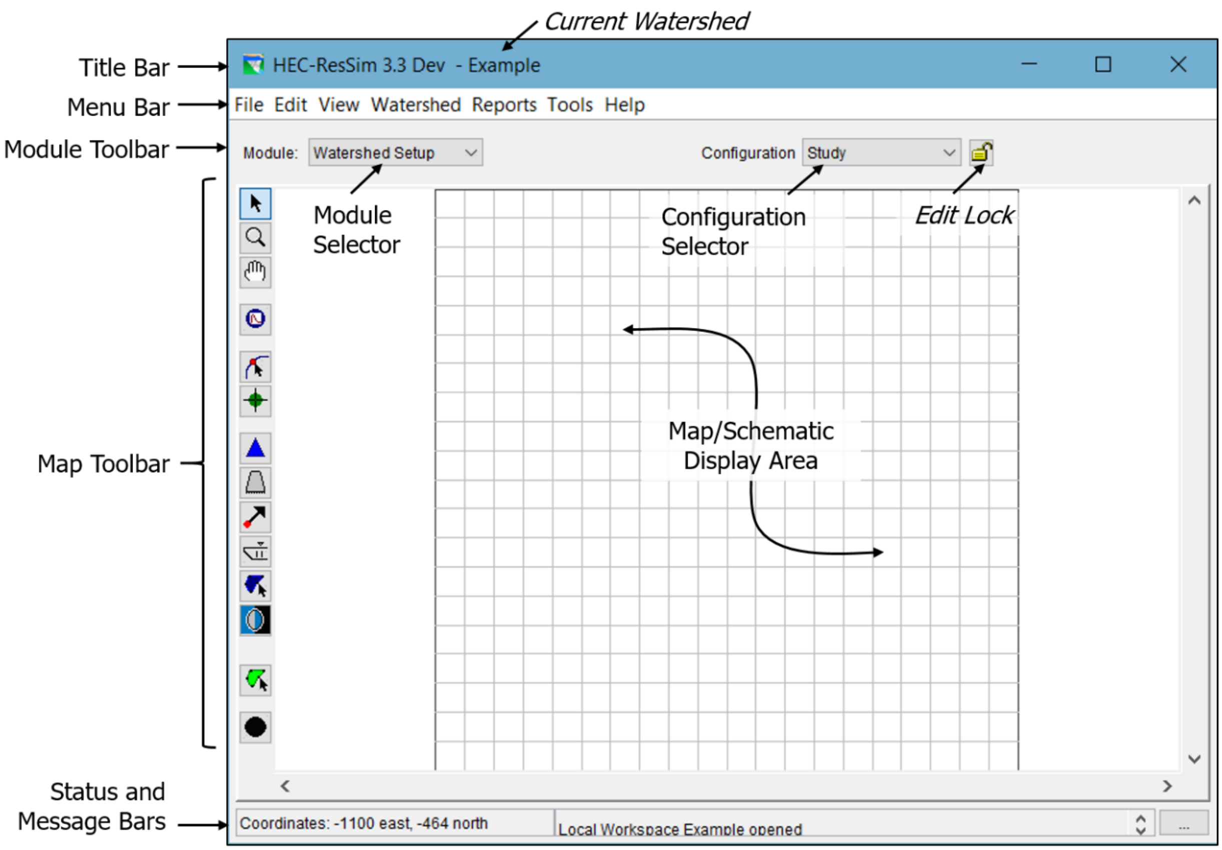


Figure 2.2 ResSim Main Window and its Common Components

The **Title Bar** displays the software name and version. After a watershed is opened, the name of the current watershed will also be displayed.

The **Menu Bar** contains menus for access to ResSim features and options. As you switch between modules, module-specific menus as well as the list of items in the common menus may change, offering module-specific commands. You can select a menu item by clicking on the name of the menu (such as File), then pointing to and clicking on the menu item you wish to select.

The **Module Toolbar** is the row of user interface objects (a.k.a. widgets) directly below the **Menu Bar**. It contains module management tools including:

The **Module Selector**—this widget gives you access to the three available modules of ResSim. Use this drop-down list to move between the **Watershed Setup** module, the **Reservoir Network** module, and the **Simulation** module.

The *Active* **Configuration** or **Network**—this widget displays the name of the currently open or "active" configuration or network. The active configuration or network is available for editing from the various editors in the module and its schematic is displayed in the Map Display area.

The **Edit Lock**—this button toggles the edit feature of the module on and off.

The **Map/Schematic Display Area** (a.k.a. map region) is the (geo-referenced) map space where all model schematic elements (stream alignment, computation points, reservoirs, etc.) and map layers are shown.

The **Map Toolbar** contains tools for interacting with the map display, manipulating schematic elements, editing element properties, and viewing results. The first three tools in the **Map Toolbar** and their functionality are common to all three modules. The other tools that appear in the Map Toolbar have module-specific functions and may appear in only one module.

The **Status Bar**, located at the below the Map Display displays "Ready" unless your cursor is somewhere in the Map Display area, then it shows the coordinates of the cursor as it relates to the current view in the Map Display area in the watershed's coordinate system.

The **Message Bar**, located to the right of the **Status Bar**, displays a scrolling list of messages from ResSim. These messages are usually produced by processes that may take a few moments (or more) to complete. The scroll arrows and the … button to the right of the Message Bar control the display and allow you to review all messages received during your ResSim session. Program diagnostic messages from ResSim are directed to the console log (not the message bar), which can be accessed from the **Tools** menu.

2.3.1 The Menu Bar

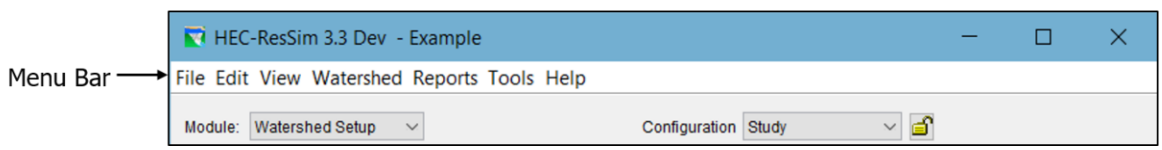


Figure 2.3 ResSim Menu Bar

The **Menu Bar** of the ResSim Main Window (Figure 2.3) contains several menus, most of which are common to all three ResSim modules. Although the menus are common, the options available in some of the menus may vary from one module to another. In addition, one or more menu options may be "greyed out" or disabled when they are not applicable for the current state of ResSim. For example, **Save Watershed** (a **File** menu option) will be disabled if there is no currently open watershed.

The most common and consistent menus in the **Menu Bar** are the **File**, **View**, **Tools**, and **Help** menus. These menus provide the same options or commands in all three ResSim modules. These menus may also contain module-specific options that appear only within individual modules. These menus and their options are described in detail below.

Two other menus, **Edit** and **Reports**, are also included in the **Menu Bar** of all three modules. These menus contain only module-specific options and are described in detail in later chapters of this manual.

2.3.1.1 The File Menu

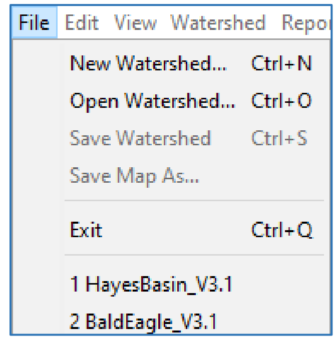


Figure 2.4 File Menu

The **File** menu (Figure 2.4) contains options specific to the ResSim watershed (project or model) that you want to work with. These options include:

New Watershed—creates a new ResSim watershed. This option is only available in the Watershed Setup module.

Open Watershed—opens an existing Watershed. See Section 2.4 for details on opening a watershed.

Save Watershed—saves the currently open watershed to disk including any changes you have made since opening it.

Save Map As—saves the current view of the Map Display Area to a file. You will be required to give the file a name and select a graphical file format (e.g., jpg, png, etc.).

Exit—prompts you to save any unsaved files and shuts down ResSim.

Last Five Watersheds—at the bottom of the **File** menu, your five most-recently—opened watersheds are listed. These entries are shortcuts for opening the identified watershed.

2.3.1.2 The Edit Menu

The **Edit** menu is different in each module of ResSim. It contains options for editing the components of the primary dataset that is the focus of the current module. For example, the Reservoir Network module's primary focus is the reservoir network, so the Edit menu provides access to the editors for the various components of a network.

2.3.1.3 The View Menu

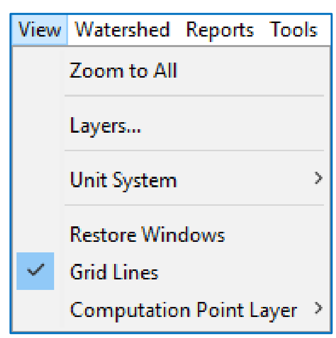


Figure 2.5 View Menu

The **View** menu (Figure 2.5) contains options related to the Map Display Area as well as to the entire ResSim user interface. The View options include:

Zoom to All—restores your map display to the full watershed extents,

Layers…—opens the **Layer Selector** dialog

Unit System—sets the data units (English or SI) you want displayed in the ResSim GUI.

Restore Windows—brings to the foreground any windows or dialogs that have become hidden or inactive.

Grid Lines—toggles on/off display of a grid in the map region.

Computation Point Layer—toggles on/off the available computation point layers. *This option is only available in the watershed setup module.*

2.3.1.4 The Tools Menu

The **Tools** menu (Figure 2.6) provides access to features that can be considered tools for assisting with model development. This menu also provides access to the program **Options** editor and the watershed **Information** viewer. The features accessible from this menu include:

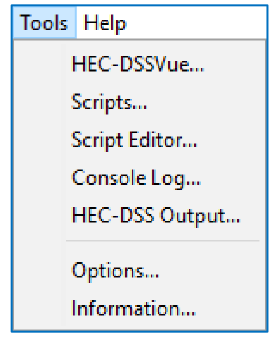


Figure 2.6 Tools Menu

HEC-DSSVue—is the primary software utility program of viewing and editing HEC-DSS data—e.g., time series data, rating curves, etc.

Scripts...—provides access to the list of utility scripts you have added to the watershed. You can launch them from here.

Scripts Editor...—opens the Scripts Editor which allows you to create, edit, import, delete, and run utility scripts.

Console Log...—displays the contents of the "HEC-ResSim.log" file which is stored by default in your AppData folder for ResSim. The path to this folder should look like: "C:\Users*yourUserName*\AppData\Roaming\HEC\HEC-ResSim\3.3\logs". The console log contains messages produced by ResSim during its current or most recent execution.

HEC-DSS Output...—displays the HEC-DSS log file, HEC-ResSim_DSS.log, which is also located in your AppData folder, The HEC-DSS log contains messages produced when DSS files are accessed by ResSim.

Options...—opens the ResSim Program **Options** Editor (Figure 2.7). The Options editor gives you access to a variety of program settings that affect how ResSim behaves. This editor is organized into tabs:

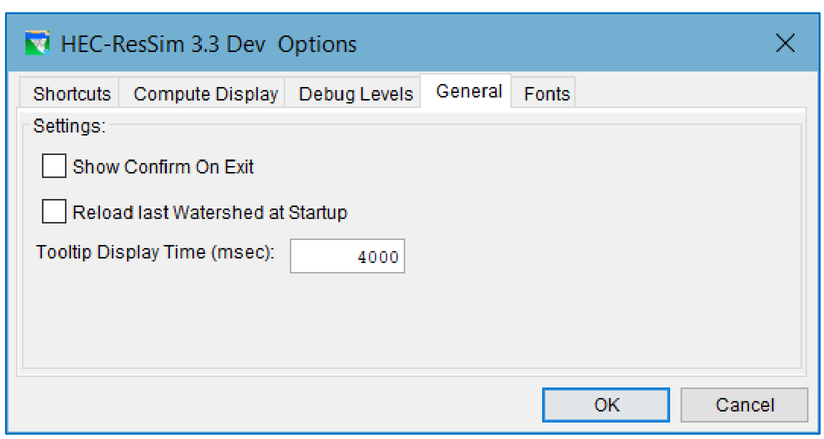


Figure 2.7 ResSim Options Editor

Shortcuts—lets you associate a name with a directory path where you will be storing your watersheds. Shortcuts were formerly called Watershed Locations.

Compute Display—contains settings that control the appearance of the Compute Window and the format of the compute, console, and DSS logs.

Debug Levels—allows you to control the quantity of various categories of debug message that are written to the logs.

General—provides options for a confirmation message to appear when you exit ResSim and to automatically open the watershed you were working on the last time you ran ResSim.

Fonts—lets you change the font used throughout the ResSim GUI.

Additional tabs and their associated program options may appear in the **Options** editor depending on the current module. The **Options** editor shown in Figure 2.7 is from the **Simulation** module and includes the **Simulation**, **Advanced**, and **ResSim Compute** tabs. The **Options** Editor is described in detail in Appendix A.

Information…—opens the **Application Properties** viewer (Figure 2.8) which displays the properties of various aspects of the current instance of ResSim. Like the **Options** editor, this viewer is also organized with tabs:

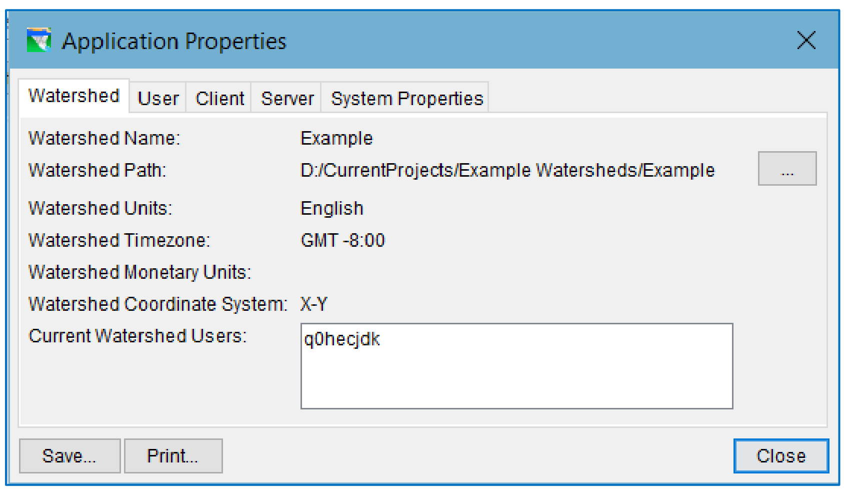


Figure 2.8 Application Properties Viewer

Watershed—displays details about your watershed

User—displays information about you (the ResSim user)

Client—displays information about ResSim itself (the Client)

Server—if ResSim is were launched in Client-Server mode, this tab displays information about the server application that ResSim is connected to. However, the Client-Server feature is disabled in the public release version of ResSim.

System Properties—displays the list of environment variables defined for the currently running instance of ResSim. This list includes variables defined for both the operating system and the Java Runtime Environment.

2.3.1.5 The Help Menu

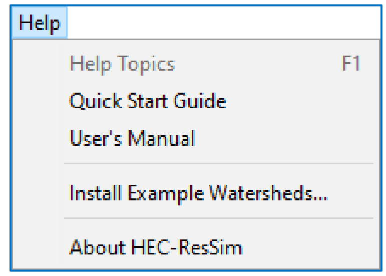


Figure 2.9 Help Menu

The **Help** menu (Figure 2.9) provides a few options to assist you in using ResSim. These options include:

Quick Start Guide—displays the ResSim Quick Start Guide.

User's Manual—displays the ResSim User's Manual (this document).

Install Example Watersheds—allows you to install a couple of watersheds that were used to create many of the screen shots in the Quick Start Guide and the User's Manual.

About HEC-ResSim—displays information about option the version of ResSim you are using.

2.3.2 The Module Toolbar

The **Module Toolbar** is located between the **Menu Bar** and the Map Display area of the main ResSim user interface (Figure 2.10). The content of the **Module Toolbar** varies depending on the current module, but for all modules, the common object in the tool bar is the **Module Selector**. Other objects on the toolbar that appear in more than one module include the current **Configuration** and/or **Network** name field and the **Edit Lock** button.

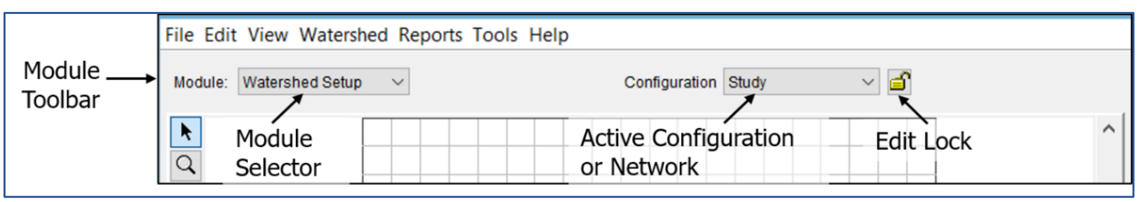


Figure 2.10 Module Toolbar

2.3.2.1 The Module Selector

The **Module Selector** is a selection list of the available ResSim modules. Use this drop-down list to move between the **Watershed Setup** module, the **Reservoir Network** module, and the **Simulation** module.

2.3.2.2 The *Active* Configuration or Network

The **Module Toolbar** in each module contains widget that shows the name of the currently open or "active" **Watershed Configuration** or **Reservoir Network.** The *active* configuration or network schematic is displayed in the **Map Display** area.

In the **Watershed Setup** module, the widget is a selection list from which you can select and/or change the active configuration. In the **Reservoir Network** and **Simulation** modules, the widget is a textbox that shows the name of the currently open (or active) network whose schematic is being displayed in the **Map Display** area.

2.3.2.3 The Edit Lock

The **Edit Lock** button is another somewhat common object on the Module Toolbar. The **Edit Lock** button appears in the **Watershed Setup** and **Reservoir Network** modules, just to the right of the current Configuration or Network name. This button toggles the edit feature of the module on and off.

If the lock button appears unlocked , the editing features of the module will be in "view only" mode. "View Only" mode means that some options in the main and context menus will be unavailable (greyed out) and the available editors will open in "view only" mode with most of their editable fields "greyed out" (non-editable).

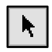
If the **Lock** button appears locked , all relevant editing features will be enabled.

> The concept of locking the module to allow editing is counter-intuitive to many users. However, if you understand where the lock originated, it may make more sense. You see, when ResSim was first designed and built, it was part of a multi-user, client/server system. In order to function properly in that multi-user environment, the ability to *lock out other users so that only you could edit the watershed* was an important feature.
>
> Unfortunately, now that the multi-user framework is no longer being actively used, the locks have become redundant. They will be removed in a future version of ResSim.

2.3.3 The Map Toolbar

Figure 2.11
Map Toolbar

The **Map Toolbar** (Figure 2.11) is located to the left of the Map Display area and contains a column of buttons. Each of these buttons changes the functionality (behavior) of your mouse cursor (tool) in the map display area, effectively changing your mouse into different map tools. Each tool button (a.k.a, tool) displays an icon to illustrate its purpose. The set of tools available in the **Map Toolbar** changes depending of the current module; however, the first three tools on the **Map Toolbar** and their functionality are common to all three modules. These tools are:

 **Pointer/Selector Tool**—use this tool to select schematic elements in the map display and to edit, rename, and delete them. To access the context (pop-up) menu of any schematic element, select the **Pointer/Selector Tool**, click the right mouse button (right-click) on an element, and a (module and element appropriate) list of functions will appear beside the element in the map space.

 **Zoom Tool**—use this tool to zoom your view of the map space in and out. To zoom in, hold the left mouse button down and drag a "box" to outline the area you want to zoom in on. To zoom out, right-click in the display area; the view will zoom out in intervals.

If your mouse has a wheel, the zoom tool is not required for zooming—just click in the Map Display area and roll the wheel up to zoom out and down to zoom in.

 **Pan Tool**—this tool can be used to move (pan) the display area when zoomed in. To pan, with the cursor in the map display area, hold down the left mouse button and drag the mouse in the direction you would like to pull the display area. Note: when zoomed in, scroll bars appear at the bottom and right edges of the map display. These scroll bars can also be used to pan the display without switching to the Pan tool.

2.4 Opening an Existing Watershed

Once you have launched ResSim, you will need to open a watershed. You can open an existing watershed from any module in ResSim, but if you need to create a new watershed you must be in the **Watershed Setup** module. See Chapter 3 for details on creating a *new* watershed. Three methods of opening an *existing* watershed are available:

- Use the **Open Watershed** browser
- Select one of the five most recently used watersheds from the **File** menu

- Reload the Last Watershed at startup.

2.4.1 The Open Watershed Browser

To access the **Open Watershed** browser:

- Select **Open Watershed** from the **File** menu (of any module). The **Open Watershed** browser will appear (Figure 2.12).

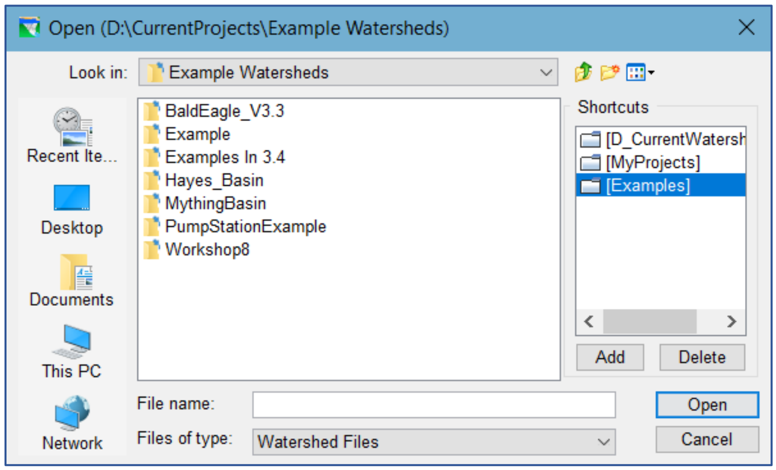


Figure 2.12 Open Watershed Browser

- Navigate (browse) to and select your watershed's .wksp file. The *.wksp* file is the ResSim "project" file. This file is located in your watershed's root or main folder and its filename is the name of your watershed with an extension of .wksp. The .wksp file identifies the watershed to ResSim.
- Click the **Open** button. The **Open Watershed** browser will close and the watershed you selected will load and appear in ResSim's main window with the watershed's name shown in the **Title Bar**.

If you intend to store several watersheds in the same folder, you can create a **Shortcut** to that folder so that you can navigate to it quickly in the **Open Watershed** browser. To create a shortcut:

- From the **Open Watershed** browser, navigate to the folder containing your watershed(s), then press the **Add** button. A dialog will appear asking you to "enter an alias" for the current folder. After you give the shortcut (or watershed location) a name and press **OK**, your shortcut will appear in the **Shortcuts** panel at the right of the **Open Watershed** browser.

Note: You can *also* create and edit Shortcuts in the Options editor's Shortcuts tab—see Appendix A for details.

2.4.2 Opening a Recent Watershed

ResSim keeps track of the last five watersheds you have opened and lists them at the bottom of the **File** Menu. If the watershed you want to open is one of those five, you do not need to access the **Open Watershed** browser to open the file, you can simply select the watershed directly from the **File** Menu (Figure 2.13). If you are unsure if the watershed you want is one of the five, hover on its name in the **File** Menu—the folder location of the watershed will appear in a tooltip box.

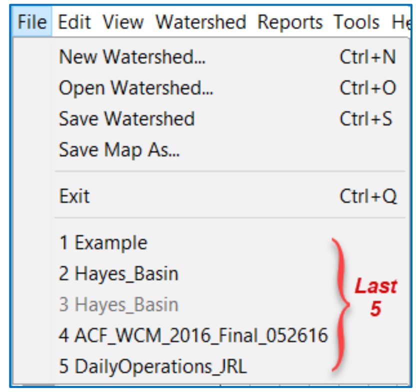


Figure 2.13 File Menu—Last Five Watersheds

2.4.3 Reload Last Watershed at Startup

If you are going to be working on the same watershed for a while, you might like the option to **Reload** the **last watershed** you had open each time you startup ResSim. To enable this option:

- Select **Options** from the **Tools** menu. The **Options** Editor will open (Figure 2.14).
- Select the **General** tab
- Click in the checkbox next to **Reload last Watershed at Startup**.
- After the checkmark appears in the checkbox, click **OK** to save your setting and close the **Options** editor.

2.5 Understanding the Two Schematic Types

The **Map/Schematic Display Area** of ResSim was designed to enable you to visualize river and reservoir systems (watersheds) in a map-based, geo-referenced context through a series of map layers. Some map layers are created outside of ResSim and can be imported and used as background maps. Other layers are created inside ResSim and represent different portions of your river and reservoir (model) schematic.

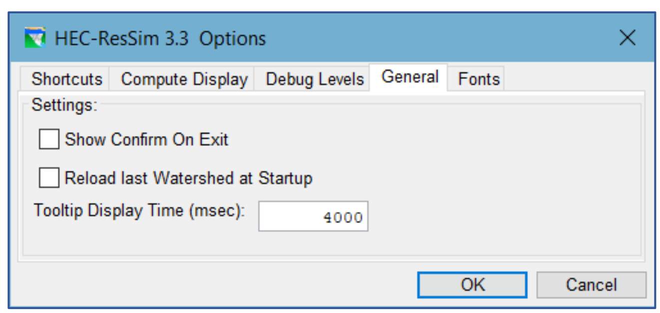


Figure 2.14 Options Editor—Reload Last Watershed

Each ResSim module displays a somewhat different schematic in the **Map/Schematic Display Area**:

In the **Watershed Setup** module (Chapters 3 through 7) you will create ***watershed schematics*** that can be shared across models that support the watershed schematic elements and features.

Tip

Since ResSim can open only one watershed at a time, to work on two watersheds at the same time, launch a second instance of ResSim.

Note: It is NOT recommended that you open the same watershed in both instances of ResSim. Be aware that both instances of ResSim will write to the same console and DSS log files. As a result, the content of these logs may become quite unintelligible.

In the **Reservoir Network** module (Chapters 8 through 13) you will create one or more ***model schematics*** (networks) that symbolize the river and reservoir system(s) you are modeling.

In the **Simulation** module (Chapter 14), the ***model schematic*** for the network of your *active alternative* will be displayed. And, by right-clicking on a schematic element in the Map Display, a context menu will appear giving you options to quickly plot and tabulate results.

2.5.1 The *Shared* Watershed Schematic

The watershed schematic and its components were designed to be shared (implemented) across the various HEC models and the model integrating software packages HEC-WAT and CWMS. In ResSim, as well as HEC-WAT and CWMS, the shared watershed schematic embodies a set of schematic components that a sequence of models are intended to represent.

The components that make up a watershed schematic include the stream alignment, computation points, projects, and impact areas. These components are created in the **Watershed Setup** module of ResSim and combined represent the watershed schematic.

Projects are collected into sets called configurations. Configurations are *unique* sets of projects that when combined with the *common* stream alignment, computation points, and impact areas defining unique watershed schematics; thus *configuration* is often used as a synonym for *watershed schematic* since the configuration is what makes each schematic unique.

For example, a watershed modeling team may create two watershed schematics (configurations): an "existing conditions" configuration that includes all the reservoirs, diversions, gages, levees, and other points that currently exist in the watershed and a companion "without levees" configuration that includes all the same schematic elements except for the levees. The expectation of the modeling team that developed the two configurations is that a different set of individual models will be needed to reflect each configuration. In reality, some of the models may be appropriate to both configurations while other models may only be appropriate to one configuration or the other.

ResSim, HEC-WAT, and CWMS have all adopted the shared schematic concepts and implemented the associated **Watershed Setup** module. Other HEC models have not included the **Watershed Setup** module in their design but have reflected selected shared watershed schematic components in their native model schematics. The stream alignment and the computation points are the most widely represented shared watershed schematic components.

2.5.1.1 The Stream Alignment

The **Stream Alignment** represents the river and stream network of a watershed (Figure 2.15). It appears in every ResSim module, but can only be created and edited in the **Watershed Setup** module. **Streams**, **Stream Nodes**, and **Stream Junctions** are the elements of a **Stream Alignment**. See Chapter 4 for detailed information on creating and editing the **Stream Alignment**.

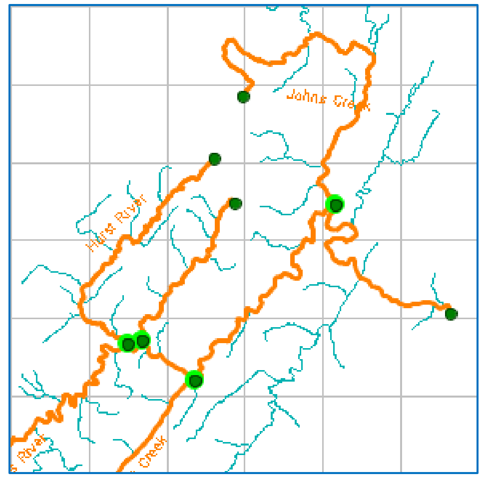


Figure 2.15 Stream Alignment

The Stream Alignment is *the* foundational element of the watershed schematic; all other schematic elements have some relationship to the stream alignment; either they are positioned at a specific station along a stream or they are located relative to a stream.

2.5.1.2 Computation Points

Computation Points (Figure 2.16) are point features within the shared schematic where data is exchanged between models (e.g., between ResSim and HEC-HMS). These points are placed on the **Stream Alignment** and usually represent gages, confluences, or model element boundaries (e.g., the inflow and outflow points of a reservoir). ResSim represents **Computation Points** with **Junction** elements in the *model schematic* of a **Reservoir Network**.

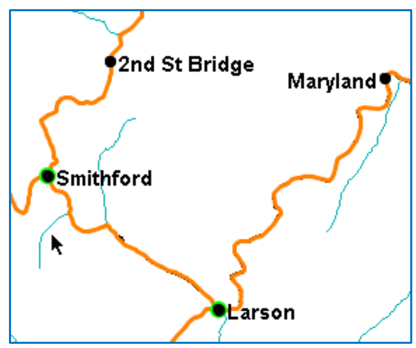


Figure 2.16 Computation Points

2.5.1.1 Project Elements

Projects are man-made features in the watershed that affect flow in the river system. Project elements that can be placed in the watershed schematic include **Reservoirs**, **Levees**, **Diversions**, **Channel Modifications**, **Off-Channel Storage**, and **Other Projects** like pump stations and in-line spillways (Figure 2.17).

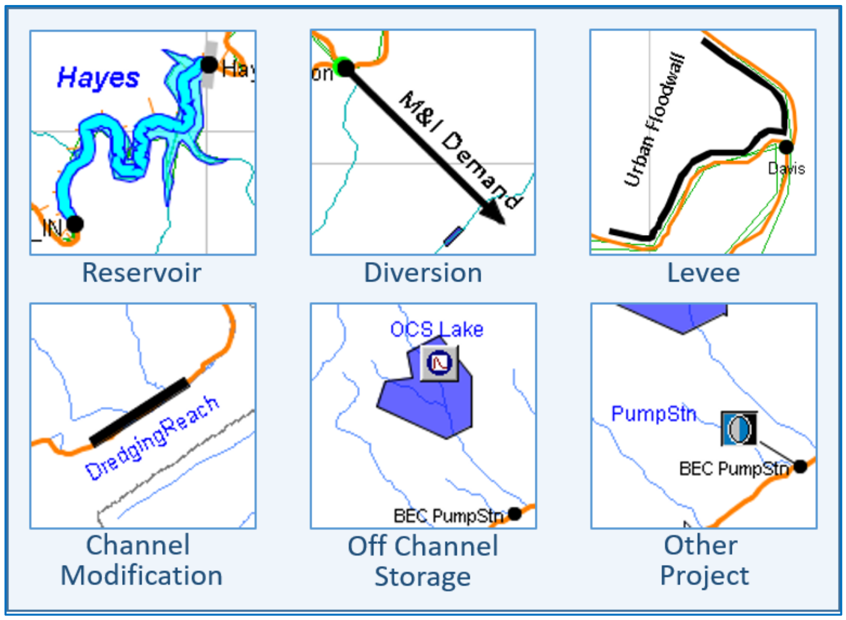


Figure 2.17 Project Elements

ResSim can directly represent reservoirs and diversions in its model schematic with reservoir and diversion model elements, but there is no direct parallel for any other project element as a ResSim model schematic element. However, with some creativity, you may be able to represent the other project elements using one or more of the available model elements. For example, a channel modification could be represented with a routing reach with specific routing parameters that reflect the smoother, faster reach.

2.5.1.2 Impact Areas

Impact Areas (Figure 2.18) are watershed schematic elements that represent distinct portions of a watershed affected by rising water where flood damages may be evaluated. Impact areas can be included in your ResSim watershed for consistency with your HEC-FIA model; however, since ResSim cannot directly represent impact areas as model elements, only the computation points corresponding to the index locations for the impact areas can be represented in your ResSim model.

For more information about Impact Areas, refer to the *HEC-FIA User's Manual* (HEC, 2003).

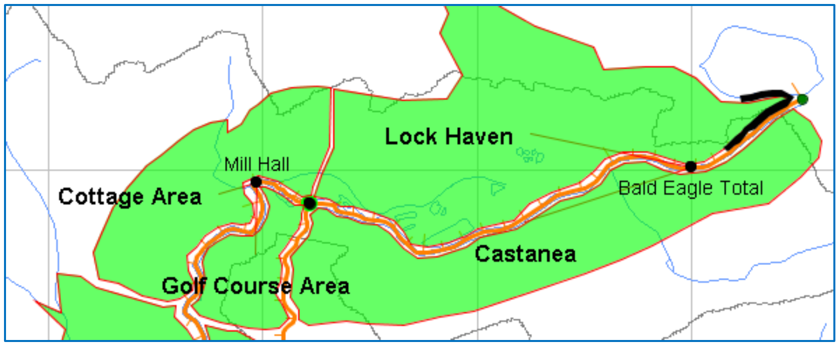
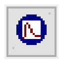


Figure 2.18 Impact Areas

2.5.1.3 Time Series Icons

Time Series Icons  may be placed in the map region to provide access to time series data. You can use Time-Series Icons to identify and visualize any external data that you associate with the icon (e.g., HEC-DSS data). Time Series Icons are included in the Watershed Setup module to facilitate their use in CWMS and RTS.

Note: Although Time Series Icons can be created in the **Watershed Setup** module of ResSim, they are not accessible from the other ResSim modules so are rarely, if ever, used in ResSim. For more information about Time-Series Icons, refer to the *CWMS User's Manual* (HEC, 2010).

2.5.2 The Model Schematic

The ResSim Model Schematic (Figure 2.19), which you will create in the **Reservoir Network** module (based on a shared watershed schematic (configuration) created in the **Watershed Setup** module), is the visual representation of your reservoir network.

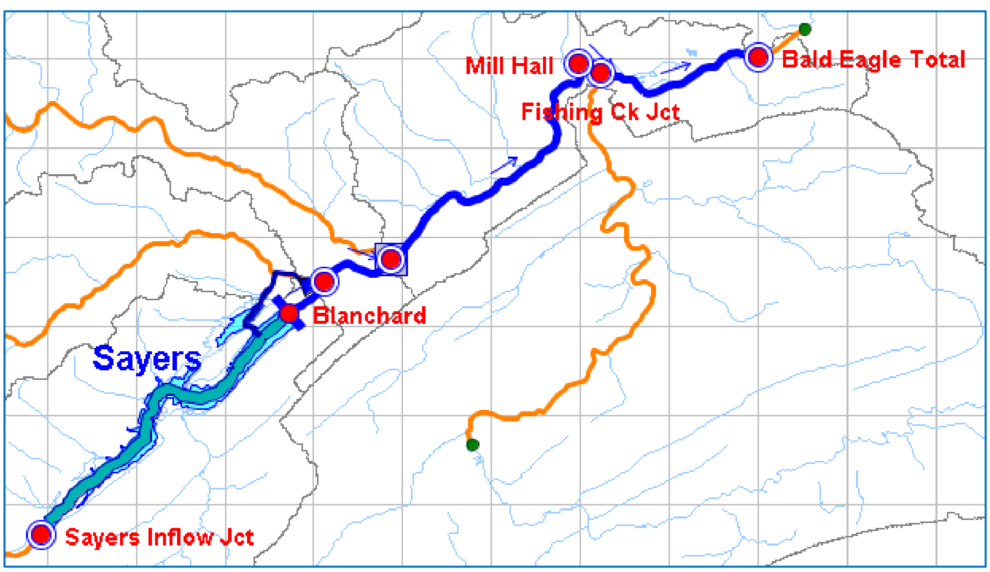


Figure 2.19 Model Schematic

The ResSim model schematic elements include reservoirs, reaches, diversions, diverted outlets, and junctions. When you create a network, reservoirs, diversion, and junctions are automatically added to your network for each reservoir, diversion, and junction element in the selected watershed configuration (schematic). You will complete the model schematic by adding routing reaches, diverted outlets, and diversions to connect the model elements. Each element of the model schematic is located *on* the stream alignment.

Through context menus, the individual elements of the model schematic allow you to access data editors to specify properties of your reservoir network elements and, from the **Simulation** module, to plot element-specific results. The ability to select schematic elements and access element-specific context menus is why we call the representation of your model in the map display area an "active schematic".

Although the stream alignment is visible in all modules, the model schematic is visible in the **Reservoir Network** module only when a Network has been opened and in the **Simulation** module only when an Alternative is **Set as Active**. See Chapters 7-14 for more details of reservoir networks, alternatives, and simulations.

2.6 Context Menus

A context menu, also commonly called a pop-up or shortcut menu, is a menu that appears when you right-click on a schematic element in the map display or on many other ResSim GUI components such as plots, tables, and tree nodes. Context menus offer a variety of context-specific commands and options that allow you to access data editors, plot and tabulate data, view reports, compute simulations, and more. Specific context menus and their options are described throughout this manual.

Tip Lots of good stuff may be hiding in a context menu.
So, when in doubt… right-click!

2.7 Tooltips

Tooltips are small text boxes that appear when you position (and hover) your cursor over an icon, label, button, map element, tree node, or other widget for which a tooltip has been defined. The text that is displayed will relate to the object you hovered over. Although it is usually informative, the text may be instructional. But, read quickly, most tooltips don't hang around long.

Chapter 3
Watershed Setup and Configurations

The purpose of the **Watershed Setup Module** is to provide a common framework for defining the features of a watershed for modeling with one or more HEC modeling applications, i.e., to setup a *shared schematic*. Aspects of this module can be found in HEC-ResSim, HEC-FIA, HEC-WAT, and the CWMS CAVI.

In the **Watershed Setup Module**, you will create new watersheds, set up your watershed's map display, and specify the *physical* arrangement of schematic elements that make up your basin. The set of schematic elements includes:

- The **Stream Alignment** is a representation of the stream centerlines of the river network in your watershed and is the backbone or framework for your schematics.
- **Computation Points** are locations where time-series information is exchanged between models in an integrated framework. Most **Computation Points** are placed on the stream alignment and represent gage locations, control points, or other key physically or operationally important locations.
- **Projects** are, typically man-made, structures that impact the flow of water in a river system. The types of projects that can be included in your schematics include **Reservoirs**, **Levees**, **Diversions**, **Channel Modifications**, **Off-Channel Storage Areas**, and ***Other* Projects**.
- **Impact Areas** are geographic regions that may experience economic damages due to high water levels (flooding).

You will also group the projects in your watershed into one or more **Configurations.** **Configurations** represent a current or potential arrangement of the watershed schematic elements (e.g., Existing Condition or Without Project).

This chapter will cover the **Watershed Setup Module** itself and the user interface features that are unique to this module. It will also cover how to create a watershed, add maps to the watershed, and define watershed configurations. Chapters 4 and 5 will cover creating and editing the stream alignment and drawing schematic elements.

3.1 The Watershed Setup Module

The **Watershed Setup Module** provides tools for setting up your watershed's *physical* arrangement, including maps, coordinate system, stream alignment, schematic elements, geo-extents, and other geo-referenced data. Figure 3.1 shows the components of the **Watershed Setup Module**'s main window. User interface components and options that are unique to this module are described in this section.

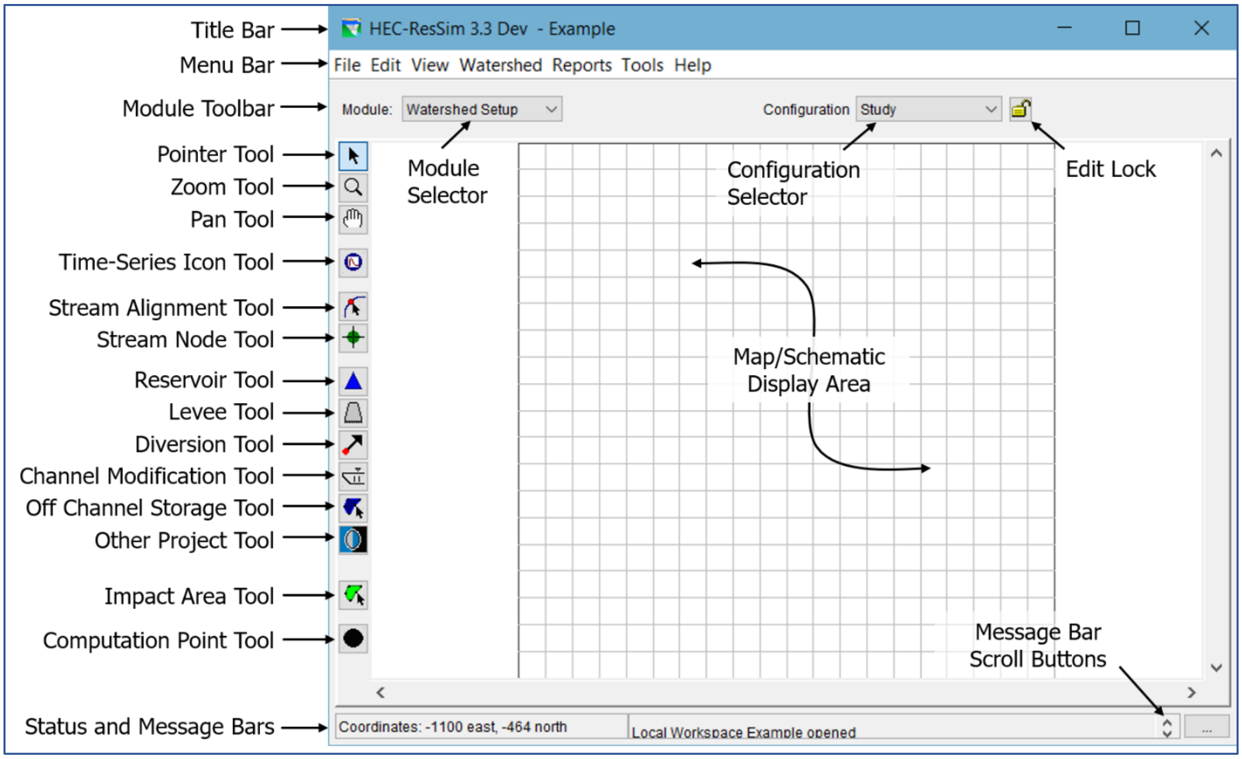


Figure 3.1 Watershed Setup Module—Main Window

3.1.1 The Menu Bar

The following is an overview of the menus unique to the **Watershed Setup Module**. The options in these menus will be described in more detail in the context of specific tasks later in this and later chapters. Refer to Chapter 2 for information about the **File**, **View**, **Tools,** and **Help** menus, which are common to all three modules.

3.1.1.1 The Edit Menu

Use the **Edit** menu (Figure 3.2) to access the following editors:

- Watershed Properties Editor,
- Impact Areas Editor,
- **Projects** Editor, and
- Drawing Properties Editor.

You must select **Allow Editing** if you wish to make changes to your watershed (or toggle the **Lock** above the Map Display area).

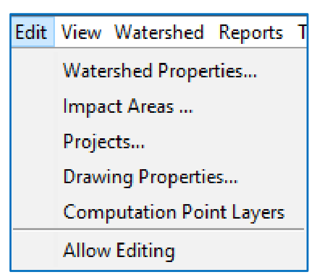


Figure 3.2 Edit Menu

3.1.1.2 The Watershed Menu

Using the **Watershed** menu (Figure 3.3), you can:

- Create and edit **Configurations**
- Update Computation Points
- **Import** and **Export** Stream Alignments
- Save Configurations.

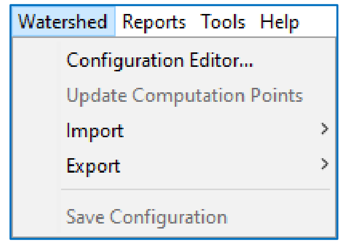


Figure 3.3 Watershed Menu

3.1.1.3 The Reports Menu

The **Reports** menu (Figure 3.4) gives you access to reports for your watershed. Reports include the **List of Streams**, **List of Impact Areas**, **List of All Configurations**, **List of Computation Points**, and **Projects** listings by project type.

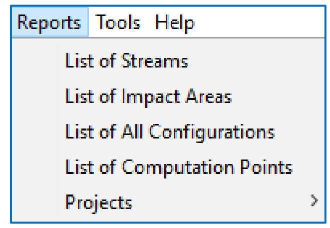


Figure 3.4 Reports Menu

3.1.2 The Configuration Selector

The map display can only display one watershed schematic at a time. That schematic is contained in a *watershed configuration.* You can use the **Configuration** Selector (drop-down box) to select the configuration you want displayed in the Map Display area. Once you select a Configuration, all project elements you add to the schematic will be included in that Configuration. The default configuration, **Study**, contains the entire set of all projects defined in the watershed, whether or not it is the active configuration.

3.1.3 The Edit Lock

Before you can add, remove, or edit schematic elements in your watershed, you must *lock* the configuration for editing. The easiest way to lock the configuration is by using the **Edit Lock** button, located to the right of the Configuration Selector. A less obvious way is by selecting **Allow Editing** from the **Edit** menu. Figure 3.5 illustrates the states of the Edit Lock. Remember, the **Lock** button must appear *Locked* in order for you to edit your configuration.

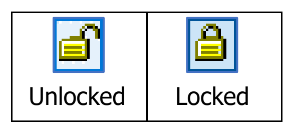


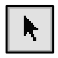
Figure 3.5 Edit Lock

The concept of locking the configuration to allow editing is counter-intuitive to most users. However, if you understand where the lock originated, it may make more sense. You see, when ResSim was first designed and built, it was part of a multi-user, client/server system. In order to function properly in that multi-user environment, the ability to *lock out other users so that only you could edit the watershed* was an important feature. Now that the multi-user framework is no longer being actively used, the lock has become redundant. It will be removed in a future version of ResSim.

3.1.4 Map Tools

The Tool Bar is located to the left of the map display area and contains a column of buttons. Each of these buttons changes the functionality (behavior) of your mouse cursor in the map display area, effectively changing your mouse into different map tools. Each tool button (a.k.a, tool) displays an icon to illustrate its purpose.

The first three tools on the Tool Bar and their functionality are common to all three modules. These Common tools are:

 Pointer/Selector Tool—use the pointer tool to select any schematic element in the map display. Left click will select the element your cursor is "pointing at". Right-click will open a context menu for the element you are pointing at.

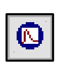
 Zoom Tool—use the Zoom Tool to zoom the map display area in and out. To zoom in, hold the left mouse button down to draw a box around the area you want to zoom in on. To zoom out, right-click in the display area; the display will zoom out in program-defined increments.

 Pan Tool—use the Pan Tool to move or scroll the display area while you are zoomed in.

In the **Watershed Setup Module**, the map tools that appear below the three Common tools are the Schematic Element tools. These tools allow you to create, edit, and manage the schematic objects in your watershed configuration(s). These tools also act as Pointer/Selector tools, but they will only interact with objects of their specific type. The schematic element tools are separated into five groups: the Time-Series Icon tool, the Stream Alignment tools, the Project tools, the Impact Area tool, and the Computation Point tool.

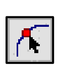
The Time-Series Icon Tool

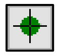
Time-series Icons are used for plotting and tabulating time-series data. This tool is provided for compatibility with CWMS; however time-series icons have very little use in ResSim itself. Creating and editing time-series icons is described in Chapter 6.

 Time-Series Icon Tool—use the Time-Series Icon tool to create and edit time-series icons.

The Stream Alignment Tools

The stream alignment represents the river system of your watershed. The tools for creating and managing the stream alignment include the Stream Alignment tool and the Stream Node tool. Working with the stream alignment is described in detail in Chapter 5.

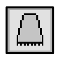
 Stream Alignment Tool—use the Stream Alignment Tool to draw and edit the multi-segmented lines that define a stream in the Stream Alignment.

 **Stream Node Tool**—the Stream Node Tool allows you to create and edit stream nodes and stream junctions (confluences), two of the primary components of a Stream Alignment.

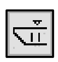
The Project Tools

The project tools include the Reservoir Tool, Levee Tool, Diversion Tool, Channel Modification Tool, Off Channel Storage Tool, and the Other Project Tool. Creating and editing the project elements is described in detail in Chapter 6.

 **Reservoir Tool**—use the Reservoir Tool to create and edit reservoir projects in the map display. The Reservoir Tool is the first of the Project tools in the toolbar. Creating and editing project schematic elements are described in detail in Chapter 6.

 **Levee Tool**—use the Levee Tool to create and edit levee projects in the map display.

 **Diversion Tool**—use the Diversion Tool to create and edit diversions in the map display.

 **Channel Modification Tool**—use the Channel Modification Tool to select and add channel modifications (such as channel straightening, concrete walls, dredging, and widening) to the map display and access editors for Diversions.

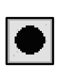
 **Off Channel Storage Tool**—use the Off-Channel Storage Areas Tool to select and create Off-Channel Storage Area polygons in the map display and edit their properties.

 **Other Project Tool**—use the Other Projects Tool to place an icon that represents a project that cannot be classified as one of the defined project types, such as pump stations and gages, etc.

The Impact Area and Computation Point Tools

Creating and editing impact areas and computation points are described in detail in Chapter 6.

 **Impact Area Tool**—use the Impact Area Tool to create and edit impact areas representing regions in the watershed where flood damages will be evaluated.

 **Computation Point Tool**—use the Computation Point Tool to create and edit Computation Points. Computation Points are locations in the watershed where individual models will share information. Typically, one model will produce results at a Computation Point that another model will use as input.

3.1.5 Using Context Menus

Context menus are accessed by right-clicking on a schematic element in the map area. The options in a context menu vary according to the schematic elements with which they are associated. Figure 3.6 shows the context menu that appears if you right-click on a reservoir element in the Map Display area in the **Watershed Setup Module.**

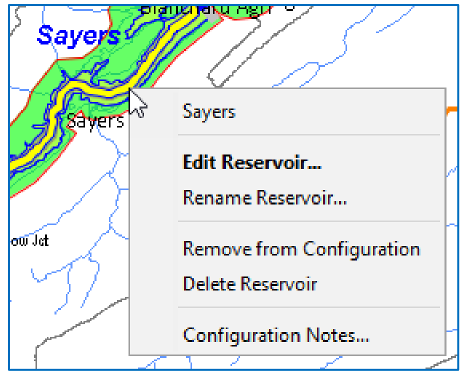


Figure 3.6 Reservoir Context Menu

If your watershed is locked, the context menu will provide options to **Edit**, **Rename**, or **Delete** the current element, **Remove** the current element **from** the current **Configuration**, and edit the **Configuration Notes** about the current element.

If your watershed is unlocked, the configuration is in "view only" mode so some of the context menu options will be unavailable (greyed out) and the available editors will open in "view only" mode.

3.2 Creating a Watershed

When you create a new watershed, you will need to provide a few details about it including:

- a name and description
- its directory location (on disk)
- the units of measure of the data and computations
- the Time Zone in which it is located

Once you have created your new watershed, ResSim will generate a new folder in the directory you specified; the folder name will be the name you gave your watershed. Your new watershed will then become the open watershed in ResSim, and the tools needed to create watershed data will become available in the ResSim user interface. After creating the watershed, you will proceed with other watershed setup tasks such as adding background maps, setting preferences for the map display, etc., as described later in this chapter.

3.2.1 Defining a Watershed Location

A **Watershed Location** is the place on disk where you store your watersheds. You can have multiple watershed locations, although an individual watershed cannot span multiple locations. Each watershed location is given an alias (or logical name) such as *My Watersheds* or *Current Projects*.

In prior versions of ResSim, you were required to create at least one watershed location (shortcut) before you could create a watershed. In this version, that requirement has been removed; however there are still features in ResSim that expect your watershed to reside in a known watershed location so we recommend that you continue to create and use specified watershed locations for storing your watersheds.

So, before you create or open a watershed, you should define at least one watershed location. To define a *watershed location*, a.k.a., a **Shortcut**:

- From the **Tools** menu, select **Options…**. The ResSim **Options** editor (Figure 3.7) will open.

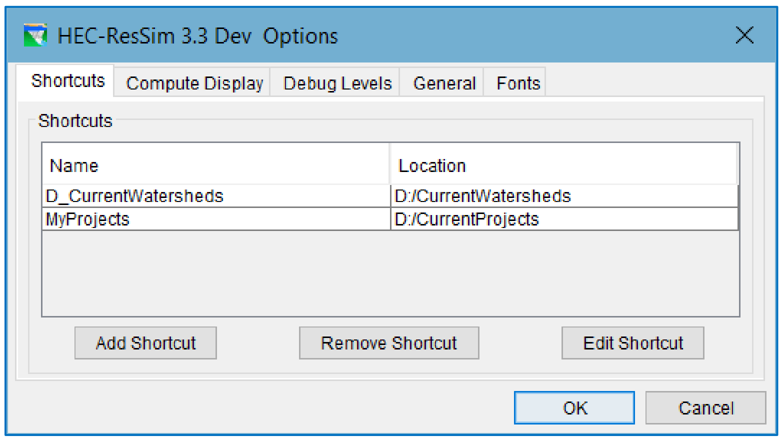


Figure 3.7 ResSim Options Dialog—Shortcuts

- Select the **Shortcuts** tab. The **Name** and **Location** of any watershed locations you have previously defined will display in the **Shortcuts** list.
- To add a new location to the list, click the **Add Shortcut** button. The **Add Shortcut** dialog will open (Figure 3.8).

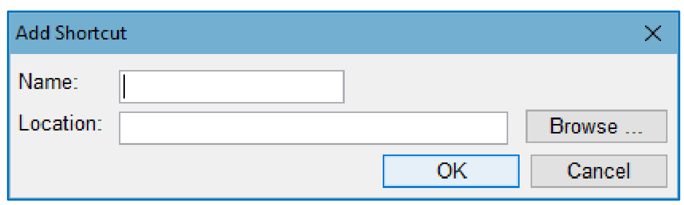


Figure 3.8 Add Shortcut Dialog

- Enter a **Name** (or Alias) for the new location/shortcut, then click **Browse…**. A file browser dialog will open (Figure 3.9). Use it to select the folder on disk where you want to store your watershed(s). Click **OK** to close the file browser and accept the selected folder as your shortcut's location.
- Click **OK** to finish creating your shortcut and close the **Add Shortcut** dialog. The new location/shortcut will now appear in the list of **Shortcuts** on the **Shortcuts** tab. Refer to Appendix A for information about other options available in the **Options** dialog.

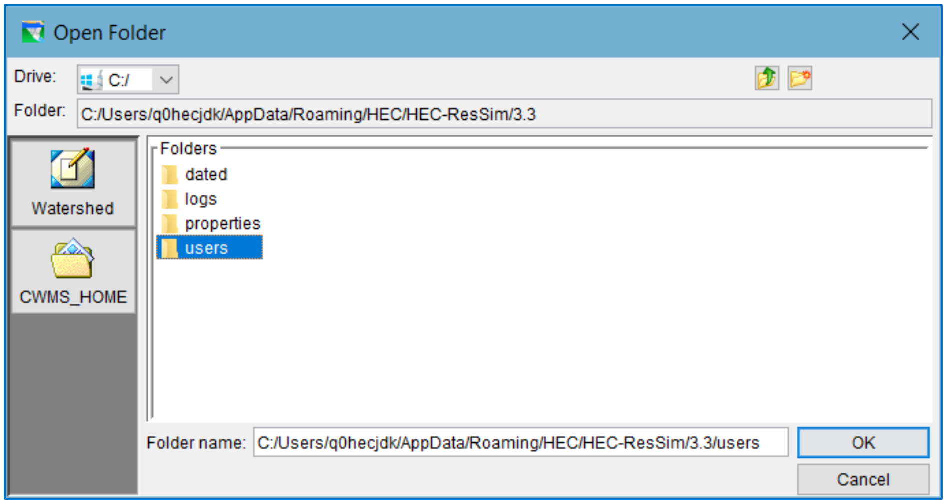


Figure 3.9 ResSim File Browser

3.2.2 Creating a New Watershed

Once you have established a watershed location, you can create a new watershed.

To create a new watershed:

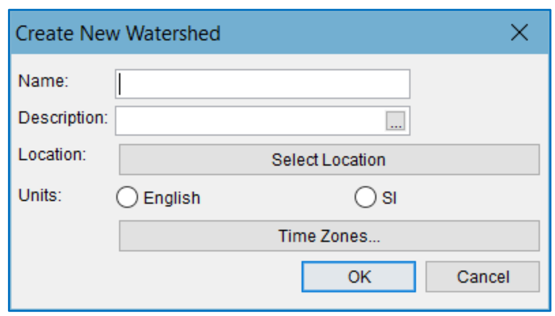


Figure 3.10 Create New Watershed Dialog

- Select **New Watershed** from the **File** menu. The **Create New Watershed** dialog (Figure 3.10) will open.
- Enter a **Name** for the new watershed. The watershed name must be less than 32 characters. Because the Name will be used for both a directory and a filename in your file system, the watershed name cannot contain any of the following characters:

  | / : * ? ” < > - \ .

- Enter a **Description** for the new watershed.
- Select a **Location** (a working directory) for the new watershed's directory from the file browser. A new folder representing your watershed will be created in the **Location** you select. The new folder will be given the **Name** of your watershed.
- Select the watershed's **Units**—your options are **English** (U.S. customary units) or **SI** (System International). See Section 3.2.2.1 *Units of Measure* below, for more information.
- Set the watershed's **Time Zone** by selecting from the list of standard time zones. See Section 3.2.2.2 *Time Zone* below, for more information.
- Click **OK** to create the watershed and close the **Create New Watershed** dialog.

- A **Watershed Summary** confirmation dialog (Figure 3.11) will open showing your selections. Review the settings carefully, then select:

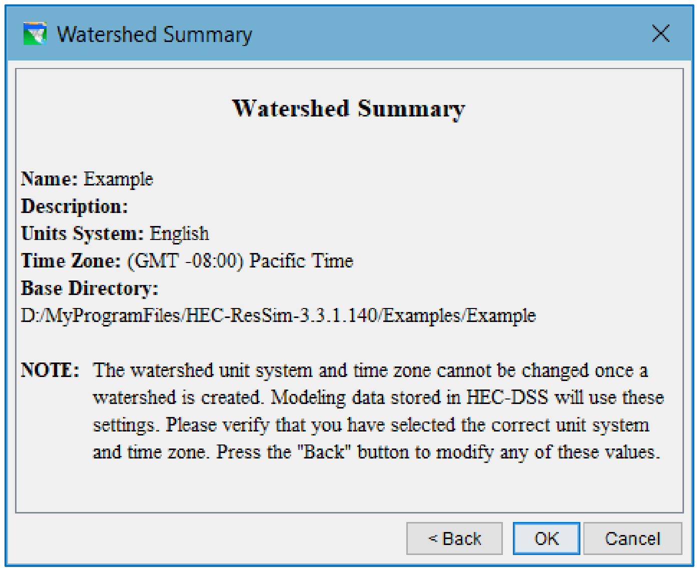


Figure 3.11 Watershed Summary Dialog

OK—if the settings are all correct and you want to proceed with creating the watershed.

< Back—If you want to change any of the settings.

> Be Careful…
>
> You cannot change the watershed's Name, Units, or Time Zone after the watershed has been created.

Cancel—if you do not want to create the new watershed.

3.2.2.1 Units of Measure

The watershed's **Units** is the unit system in which the watershed will perform its calculations and store its data and results. The options are **English** and **SI**. **English** (or U.S. Customary) units include feet, acres, cfs, Fahrenheit, etc. **SI** (or Metric) units include meters, hectares, cms, Celsius, etc.

Be aware that you *cannot change* the watershed units once you have created the watershed. However, you *can* (at any time) change the *display units* using the **View** menu; so even if you select **SI** as the watershed units, you can set the display units to **English** which will allow you to enter data and view results in English units.

ResSim will check the units of all input time-series data and will convert the data, if necessary, to watershed units before beginning the computations. This means that input time-series data can be provided in English or SI units, or both.

3.2.2.2 Time Zone

When you create a new watershed, you must select a time zone for the watershed. This typically would be the time zone for the dam or for most of the watershed, although you may have reasons for selecting a different time zone.

All time zones in the list provided when you create the watershed are considered "standard" time zones. They each utilize a fixed time offset from GMT and do not adjust for Daylight Savings Time.

ResSim will use the watershed time zone to store all time-series data that it generates. It will also adjust any input time-series data with a specified time zone that is different from that of the watershed into "watershed time" before beginning the computations. If any input time-series data does not have a specified time zone, ResSim will assume that the data is in watershed time.

3.3 Adding Background Maps

Layers in ResSim are like transparencies laid one on top of the other. Static images of physical, political, or logic features of the watershed, such as rivers, subbasins, roads, state boundaries, etc., can be layered in the display as color pictures. Each of these images, along with its associated data, is called a map layer and is added to the watershed and managed by the **Layer Selector**. Access to the **Layer Selector** is available from the **View** menu. In addition to the sections below, you can refer to Appendix B for more information on managing Map Layers.

Several map file formats are supported in ResSim including:

- ArcView® Shapefiles
- AutoCAD® DXF files
- ArcInfo® DEM files
- USGS Digital Line Graphs (DLG) files
- USGS Digital Elevation Model (DEM) files
- ASCII NET TIN files
- Raster images

3.3.1 Adding a Map Layer

Before adding a map layer to your watershed, we recommend that you copy the map files into the maps folder of the watershed. Also, you should verify that all maps you plan to add to your watershed share the same coordinate system.

To add a map layer to your watershed:

- Select **Layers** from the View menu. The **Layer Selector** dialog will appear (Figure 3.12).
- From the **Edit** menu, select **Allow Layer Editing**. This option is a toggle switch that, when set, will allow you to add and remove map layers, rearrange their drawing order, and modify the map display properties.

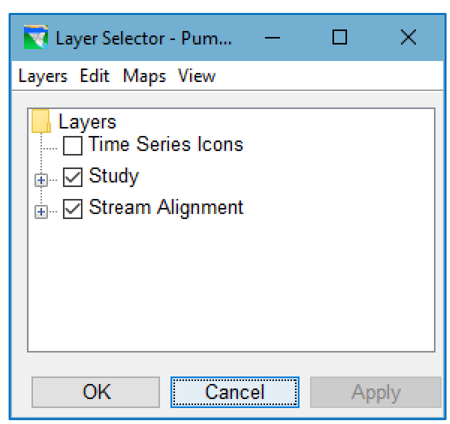


Figure 3.12 Layer Selector Dialog

- From the **Maps** menu, select **Add Map Layer**. The **Open File** dialog (Figure 3.13) will open.
- Locate the map file you wish to use on your local computer (for example, "np_shed.shp").
- After you have selected the map you wish to use, click **OK**. The new map will now appear in the Layer Selector dialog.

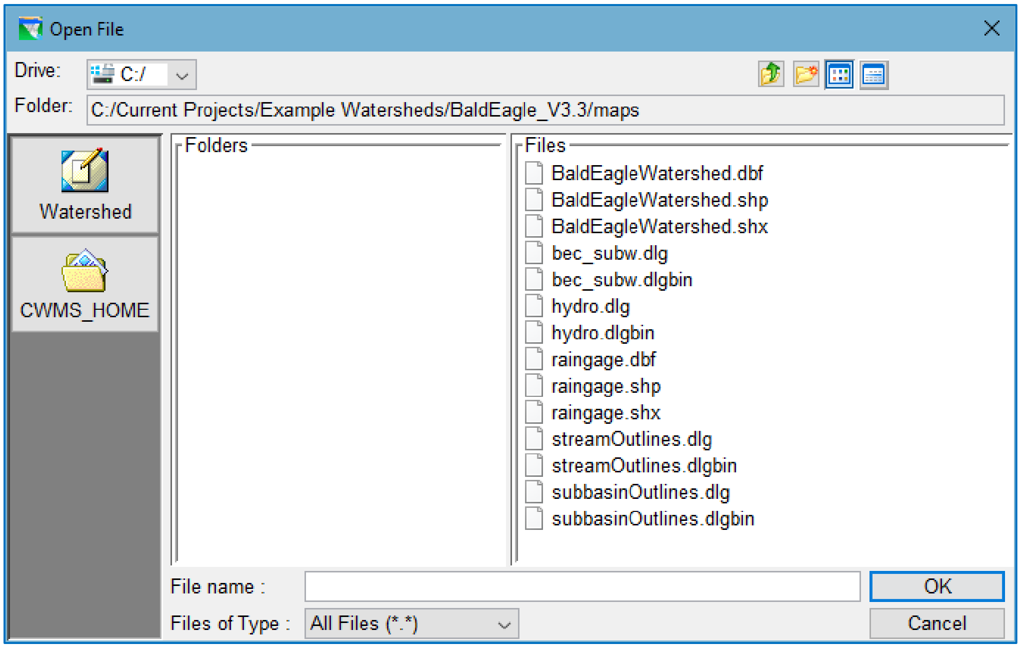


Figure 3.13 Open File Dialog to Add Map Layer

3.3.2 Setting the Geographic Coordinate System

To maintain a geographic reference, you must specify a coordinate system for each watershed. To establish the grid size and coordinate system:

- Select **Layers** from the **View** menu to open the **Layer Selector** dialog.
- From the **Maps** menu, select **Map Display Coordinates** to open the **Display Coordinates** dialog (Figure 3.14).

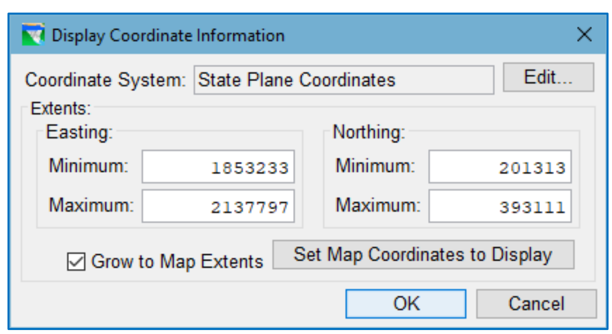


Figure 3.14 Display Coordinates Dialog

The **Display Coordinate Information** dialog includes the following data:

Coordinate System

This box identifies the coordinate system established for the watershed. The coordinate system can be edited, but remember that all maps and data must exist in one unified coordinate system.

Therefore, some layers or data might require the use of a GIS program to transform coordinate system information. For additional information about editing the Coordinate System, see "Editing the Coordinate System" paragraph below.

Extents

The **Easting Minimum** & **Maximum** and the **Northing Minimum** & **Maximum** values indicate the location of the left, right, top and bottom borders of the grid in the display area. ResSim displays coordinates in the **Status Bar** as you move the cursor within the map display panel.

Grow to Map Extents

When you check this box, ResSim automatically sets the geographic region to define the smallest rectangle that encompasses all the objects (maps, icons, alignments, etc.) in the display area. If you add any objects to the watershed that lie outside the geographic region, ResSim automatically updates the extents to include the new objects.

You must specify the geographic extents of your watershed carefully to ensure that all map layers in your watershed setup are included. The extents you specify and the coordinate system you use must be consistent for all layers in the watershed. You should use GIS tools to transform map layers from one coordinate system to another before you use them in ResSim.

3.3.3 Setting Map Coordinates to Display

If the extent of your model schematic covers only a small portion of the extent of your map layers, you can automatically set the geographic extents of the Map Display area to the coordinates in your current view. Here's how:

- First, zoom in or out in the display area until the Map Display shows only your model schematic (with a little bit of margin),
- Next, open the **Display Coordinate Information** dialog from the **Layer Selector** and click **Set Map Coordinates to Display**. ResSim will set the maximum view extents of the Map Display area to the current view extents.
- Click OK to save your selection and close the dialog.

To return the view to the extents covered your map layers:

- Check the box labeled **Grow to Map Extents** then click the **OK** button.

3.3.4 Editing the Coordinate System

You can edit the Coordinate System of the Watershed by selecting **Edit** in the **Geographic Region** dialog. This opens the **Map Coordinate Information** dialog, where you can configure the System, Units, Spheroid, and other system-specific data. See Appendix B for more information on configuring map coordinate information.

3.4 Watershed Configurations

Before you begin to create the watershed schematic for your model (by establishing the stream alignment and adding projects and computation points), you should create a *Watershed Configuration*. As you learn more about configurations in this section, it may seem as though creating a configuration at this point is too early but doing so facilitates the ease of later steps. In addition, learning about configurations early will help make other concepts regarding reservoir networks and configurations clearer.

A Watershed Configuration is a specific physical arrangement of projects that will later be used as a template to create a reservoir network model. Configurations should be created to reflect particular watershed conditions (physical arrangement of projects) needed for your study. For example, you might create a configuration named *Existing Conditions* and another named *With Project Conditions* in which one or more projects have been added or removed with respect to the *Existing Conditions* configuration.

Configurations can be used in both real-time and planning contexts, although fewer configurations are usually needed to identify the set of projects to be modeled by real-time alternatives.

To create or edit watershed configurations, you must be in the **Watershed Setup** module *and* you must ***lock*** the configuration for editing:

- If necessary, select **Watershed Setup** from the **Module** selector on the Module Bar
- Click the **Lock** button on the Module Toolbar (so that it appears locked) OR select **Allow Editing** from the **Edit** menu.

3.4.1 The Configuration Editor

The **Configuration Editor** (Figure 3.15) is used to create, edit, and delete configurations. To open the **Configuration Editor**:

- Select **Configuration Editor…** from the **Watershed** menu.

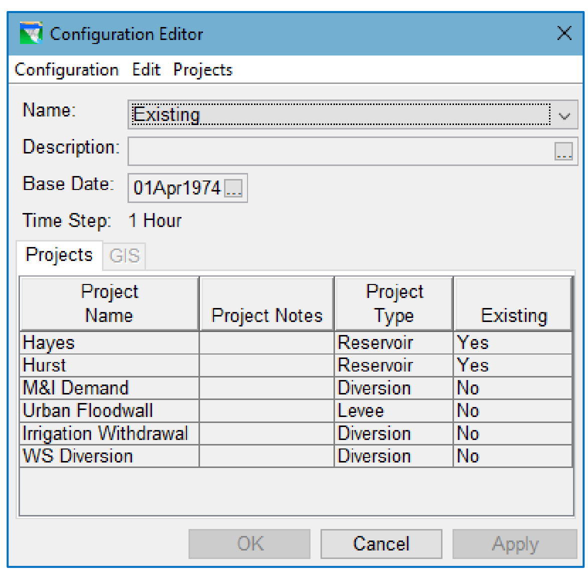


Figure 3.15 Configuration Editor

3.4.2 Creating Configurations

To create a *new* configuration:

- In the **Configuration Editor**, select **New** from the **Configuration** menu. The **Create a New Configuration** dialog (Figure 3.16) will open.
- Enter a **Name** and an optional **Description** for your configuration.
- Select a configuration **Time Step**. This timestep is not used by ResSim but it may be used by another program that shares the watershed such as CWMS or HEC-WAT. Select a timestep that you believe will be the typical timestep used by the alternatives you plan to develop in this watershed.

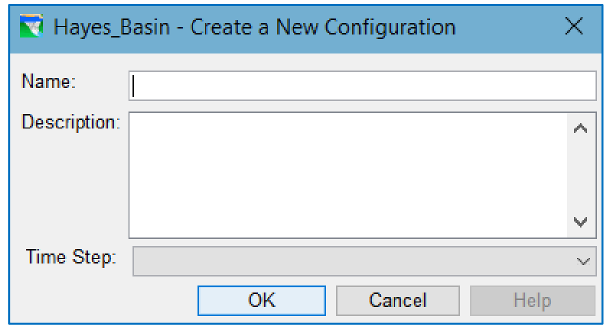


Figure 3.16 Create a New Configuration

- Click **OK** to close the **Create a New Configuration** dialog and complete the configuration creation process. The Configuration Editor, Figure 3.17, will open.
- The name of the new configuration will appear in the **Name** field of the **Configuration Editor**, as will its **Description** and **Time Step** in their associated fields.

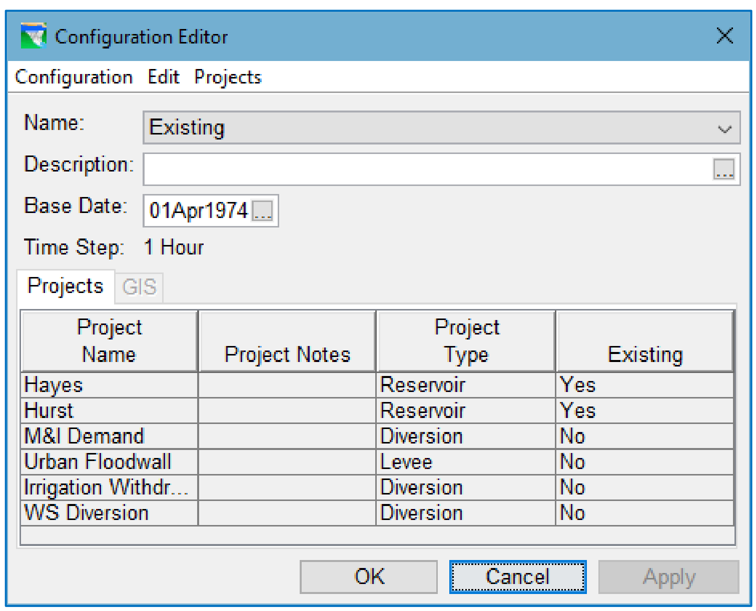


Figure 3.17 Configuration Editor

- Any project configuration element added to the watershed and flagged as an *Existing Project* (Section 5.2.4) before you created the new configuration, will automatically be included in the **Projects** table. To add projects to the configuration (that are not flagged as *Existing*) see Section 3.4.3 below.

3.4.3 Adding and Removing Projects from a Configuration

As you set up your watershed and manage it over time, you may need to add or remove projects from the configurations you have created. To do this:

- In the **Configuration Editor**, select the configuration you want to edit from the **Name** dropdown list (selector).
- From the **Projects** menu, select **Edit Project List** (Figure 3.18).

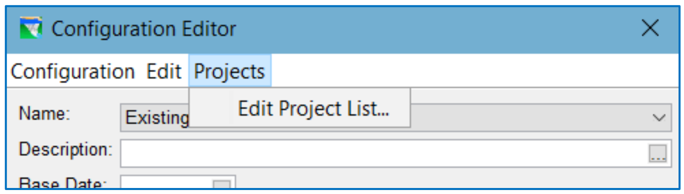


Figure 3.18 Configuration Editor—Projects Menu

- The **Project Selector** (Figure 3.19) will open.
- The **Project Selector** displays the active configuration's **Name** and **Description** for reference. Below the Description are two list panes with some buttons in between. The left pane lists the **Available Projects** that are not already associated with the active configuration. The **Projects in Configuration** pane on the right lists the projects that are currently associated with the active configuration. Use the buttons in the middle to add or remove projects from the active configuration by moving them from one pane to the other.
- Click **OK** to save your changes and close the Project Selector dialog.

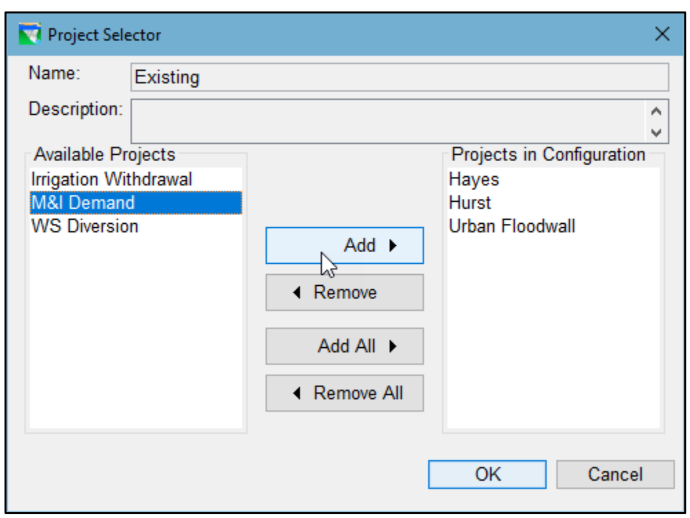


Figure 3.19 Configuration Editor—Project Selector

3.4.4 Setting the Configuration's Base Date

Once you have created a configuration, you can *optionally* associate a **Date** with it. The **Base Date** field is used by some HEC applications (e.g., HEC-FIA) that may share the watershed, particularly in a planning context. You can either type the date into the field in the appropriate format (e.g., 01Feb2002) or use the **Calendar Tool** (Figure 3.20).

To use the Calendar Tool to specify a **Base Date**:

- In the **Configuration Editor**, click the button to the right of the **Base Date** field. The Calendar Tool will appear.
- Select the month and year by clicking the left and right arrows to navigate backwards and forwards. Select the day by clicking on the appropriate day in the month panel.
- Click **OK** to accept the date you have selected and close the Calendar tool. The date you selected will appear in the **Base Date** field of the **Configuration Editor**.

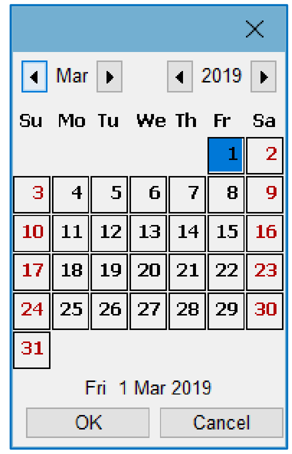


Figure 3.20 Calendar Tool

3.4.5 Adding Project Notes to a Configuration

Project notes within a Configuration are comments that pertain specifically to a project in a particular Configuration. Although ResSim does not use these project notes, they may be useful to one of the models that share the watershed.

To **Add** or **Edit** project notes within a Configuration:

- In the **Configuration Editor**, select the configuration of interest from the **Name** selector.
- Select a project from the list on the **Projects** tab.
- Select **Project Notes** from the **Edit** menu. The **Project Notes Editor** will open (Figure 3.21).

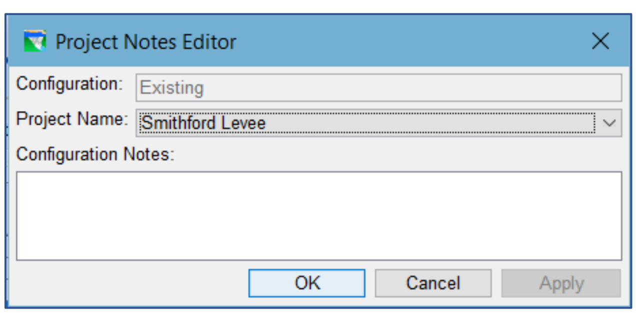


Figure 3.21 Project Notes Editor

- In the **Configuration Notes** textbox, enter configuration-specific notes as needed.
- Click **OK** to apply your changes and close the **Project Notes Editor**.

3.4.6 Copying a Configuration

Sometimes you might want to make an exact copy of an existing configuration and then make some changes to it so you have a different configuration. To make a copy of a configuration:

- In the **Configuration Editor**, select the configuration you want to copy from the **Name** selector.
- Select **Save As…** from the **Configuration** menu. The **Save As…** dialog (Figure 3.22) will open.

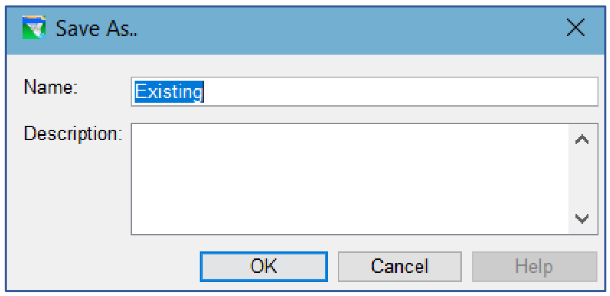


Figure 3.22 Configuration Menu—Save As…

- The **Name** field contains the name of the selected configuration. Replace this name with a new one and enter/edit the description.
- Click **OK** to complete the creation of the new configuration and close the **Save As…** dialog. The name of your new configuration will be displayed in the **Name** field/selector of the **Configuration Editor**.

3.4.7 Deleting a Configuration

To delete a configuration:

- In the **Configuration Editor**, select the configuration you want to delete from the **Name** selector.
- Select **Delete** from the **Configuration** menu.
- A confirmation dialog will open asking you if it's ok to delete the configuration. Click **Yes** to complete the deletion and close the confirmation dialog. **No** will cancel the delete.

3.4.8 Saving Configuration Data

To save the data you have defined for your configuration(s):

- Close the **Configuration Editor** and choose **Yes** when prompted to "Save/Apply Changes?" This saves you change in memory. Then,
- Select **Save Configuration** from the **Watershed** menu of the **Watershed Setup** module. This will cause the ResSim to save all configuration specific data to disk.

3.4.9 The List of Configurations Report

Once you have all your configurations defined, you might like to review the basic data for the configurations in your watershed. In the Reports menu of the **Watershed Setup Module** is a report, called **List of Configurations**, which contains this information.

To view the **List of Configurations** report (Figure 3.23):

- Select List of All Configurations from the Reports menu of the Watershed Setup module.

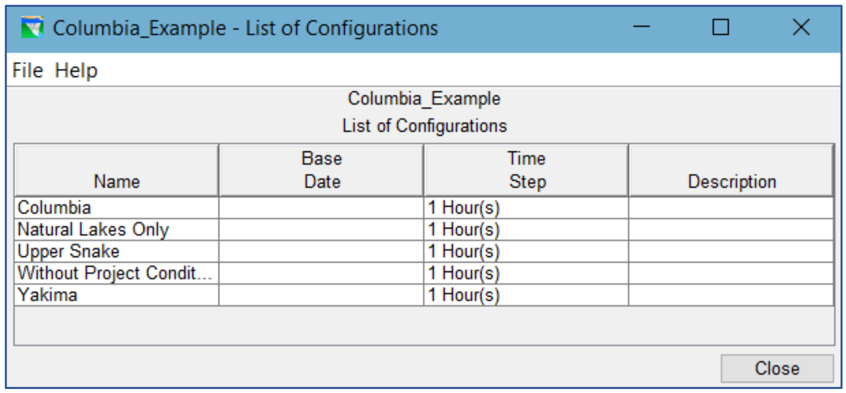


Figure 3.23 List of Configurations Report

- This report provides a list of the configurations that have been defined for the watershed. Data included in the list are the **Name**, **Description**, **Time Step,** and **Base Date** specified for each configuration.
- From the **File** menu of the report, you can send the report to a printer or export it to a text file. See Appendix E for details about printing and exporting tabulated data.
- Click **Close** to close the **List of Configurations** report.

Chapter 4
Defining the Stream Alignment

The **Stream Alignment** represents the river system of the watershed. It indicates where confluences and bifurcations occur and provides a sense of distance and scale. It also imposes a flow direction for reaches and reservoirs that are created with respect to it. Figure 4.1 shows the stream alignment created for one of the example watersheds that are provided with ResSim.

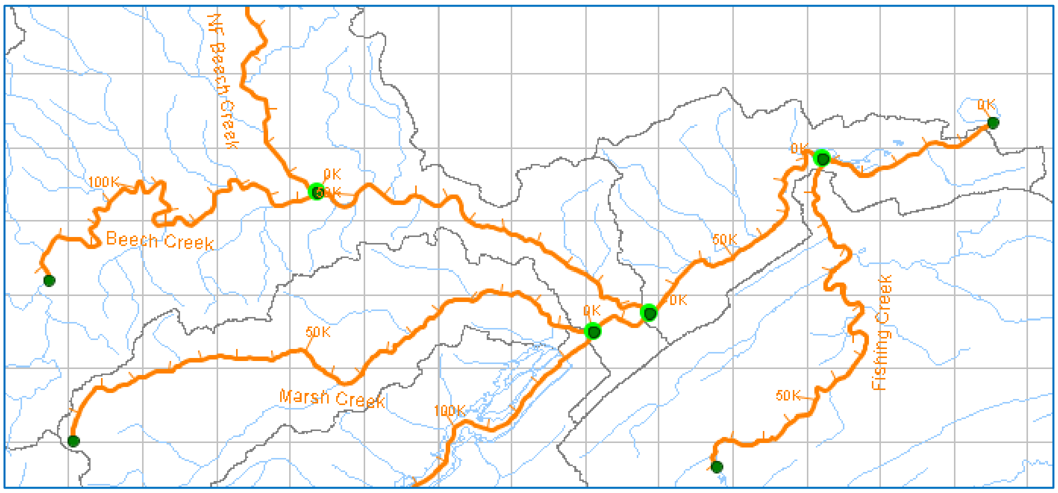


Figure 4.1 Stream Alignment

The **Stream Alignment** is made up of *streams*, *stream nodes*, and *stream junctions*. Figure 4.2 illustrates these components of the **Stream Alignment**. Although you can change the drawing properties of the elements of the stream alignment, by default, streams are drawn in orange, stream nodes in dark green, and stream junctions are symbolized by a bright green halo around a stream node.

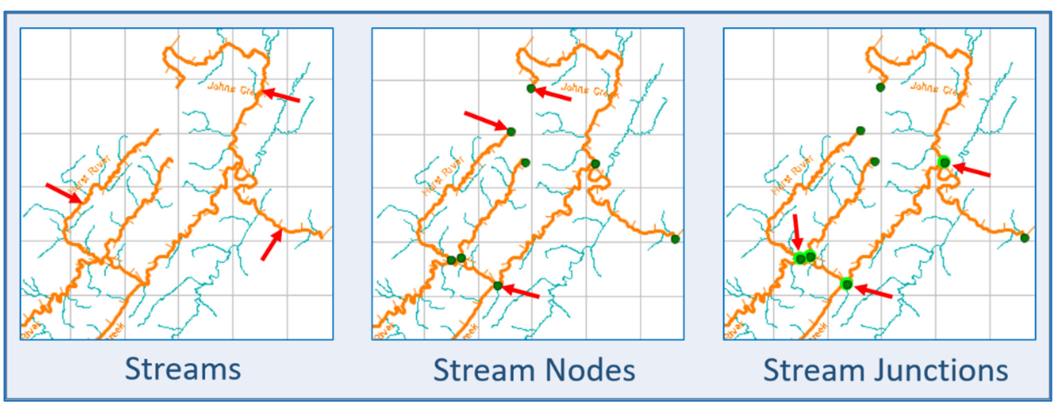


Figure 4.2 Stream Alignment Conponents

A *stream* in the stream alignment is defined by an ordered set of points or vertices that ResSim connects with straight line segments to form a "polyline". The **Stream Alignment Tool** is used to draw and edit streams in the Stream Alignment. When you create a stream, you will draw it, *from upstream to downstream*, as a series of points. Figure 4.3 illustrates the points and line segments that define a stream in the stream alignment. NOTE: Each stream should be a continuous element that represents a named watercourse in the river network; a river should not be broken up into discrete streams (reaches) at each confluence.

For each stream you create, ResSim will create two *stream nodes* and place them on the stream, one at the upstream-most end of the stream and one at the downstream-most end. When you connect one stream to the middle of another stream, ResSim will add an additional stream node to the existing stream at the point of connection. That connection, usually representing a confluence of two streams, is referred to as a *stream junction* and is denoted by the bright green halo around the stream node(s). Each stream junction contains a stack of at least two stream nodes, one for each stream entering and/or leaving the junction. *Stream nodes* and *stream junctions* are managed using the **Stream Node Tool** in the **Watershed Setup** module.

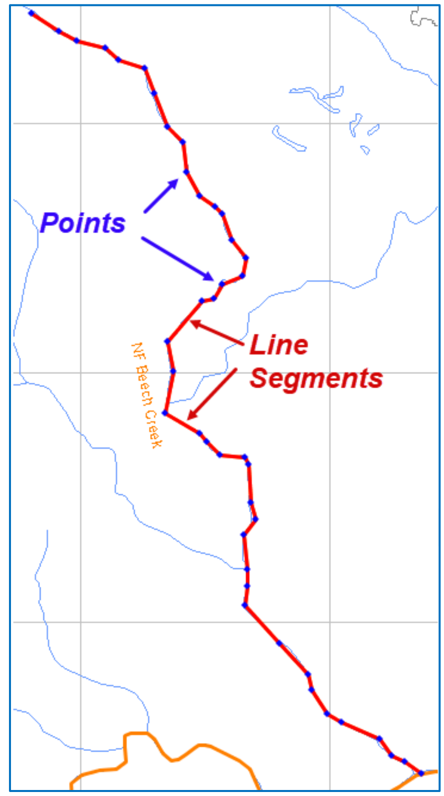


Figure 4.3 Visualization of a Stream

ResSim applies stream stationing to each stream in the stream alignment. By default, the stationing of a stream starts at zero at the downstream end and increases upstream along the stream as an accumulation of the lengths (in the watershed coordinate system and units) of the line segments that describe the stream. The stream stationing is reflected in the drawing of the stream alignment with tic marks and tick mark labels. You can override the default stationing by editing the station values of the stream nodes at the upstream and downstream ends of the stream element. You can also add additional stream nodes along the stream and override their default station values, thereby imposing a "piece-wise" stationing between each pair of stream nodes.

To work with the stream alignment in any way, you must be in the **Watershed Setup** module *and* you must *lock* the configuration for editing:

- Select **Watershed Setup** from the **Module** selector on the Module Bar
- Click the **Lock** button on the Module Toolbar. (so that it appears locked) OR select **Allow Editing** from the **Edit** menu.

4.1 Defining the Stream Alignment

You have two options for defining the stream alignment. You can:

1. Digitize (draw) each stream by hand
2. Import your stream alignment from an ArcView® shapefile.

Although it is not necessary, you may find it useful to display a background map of the river system as a guideline for drawing the streams for your stream alignment. If your stream alignment is going to be imported from a shapefile, that shapefile should already have been included as a map layer in the watershed. See Section 3.3 or Appendix B to learn how to add map layers to your watershed.

To be useful as a guide for drawing the stream alignment, the map layer should contain water features, such as stream channels and reservoirs or lakes. Also, by adding at least one background map layer to the watershed, you have implicitly established the coordinate system and units of the watershed; the stream stationing of your stream alignment will be determined using in the watershed coordinate system and units. See Chapter 3 for more information about specifying the watershed coordinate system and units.

4.1.1 Drawing the Stream Alignment

The following are the steps for digitizing a stream alignment. (See Section 4.1.2 for details on importing the stream alignment.)

To create a stream in the stream alignment:

- Select the **Stream Alignment Tool**  from the Toolbar.
- Determine an *upstream* location where you want to start a stream element.
- Hold down the **Ctrl** key. Starting at the *upstream* end, draw the first vertex of the stream element by clicking the left mouse button.
- While *continuing to hold down* the **Ctrl** key continue moving the cursor downstream and clicking the mouse button to place more vertices and thus draw more of the stream element (Figure 4.4)

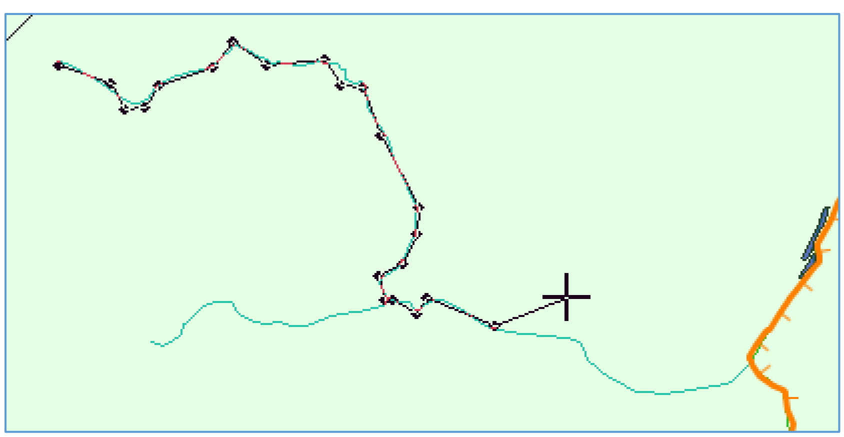


Figure 4.4 Drawing a Stream Element

- To place the *last* point of the stream, release the **Ctrl** key, then click at the last downstream location. An ending stream node will be created at the last point of the stream element.

- The **Create New Stream** dialog (Figure 4.5) will open after you place the last point of your stream. Enter a name (and an optional description) for the stream element, then click **OK** to close the **Create New Stream** dialog.

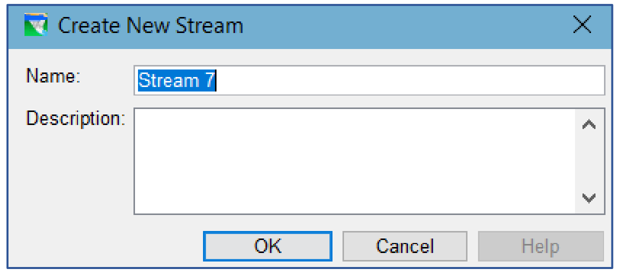


Figure 4.5 Create New Stream

- Repeat these steps to add more stream elements to your stream alignment.

> ***Tip*** When drawing the stream of your stream alignment…
> - Draw the mainstem (primary river) of your watershed) first.
> - Draw the major tributaries (of the mainstem) next.
> - Then draw the minor tributaries (of the major tributaries), etc.

To create a **Stream Junction** at a confluence (or bifurcation):

- Place the last (or first) point of the stream you are drawing directly on an existing stream.

- After naming the new stream, the **Connect Stream Reaches** dialog (Figure 4.6) will appear. You must determine whether you want to connect the new stream to the existing stream. Click **Yes** if you want the two stream elements connected. If you selected **No**, and if there are other existing streams that are possible candidates for connecting your new stream element to, then a subsequent **Connect Stream Reaches** dialog will appear to give you the opportunity to connect to the next existing stream.

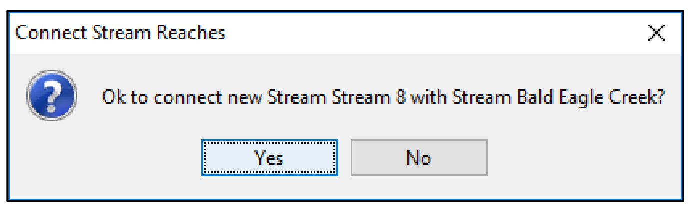


Figure 4.6 Connect Stream Reaches

- When multiple stream elements are connected (e.g., at a confluence), a *stream junction* is automatically created. A stream junction looks like a stream node highlighted with a bright green circle (a "bright green halo") as shown in Figure 4.7.

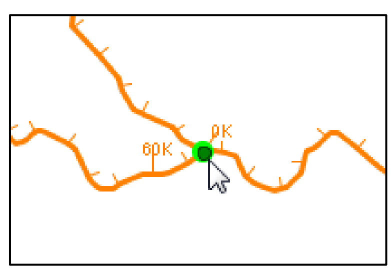


Figure 4.7 Stream Junction

- Once you have completed the creation of your stream alignment, save your work. From the

File menu, select **Save Watershed** and the stream alignment for your watershed will be saved.

4.1.2 Importing a Stream Alignment

Since creating and editing the stream alignment is a Watershed Setup activity, the stream alignment **Import** feature is only available in the **Watershed Setup Module**. It can be found in the **Watershed** menu in a submenu under the **Import** option.

The Importer can only import streams from an ArcView® shapefile containing polylines that represent the stream system of your watershed. The polylines in the shapefile must have an attribute that is the name of each stream, and each stream must form a single contiguous line. In addition, the Importer has been designed to look for appropriate shapefiles from the active map layers of your watershed. Although not required, before importing, you should first add the shapefiles that represent the centerlines of the rivers and creeks of your watershed as *map layers* and make them *active*. Refer to Chapter 3, Section 3.3.1 for instructions on "Adding a New Map Layer" to your watershed.

To import a stream alignment:

- You must be in the **Watershed Setup** module.
- If you haven't already, use the **Layer Selector** (**View** menu) to add the relevant shapefile as a map layer and make it active (check it on).
- From the **Watershed** menu, select **Import**, and then select **Stream Alignment** from the Import submenu.
- The **Import Stream Alignment** dialog (Figure 4.8) will open.

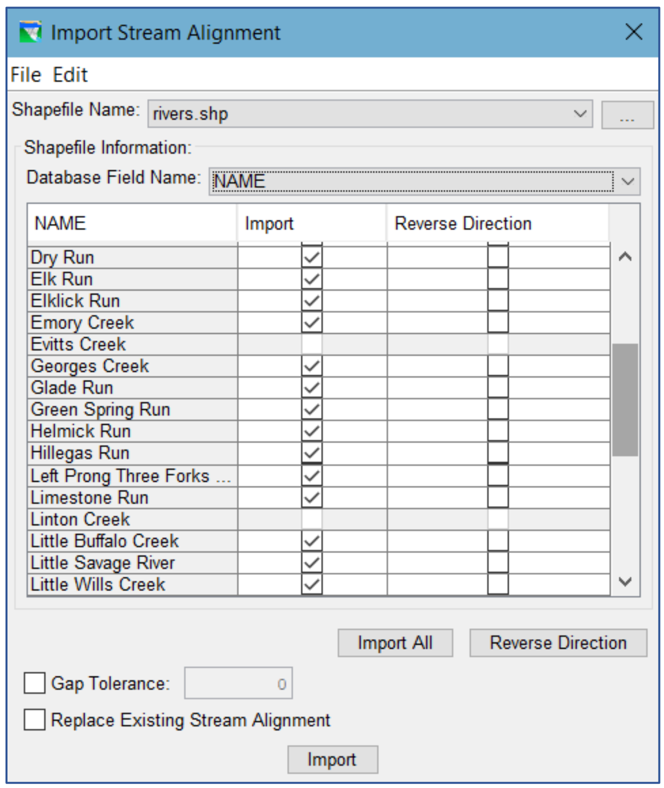


Figure 4.8 Stream Alignment Importer

- The **Shapefile Name** selector (drop-down list) should show the name of one of the *active* line shapefile map layers that have been added to your watershed. If the first entry is not the desired shapefile, select the correct one from the drop-down list.

  Note: if the desired shapefile is not in the **Shapefile Name** selection list, then it is not an active map layer in the watershed. However, you can still import from a shapefile that is not a map layer in the watershed:

- From the **File** menu of the Importer, select **Choose Shapefile**. The **Choose Shapefile** dialog (Figure 4.9) will open. Browse to the folder containing the shapefile you want to add, select it, and click **OK**.

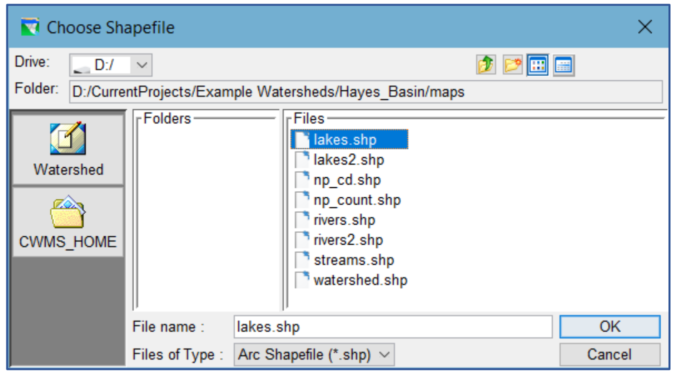


Figure 4.9 Choose Shapefile for Importing Stream Alignment

- Once the correct shapefile has been chosen, select the appropriate *attribute* that identifies the individual streams from the **Database Field Name** selector. This is typically a "name" attribute or field.
- The Importer automatically searches for an attribute of **Stream_ID**; if it doesn't find one, it will select the first attribute in the selection list by default. You should review the available attributes in the selection list to find the one that contains the names of each stream.
- The table below the **Database Field Name** selector will fill with the values of the selected **Database Field Name** (attribute). Hopefully, these values will reflect the names of the streams in your watershed.
- If the polyline(s) identified by each displayed attribute value forms a single contiguous line, then the checkbox in the **Import** column of the table will be set (checked). If there is something wrong with the line that represents a stream, the checkbox will *not* be set, and ResSim will not import that stream element.

Tip

You can view the streams before Importing!!!

- Move the Import Stream Alignment dialog (the Importer) so that it doesn't overlap the main ResSim window. You need to be able to see the Map Display Area.
- Click (select) a row in the table of attribute values.
- The polyline(s) identified by the attribute value in the selected row will highlight in red in the Map Display. Even polylines that the Importer has identified as "non-contiguous" and won't import can be highlighted in this manner—making them easy to identify and review.

Note: Only <u>active map layers</u> are visible in the Map Display so you cannot view streams from a shapefile you have selected with the Choose Shapefile browser opened from the Importer's File menu.

- The **Gap Tolerance** (Figure 4.8) value is used to enable the Importer to create stream junctions (connections) between connect stream elements that have a gap between the end point of one stream and the nearest line segment or endpoint of another stream. By default, the **Gap Tolerance** is set to zero (0.0), so ResSim can only connect streams where the endpoint of one stream is exactly coincident with a stream segment or vertex of another stream. If you click **Gap**

Tolerance, then you need to enter a value larger than zero in the **Gap Tolerance** box, which will allow ResSim to connect streams that might have a larger gap between them.

- The **Replace Existing Stream Alignment** box, allows you to replace the existing stream alignment with the one being imported from the selected ArcView® shapefile.
- Once everything is set, click **Import**. The stream alignment will be automatically drawn in the display area. From the **File** menu, choose **Close** and the **Import Stream Alignment** dialog (Figure 4.8) will close.

4.2 Editing the Stream Alignment

After drawing a stream alignment in the display area, there are many options available for revising your stream alignment. To edit your stream alignment, you must be in the **Watershed Setup Module**. The **Zoom Tool** can be used to help magnify the stream network, thus providing more detail to make editing the stream alignment easier.

This section describes the following capabilities for editing your Stream Alignment:

- how to refine and re-shape your stream alignment (by moving/adding/deleting vertex points of the stream elements)
- how to review stream stationing and provide a stream element description (by using the stream element editor)
- how to rename and delete stream elements
- how to add and edit stream nodes (for defining stream stationing)
- how to move and edit stream junctions
- how to reverse the flow direction of a stream element
- how to disconnect a stream element

Since the Stream Alignment may be referenced by other models, care should be used when making any changes to it.

4.2.1 Reshaping a Stream

Reshaping a stream means that you must reposition (move) the vertices that form the stream element.

To move a vertex of an existing Stream Element:

- Using the **Stream Alignment Tool**, *double-click* on the stream you want to reshape. The stream you selected will turn red and all the vertices of that stream element will turn blue.
- Click on the vertex you want to move, and drag it to a new location, as illustrated in Figure 4.10.

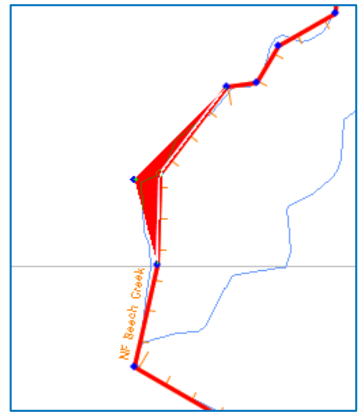


Figure 4.10 Reshaping a Stream

4.2.2 Adding Vertices to a Stream

To add a vertex to an existing stream element:

- With the **Stream Alignment Tool** selected, *double-click* on the stream so that the vertices are visible—they should appear as blue dots.
- Place the pointer on the selected stream where you want to add a vertex. Hold down the **Ctrl** key and click. A new blue dot will appear on the selected stream where you have added a new vertex.

4.2.3 Deleting Vertices from a Stream

To delete a vertex from an existing stream element:

- With the **Stream Alignment Tool** selected, *double-click* on the stream so that the vertices are visible—they should appear as blue dots.
- Hold down the **Shift+Ctrl** keys and click on the vertex point to be deleted. The blue vertex point will disappear from the stream element, and the stream will straighten between the two adjacent vertices.

4.2.4 The Stream Editor

A **Stream Editor** (Figure 4.11) is not really an *editor* but rather a report that shows the list of nodes that belong to the stream and their positions. The nodes and their positions are not editable from the table. The only editable field on this *editor* is the **Description**.

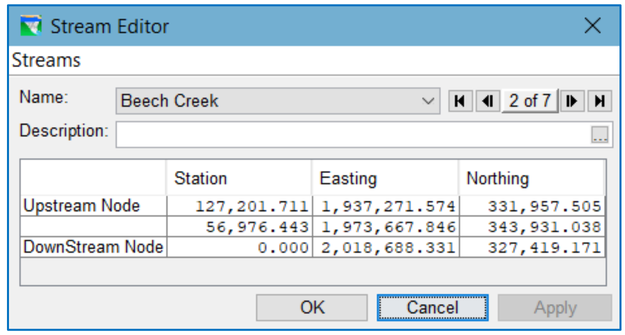


Figure 4.11 Stream Editor

To open the **Stream Editor**:

- Using the **Stream Alignment Tool** , *right-click* on the stream you want to edit.
- Select **Edit Stream Element** from the context menu (Figure 4.12) to open the **Stream Editor**.

The **Stream Editor** fields include:

Name: This is the name of the stream element you selected. However, you can view information for any stream in the Stream Alignment by either selecting it from the Name selector or by using the left and right arrows (navigator buttons) to click through the available streams.

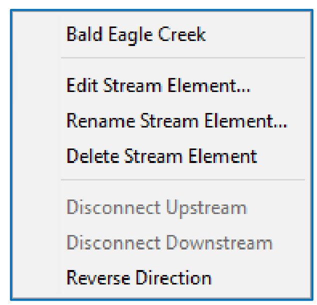


Figure 4.12 Stream Alignment Context Menu

Description: To edit the **Description** of the stream, you can type directly into the textbox or click the  button at the end of the Description textbox to access the larger **Enter Description** dialog (Figure 4.13).

Node List: the table below the **Description** is the Node list. It lists the nodes from upstream to downstream and shows the Station, Easting, and Northing of each node. Since the position of the nodes cannot be edited from the Stream Editor, use the **Stream Node Editor** (described in Section 4.2.8) to edit the stream stationing of the nodes.

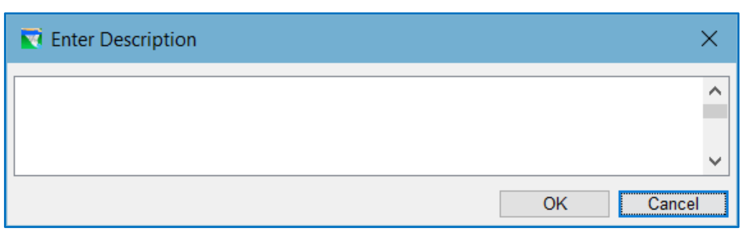


Figure 4.13 Enter Description Dialog

4.2.5 Renaming a Stream Element

To rename a stream element:

- Using the **Stream Alignment Tool** , *right-click* on the stream element.
- Select **Rename Stream Element** from the context menu.

  Enter a new **Name** (and optional **Description)** for the stream element in the **Rename Stream** dialog (Figure 4.14).

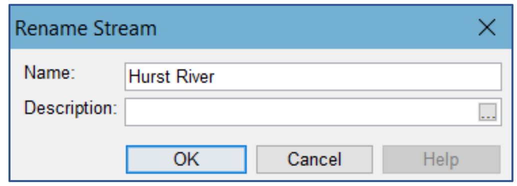


Figure 4.14 Rename Stream

4.2.6 Deleting a Stream Element

To delete a stream element:

- Using the **Stream Alignment Tool** , *right-click* on the stream element.
- Select **Delete Stream Element** from the context menu.
- When the confirmation message appears (Figure 4.15), select **Yes** to delete the stream or **No** to cancel the delete.

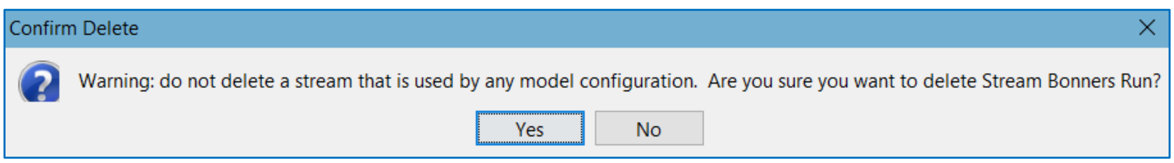


Figure 4.15 Confirmation Message when Deleting a Stream Element

4.2.7 Inserting a Stream Node

By default, ResSim generates beginning and ending stream nodes for each stream element. Also, stream nodes are automatically created where stream elements connect (stream junctions). Since stream nodes are used for establishing the stream stationing, you may want to include additional stream nodes along your stream element for locations where you want to define specific stream stationing.

To insert a stream node on an existing Stream Alignment:

- Using the **Stream Node Tool** , hold down the **Ctrl** key and *right-click* on the Stream Alignment in the place where the stream node is to be inserted.
- When you right-click on the stream station node, a context menu appears (Figure 4.16).

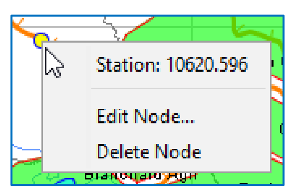


Figure 4.16 Stream Node Context Menu

- From the context menu on the stream node, you can choose **Edit Node**, which will bring up the **Stream Node Editor** illustrated in Figure 4.17.

Remember: other models that share the watershed (e.g., HEC-FIA) may use the same stream alignment, so be sure to confirm with other modelers before making any changes to the stream alignment including deleting and re-stationing streams.

4.2.8 Editing a Stream Node

The **Stream Node Editor** (Figure 4.17) displays the location information for the node. The Easting and Northing are for your information only and should not be edited. The Station is the distance in map units of the stream node from the downstream-most end of the stream element.

Use Default Stationing—this check box is *checked* by default. When checked, the station of the node is not editable.

If you want to specify the station (distance) for the node on the stream element, then uncheck the **Use Default Stationing** checkbox. This will make the Station field editable. Enter the relative distance (from the downstream end of the stream element) of the node directly into the **Station** field, then click **OK**.

NOTE—changing the station of a stream node does not change the node's position along the stream element. Instead, it changes the relative (piecewise) stationing of the stream element.

4.2.9 Deleting a Stream Node

Since stream nodes define the stream stationing that may be referenced by other models, care should be used when deleting stream nodes.

To delete a stream node from an existing Stream Alignment:

- Find the stream node you wish to delete.
- Using the **Stream Node Tool** , *right-click* on the stream node.
- Select **Delete Node** from the context menu.
- A **Confirm Delete** message will display (Figure 4.18) asking you whether you really want to delete the selected stream node.
- Click **Yes** and the message will close, and the stream node will be deleted from the stream element.

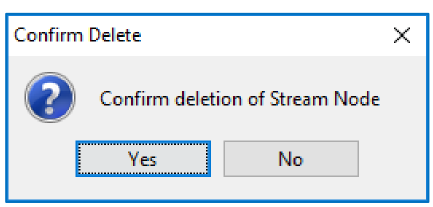


Figure 4.18 Confirm Delete Dialog

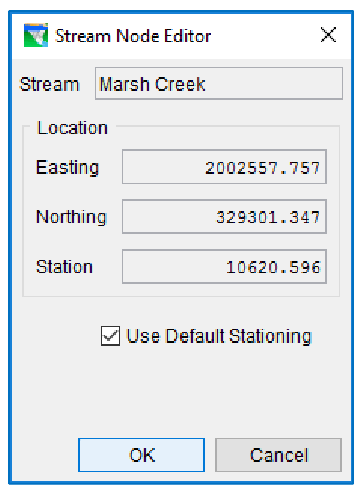


Figure 4.17 Stream Node Editor

4.2.10 Moving a Stream Junction

When two stream elements intersect (e.g., at a confluence), a **Stream Junction** is automatically created. A stream junction represents multiple stream nodes at one location and is drawn in the Map Display with a dark green dot surrounded by a bright green halo (Figure 4.19). Although ResSim generates default stream junctions when you connect one stream to another in the Stream Alignment, you can move them as needed.

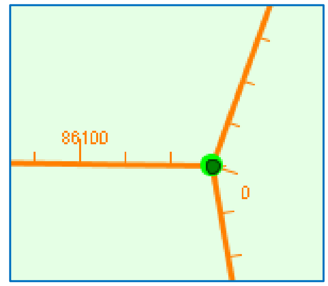


Figure 4.19 Stream Junction

To move a **Stream Junction** along an existing Stream Alignment:

- Using the **Stream Node Tool**, *double-click* on the stream junction. The stream junction will turn dark green and will have small squares called *move handles* at each corner (Figure 4.20).
- Drag the stream junction along the (downstream) stream element to its new position.

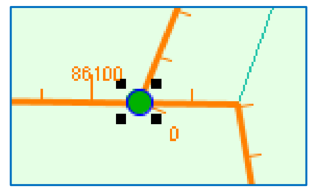


Figure 4.20 Stream Junction with Move Handles

4.2.11 Editing a Stream Junction

To specify the stream stationing of the stream nodes at a Stream Junction:

- Using the **Stream Node Tool**, *right-click* on the Stream Junction to access its context menu (Figure 4.21).
- Select **Edit Node** to open the **Stream Junction Editor** (Figure 4.22).

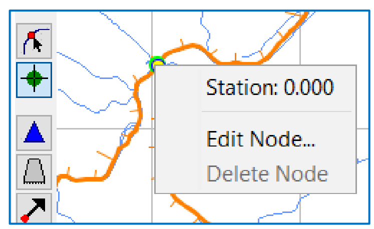


Figure 4.21 Stream Node Context Menu

The **Stream Junction Editor** displays the list of Stream Nodes associated with a stream junction and allows you to edit the stationing of each of the nodes by deselecting the "Use Default Stationing" box and entering the desired station. Did you notice that the **Stream Junction Editor** is a just a special form of the **Stream Node Editor**?

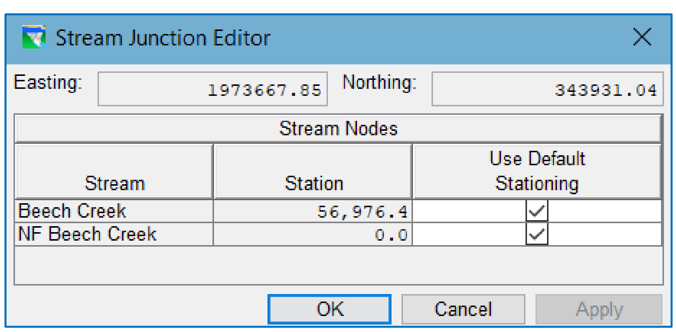


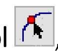
Figure 4.22 Stream Junction Editor

4.2.12 Deleting a Stream Junction

Occasionally, due to user or software error, a stream junction is created connecting a stream to the wrong stream at the confluence. To delete the stream junction, you simply need to **disconnect** the stream that ends (or begins) at the junction. See Section 4.2.14 for details.

4.2.13 Reversing the Direction of a Stream

If you find that you have inadvertently drawn your stream element in the wrong direction (i.e., downstream to upstream when it should be upstream to downstream), don't and redraw it; reverse it. To reverse the direction of a stream element:

- Using the **Stream Alignment Tool** , *right-click* on the stream element and select **Reverse Direction** from the context menu.
- A warning message (Figure 4.23) will open asking you whether you really want to reverse the direction of the stream element.

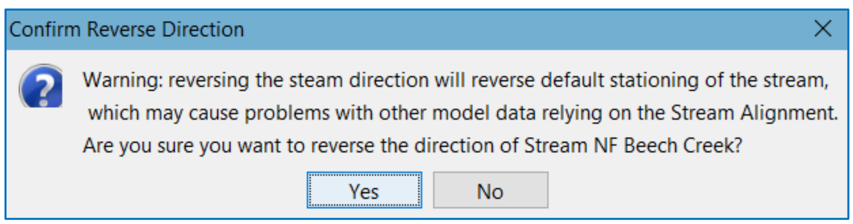


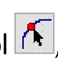
Figure 4.23 Confirm Reverse Direction of Stream Element

- Click **Yes**. The warning message will close, and the stream element's direction will be reversed.

4.2.14 Disconnecting a Stream Element

One stream element can may connect to another stream element at a Stream Junction (the confluence of two streams). If, for some reason, you need to disconnect one stream element from another, you can do so.

To disconnect a stream element from another stream element:

- Using the **Stream Alignment Tool** , right-click on the stream element you want to disconnect and select either **Disconnect Upstream** or **Disconnect Downstream**, as appropriate, from the context menu (Figure 4.23).
- The stream element will now be disconnected from the other stream element (Figure 4.24). In the display area, the end of the disconnected stream will be drawn a short distance away from the stream node where the stream junction had been and the stream junction (bright green halo) will disappear, leaving behind a stream

node where the stream junction had been. The stream node that was left behind is innocuous, but if you wish to remove it, see Section 4.2.9 above.

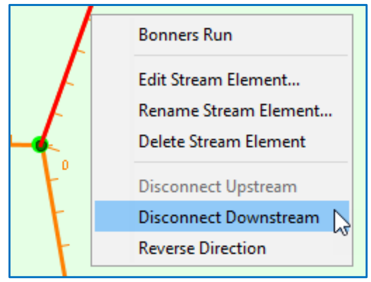


Figure 4.24 Stream Alignment—Context Menu

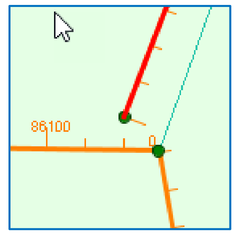


Figure 4.25 Disconnected Streams

4.3 The Stream Alignment Properties Editor

You can change the drawing properties of the Stream Alignment using the **Stream Alignment Properties** editor. Refer to Appendix B, Section B.7 for a detailed description of the Stream Alignment Properties editor.

To open the **Stream Alignment Properties** editor:

- From the **View** menu, select **Layers**. The **Layer Selector** will open.
- Right-click on the **Stream Alignment** layer in the **Layers** tree.
- From the context menu, select **Properties**.
- The **Stream Alignment Properties** editor (Figure 4.26) will open.

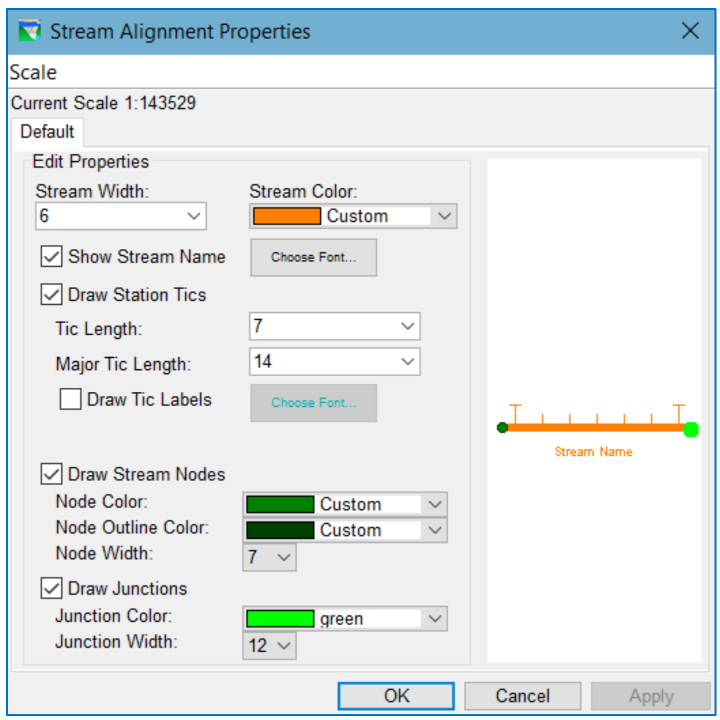


Figure 4.26 Stream Alignment Properties

4.4 Saving the Stream Alignment

Since creating a stream alignment can be a very detailed activity, you should save it frequently during its creation. From the **File** menu, click **Save Watershed**. This command will save all the stream alignment properties and the appropriate files created and used by ResSim for the stream alignment.

4.5 Exporting the Stream Alignment

If you have digitized a stream alignment by hand, you might want to save that stream alignment as an ArcView® shapefile. To export a stream alignment, you must have a stream alignment in the display area.

To export a stream alignment:

- From the **Watershed** menu, select **Export**, and then select **Stream Alignment** from the Export submenu.
- A **Save File** browser (Figure 4.27) will open.

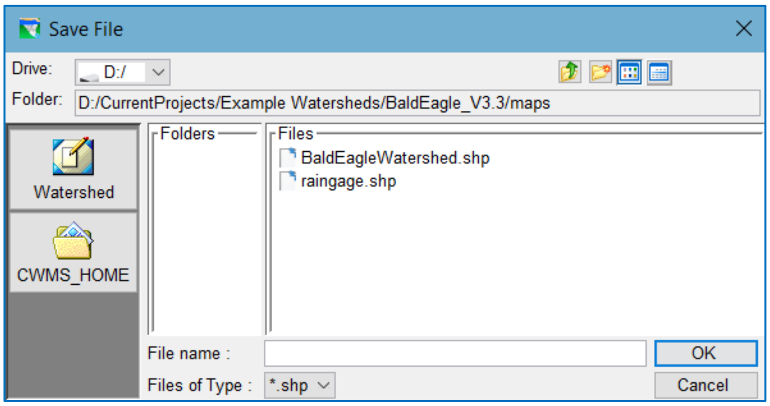


Figure 4.27 Save File Browser

- Enter a name in the **File Name** box. Click **OK**, the **Save File** browser will close, and you will have an ArcView® shapefile of the displayed stream alignment.

4.6 The List of Streams Report

Once you have your stream alignment completed, it's a good idea to review the streams in your stream alignment. The List of Streams report, accessible from the **Watershed Setup Module,** contains this information.

- From the **Reports** menu, select **List of Streams.** A report similar to the one shown in Figure 4.28 will appear.

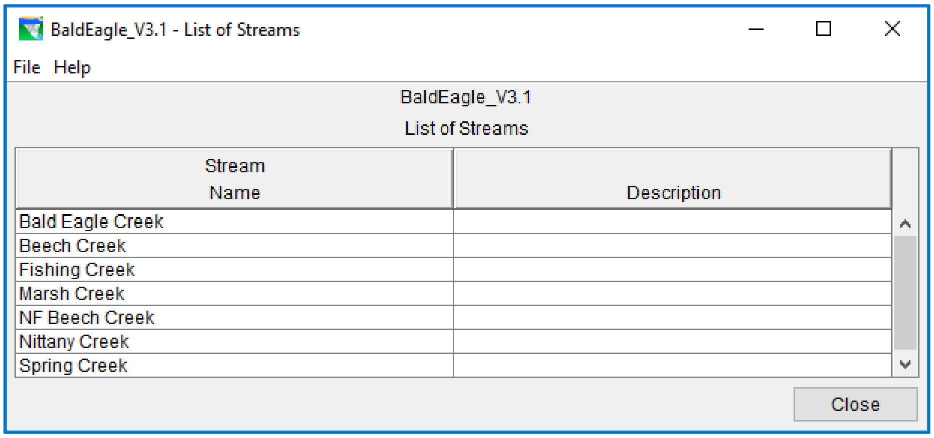


Figure 4.28 List of Streams in Stream Alignment

- This report provides a list of streams and their descriptions that have been defined in the Stream Alignment.
- Select **Print** from the **File** menu to print the report
- Select **Export** from the **File** menu to save the report as a tab-delimited text file.
- Select **Close** from the **File** menu or the **X** in the Title Bar to close the **List of Streams** report.

Chapter 5
Watershed Schematic Elements

In the **Watershed Setup Module** you can define the Watershed Schematic Elements (Projects, Computation Points, and Impact Areas) used by ResSim and by other models that may share the watershed.

The ResSim Map Toolbar provides the tools for drawing and editing the geographically-referenced schematic elements in the Map Display area. The drawing tools also provide access to context menus for managing the schematic elements directly from the map display.

When you add a Project (reservoir, diversion, etc.) to the Map Display, it becomes part of the active configuration (the one displayed in the Configuration selector on the Module bar). A "superset" of all configurations is named *Study* and includes all of the Projects for the watershed; although not technically a configuration, if you have not created a configuration, *Study* will be selected as the active configuration by default. For more information on associating projects with Configurations, refer to Section 5.2.4.

Of the array of available watershed schematic elements, ResSim models only use computation points, reservoirs, and diversions so these configuration elements will be covered first. Creating the other schematic elements will be described later in this chapter.

To create or edit the watershed schematic elements you must be in the **Watershed Setup** module *and* you must *lock* the configuration for editing:

- Select **Watershed Setup** from the **Module** selector on the Module Bar
- Click the Lock button on the Module Toolbar (so that it appears locked) OR select **Allow Editing** from the **Edit** menu.

5.1 Computation Points

Computation Points are locations where time-series information will be computed for possible exchange between models (e.g., ResSim and FIA). ResSim automatically generates computation points when projects are placed on the Stream Alignment so you should create computation points at locations that are not project related such as inflow and gage locations, control points, and confluences.

5.1.1 Drawing Computation Points

To draw a Computation Point:

- Using the Computation Point Tool , hold down the **Ctrl** key and *click* at the desired location on the Stream Alignment where you want a computation point.
- A **Name**… dialog will appear (Figure 5.1). Enter a **Name** for the computation point and an optional **Description**. When you are finished, click **OK**.

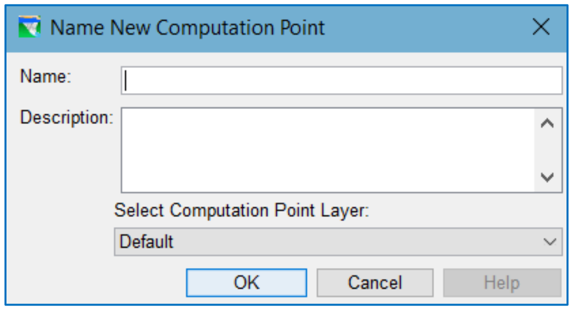


Figure 5.1 Name New Computation Point Dialog

The new computation point will appear in the Map Display. By default, computation points snap to the Stream Alignment. If you prefer that the computation point not exist on the Stream Alignment, de-select the **Snap to Stream Alignment** check box in the **Computation Point Properties Editor**.

5.1.2 Importing Computation Points

Another way to create computation points is to import them from a point shapefile. The shapefile must be an active map layer in the watershed. To import computation points:

- From the **Watershed** menu, select **Import,** then select **Computation Points …** from the **Import** submenu. The **Import Computation Points** importer will open (Figure 5.2).
    - Select the attribute in the **Database Field Name** selector that identifies the individual computation points. The table will fill with information about the points in the shapefile including the value of the selected attribute in the Name column, the stream in the Stream Alignment that it will import to, whether the point is at a stream junction in the Stream Alignment, and its stream station.
    - The Import column in the table is for you to edit—use it to select or de-select the point you want imported. If you want them all, click **Import All**.
    - Like the Stream Alignment Importer, you can locate the individual computation points on the map by selecting them in the list. The selected point will highlight in Red on the Map Display.

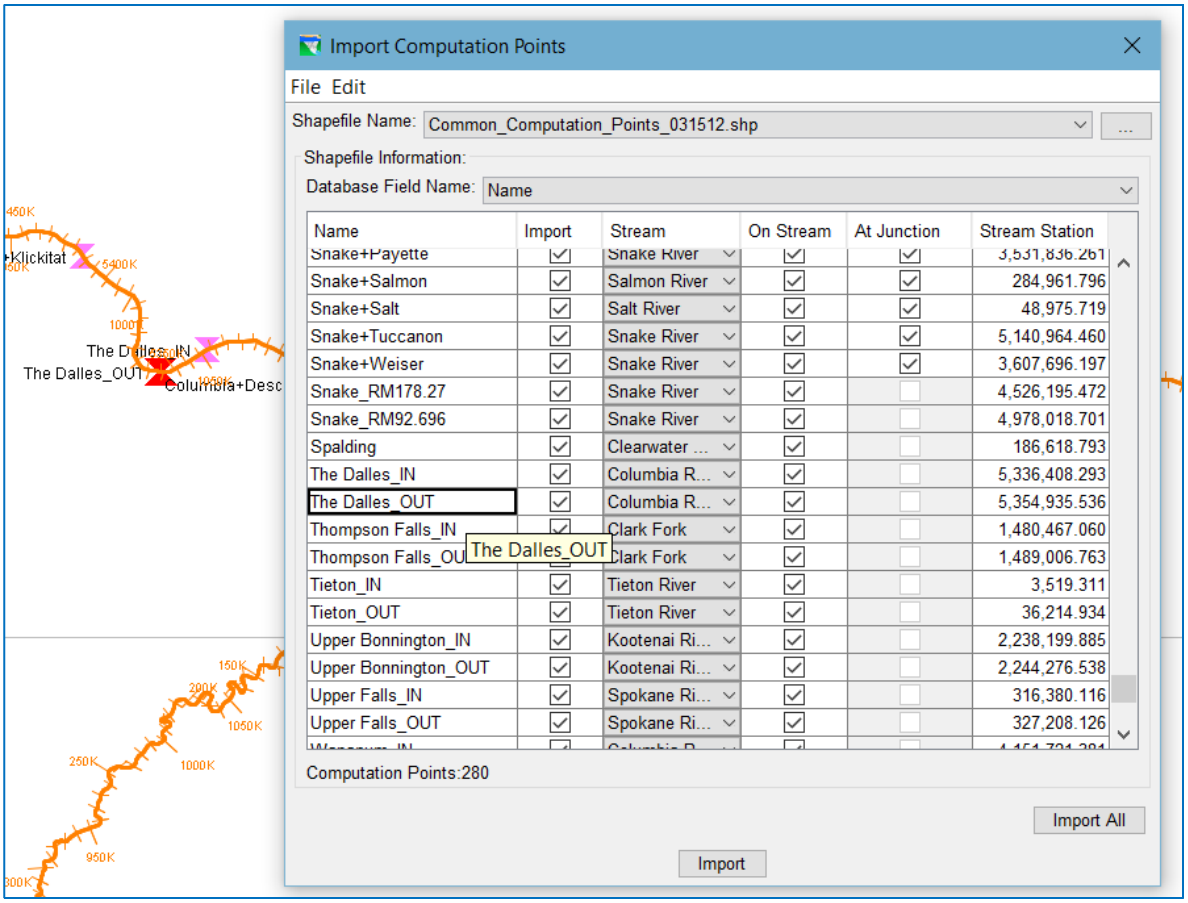


Figure 5.2 Computation Point Importer

- Once you have identified which points to import, click Import. The Importer will close, and you will get a message box telling you how many computation points imported (Figure 5.3).

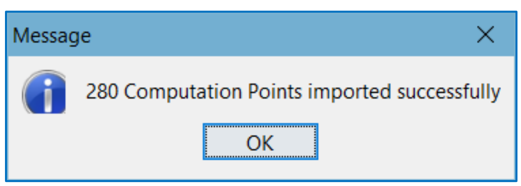


Figure 5.3 Successful Import Message

5.1.3 Editing Computation Point Data

Use the **Computation Point Editor** to edit a Computation Point:

- Using the Computation Point Tool, *right-click* on the computation point.
- Select **Edit Computation Point** from the context menu. The **Computation Point Properties Editor** (Figure 5.4) will appear.

The **Computation Point Properties Editor** displays information associated with a Computation Point and allows you to edit its location information.

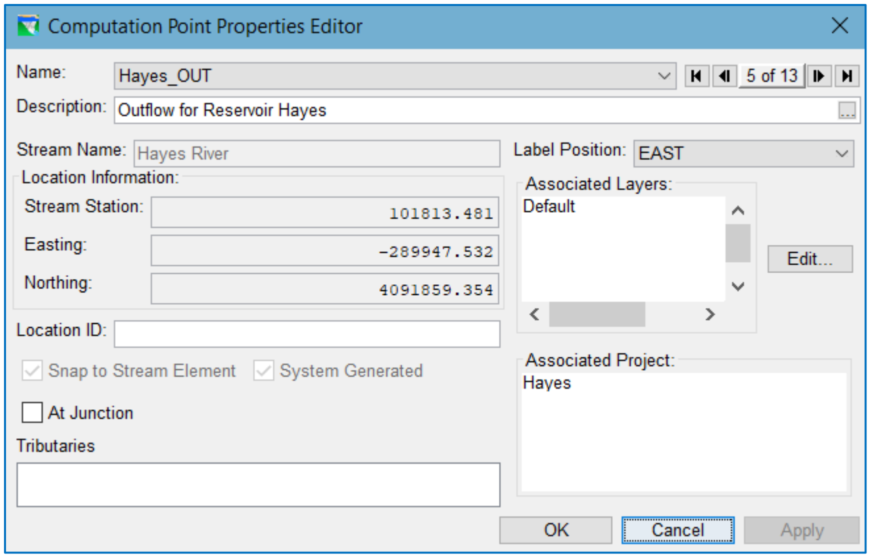


Figure 5.4 Computation Point Editor

Computation Point Name—The Computation Point Editor displays the name of the Computation Point you have selected (in the list along with all available Computation Points in the watershed). Also, left and right arrows allow you to click through the Computation Points in the watershed.

Description—To edit the Description of the computation point, click in the text box and start typing or click the  button to access the larger **Enter Description** dialog.

Stream Name—The stream name automatically appears in the Stream Name box. In the case of a computation point situated at a confluence of two streams, you need to verify that the computation point resides on the appropriate stream.

Label Position—ResSim automatically positions the text label for a computation point where it is least likely to overlap other labels. However, you can move the position of the layer by selecting a position from the Label Position list.

Location Information—The following allow you to provide location information.

Stream Station—The Stream Station field displays the location of the computation point along the stream element.

Easting / Northing—ResSim automatically fills in the coordinate information. You may type in new coordinates to relocate the computation point on the map display.

Location ID—Optionally, you can enter a Location ID to reference the computation point to a ResSim ID.

Snap to Stream Alignment—By default, each time you can draw a computation point, it will snap to the nearest stream in the Stream Alignment. If a computation point is snapped to the stream alignment, only the Stream Station of its location information can be edited. If you want the point located off-stream, uncheck this checkbox. An off-stream computation point does not have a valid

Stream Station, but its Northing and Easting values can be edited to move the point to the exact position you desire.

System Generated—ResSim automatically generates computation points for the watershed when you create certain project elements. The *System Generated* checkbox will be checked if the computation point has been generated with a project element. This checkbox is not editable. If checked, the relevant project should be listed in the **Associated Project** field.

5.1.4 Renaming a Computation Point

To rename a Computation Point or edit its description:

- Select the **Computation Point Tool** and right-click on the Computation Point.
- Select **Rename Computation Point**…
- Use the **Rename** dialog (Figure 5.5) to edit the **Name** and **Description**. Click the button to access the **Enter Description** dialog for longer descriptions.

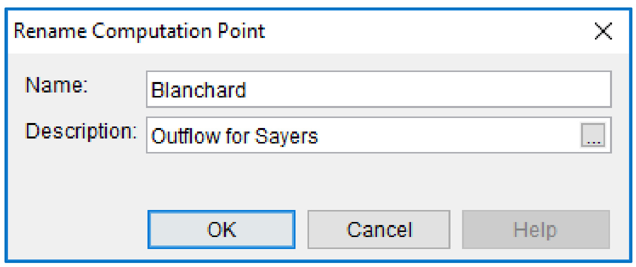


Figure 5.5 Rename Computation Point

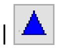
5.1.5 Deleting a Computation Point

To delete a Computation Point:

- Select the **Computation Point Tool** and right-click on the Computation Point.
- Select **Delete Computation Point** from the context menu. The **Confirm Deletion** dialog will appear.
- Click **Yes** to confirm the deletion.

5.2 Reservoirs

5.2.1 Drawing a Reservoir Element

A reservoir is created by using the Reservoir Tool  to draw it from upstream to downstream on the stream alignment. As soon as a reservoir has been drawn, several visual elements will appear that represent the reservoir (Figure 5.6). These elements include:

- a storage reach (cyan),
- a pool (cyan triangle),
- a dam (small gray rectangle),
- and 2 or more computation points (black dots) at the upstream and downstream end of the reservoir on the stream alignment.

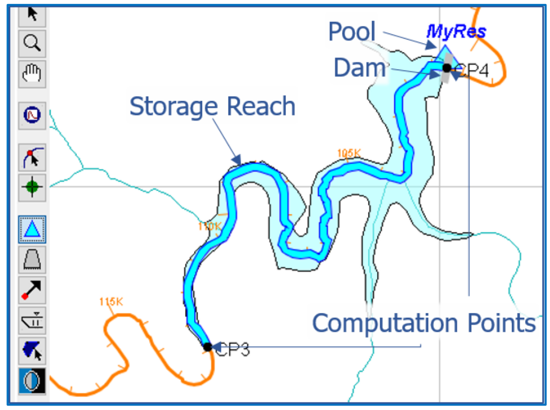


Figure 5.6 Basic Reservoir Element

To add a reservoir to the watershed:

- Select the Reservoir Tool 
- Press the **Ctrl** key and click on the stream alignment at the *upstream* extent of the reservoir pool. If more than one stream flows into the reservoir pool, continue pressing the **Ctrl** key and click at each location of inflow into the reservoir on each inflowing stream.
- Release the **Ctrl** key and click on the stream alignment where the dam is located. This is the downstream extent of the reservoir. *Note: When drawing a reservoir, do not overlap or encompass existing computation points.*
- The **Name New Reservoir** dialog will appear (Figure 5.7).
- Name the Reservoir by entering a name in the **Name** field. Optionally, you can also enter a **Description** for the Reservoir. Click **OK** when you are done. The new Reservoir will appear in the Map Display as illustrated in Figure 5.6.

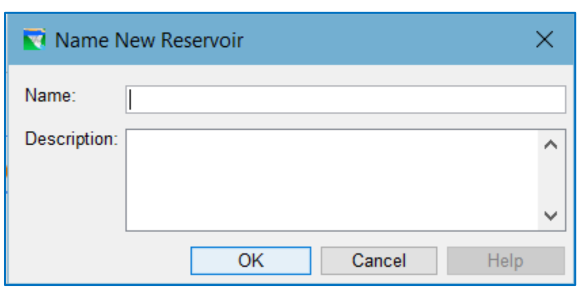


Figure 5.7 Name New Reservoir

5.2.2 Reshaping a Reservoir's Pool

The pool of a reservoir is represented by the storage reach as well as the triangle that is drawn near the dam. The storage reach follows the stream alignment so it cannot be reshaped without reshaping the stream alignment. However, the triangle is simply a polygon attached to the downstream end of the storage reach. The triangle can be reshaped to reflect the shape of the reservoir pool if you desire or it can be stretched to encompass a larger area so that it can be easily seen and selected when the Map Display is zoomed-out.

To **enlarge** the reservoir triangle:

- Using the **Reservoir Tool** , *double-click* on the reservoir's storage reach or triangle. The storage reach and triangle will highlight (turn yellow) and the vertices that define the triangle will display as blue dots as show in Figure 5.8.

Figure 5.8 Reservoir Ready for Reshaping

- The yellow dot at the apex of the triangle is the label handle for the Reservoir. Move the label handle away from the triangle.
- Drag one of the blue dots that define the triangle away from its current position. Repeat this for each corner dot of the triangle in order to enlarge the triangle.

If you would like to reshape the triangle so that it follows the outline of the reservoir, you will need to add additional points or vertices to the triangle polygon. To **add points** to the pool (triangle) polygon:

- Hold down the **Ctrl** key while clicking along one of the sides of the triangle. A new blue dot (point) will appear where you clicked.
- Add several new points to the polygon then drag the points around on the Map Display to form an outline of the lake.

You may end up adding more points to the polygon that you need. To **delete** points from the pool polygon:

- Hold down both the **Ctrl** and **Shift** keys while clicking on a point you want to delete from the polygon. The point will disappear.

5.2.3 Drawing a Reservoir with two or more dams

In Section 5.2.1, you saw how to create a reservoir with one dam and one or more inflow points. Those are the typical reservoirs you are likely to need, but every once in a while, you may encounter a situation in which the reservoir has multiple rivers it can release water into. By drawing the reservoir across two or more streams that flow away from the inflow(s), a dam will be created at each outflow junction of the reservoir. Figure 5.9 illustrates how to create a reservoir with two dams. In this example, an L-shaped dam was constructed upstream of the confluence of two streams, but outlets were included that could release water into both streams. This dam is treated as two dams in ResSim due to the need to produces releases to the two separate streams.

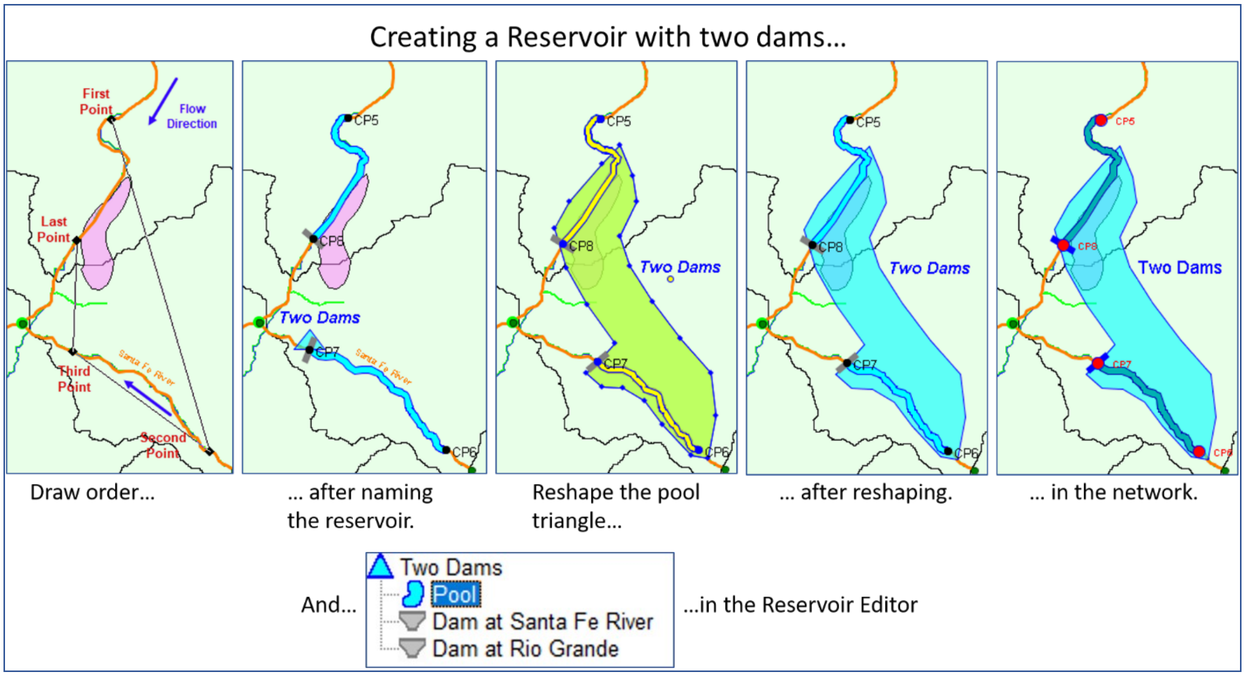


Figure 5.9 Creating a Reservoir with Two Dams

5.2.4 Editing Reservoir Data (Watershed Setup)

Physical and operational data of a reservoir belong to the reservoir *model element* and is entered in the Reservoir Network module. However, to edit configuration data related to the reservoir:

- Using the **Reservoir Tool** or the **Pointer Tool**, right-click on the reservoir.
- From the context menu, select **Edit Reservoir** to open the **Reservoir Properties Editor**.

The fields of the **Reservoir Properties Editor** (Figure 5.10) include:

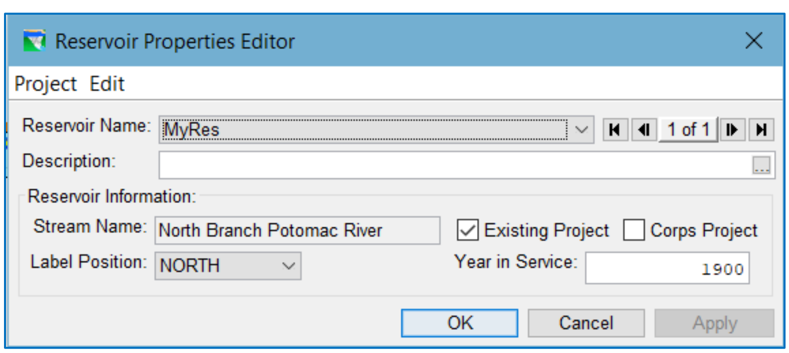


Figure 5.10 Reservoir Properties Editor

Name—For a reservoir schematic element, the only data that you need to specify is its **Name** which you took care of when you created it. The name field in the **Reservoir Properties Editor** is really a selector so that you can switch from one reservoir to another to review its configuration element properties.

Label Position—Select where to draw the reservoir name label relative to the reservoir's label handle. The label handle can be found by double-clicking on the reservoir using the **Reservoir Tool**; it will appear as a yellow dot (on the top of the reservoir triangle, if you haven't already moved it).

Existing Project—If the reservoir currently exists, this box should be checked so that the reservoir will be added automatically to any new configurations. This is a useful option if you are creating multiple configurations that use the same set of reservoirs. See Section 5.2.6 for removing a reservoir from a configuration.

Corp Project—Certain modeling software needs to know if the project is owned or operated by the Corps. For example, FIA needs to know in order to compute project benefits, therefore, use the **Corps Project** checkbox to indicate if the reservoir is a Corps Project. (This field is not used by ResSim.)

Year in Service—This field is optional and is used by some modeling software in planning studies. You do not need to specify a year in service when defining projects for real-time studies. (This field is not used by ResSim.)

5.2.5 Renaming a Reservoir

To rename a Reservoir:

- Using the **Reservoir Tool** or the **Pointer Tool**, right-click on the reservoir.
- From the context menu, select **Rename Reservoir**. This opens the Rename Reservoir dialog (Figure 5.11).

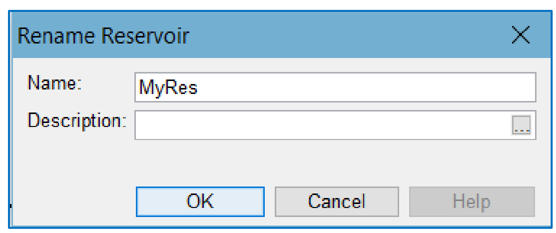


Figure 5.11 Rename Reservoir Dialog

- Enter a **Name** and an optional **Description**, then click **OK**.

- An "are you sure…?" message (Figure 5.12) will appear asking you to confirm that you really want to rename the reservoir. Choose:
  - **Yes** to complete the rename process.
  - **No** to abort the rename process.
  - **Cancel** to return to the Rename Reservoir dialog.

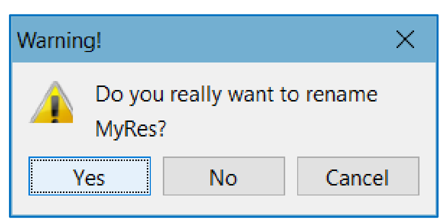


Figure 5.12 Rename Reservoir Query

5.2.6 Removing a Reservoir from a Configuration

To remove a Reservoir from the current Configuration,

- Using the **Reservoir Tool** or the **Pointer Tool**, *right-click* on the reservoir.
- From the context menu, select **Remove from Configuration**. This removes the reservoir from the current or active Configuration (the configuration that is currently displayed in the **Configuration** box on the Module bar).
- A **Confirm Removal** dialog will appear (Figure 5.13). Click **Yes** to confirm.

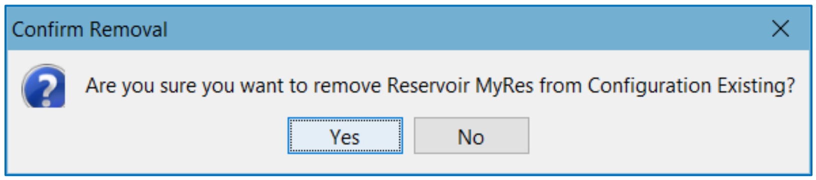


Figure 5.13 Confirm Removal of Reservoir

You can also remove a Reservoir from a Configuration by opening the **Configuration Editor**, then selecting **Edit Project List…** from the **Project** menu.

5.2.7 Deleting a Reservoir

To delete a reservoir configuration element and all data associated with it from the watershed:

- Using the **Reservoir** or **Pointer** **Tool**, right-click on the reservoir.
- From the context menu, select **Delete Reservoir**.
- A **Confirm Deletion** dialog will appear (Figure 5.14). Click **Yes** to confirm.

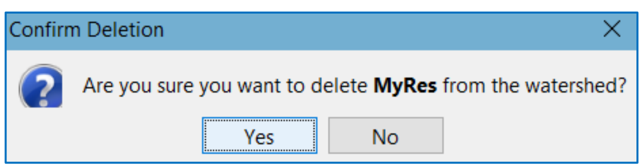


Figure 5.14 Confirm Deletion of Reservoir

- A Confirm Removal dialog (Figure 5.15) will appear next, asking if you also want to remove (delete) the computation points that were created when the reservoir was created. Click **Yes** to confirm deletion of the computation points. Click **No** to keep the computation points; the reservoir will still be deleted.

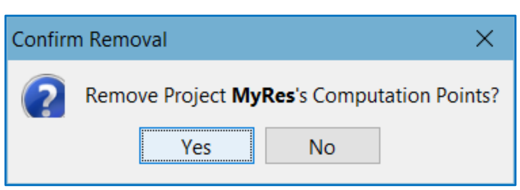


Figure 5.15 Confirm Removal of the Deleted Reservoir's Computation Points

5.2.8 Adding Configuration Notes for a Reservoir

Although ResSim does not use Configuration Notes, other models that share the watershed may. To open the Configuration **Notes** editor for a reservoir:

- Using the **Reservoir** or **Pointer** **Tool**, *right-click* on the reservoir.
- From the context menu, select **Configuration Notes**.
- In the **Configuration Notes** textbox of the **Notes** editor (Figure 5.16), enter information about the selected reservoir relevant to the current configuration.

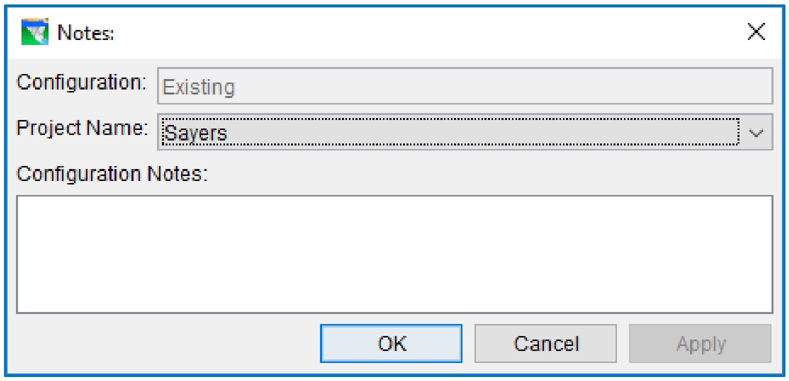


Figure 5.16 Configuration Notes for Reservoir

You can also access the configuration **Notes** editor by selecting **Notes…** from **Edit** menu of the **Reservoir Properties Editor**.

5.3 Diversions

5.3.1 Drawing a Diversion Element

To draw a diversion from a stream:

- Using the Diversion Tool , position your pointer at the location on the stream alignment where the diversion will occur (the "from" or *inflow* location) then press the **Ctrl** key and click.
  - If a computation point exists at (within the "snap" range of) that location, a **Computation Point Found** query dialog will ask if you want to connect the diversion to the **Existing Computation Point** or to a **New Computation Point** (Figure 5.17).

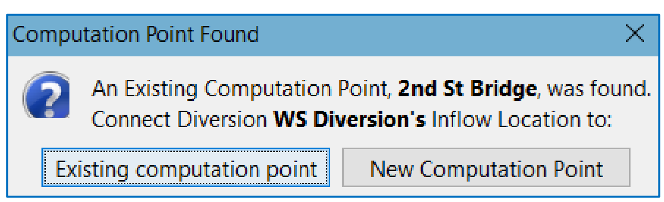


Figure 5.17 Connect to Existing Computation Point Query

- If you select **New Computation Point**, ResSim will create a new computation point on the stream alignment as near as possible to the location of your pointer when you "clicked" and will connect the diversion to it. The new computation point will be given a default name and will be located very near the existing computation point that was identified in the query. Don't forget to rename the new computation point(s).

- You can give shape to the diversion reach by continuing to hold down the **Ctrl** key and clicking as you move the pointer across the map display. With each **Ctrl+click**, a point will be added to the polyline that represents the diversion.
- To place the last point of the diversion, move the pointer to the location where the diverted water will go (the "to" or *outflow* location) then *release* the **Ctrl** key and click. Note: the outflow of the diversion can be placed at a location…

***on** the stream alignment*. This will make the diversion *connected* and indicates that the diverted water will remain within the river system. Like the inflow location, if a computation point exists at this location, you will be queried to be sure you want to connect the diversion to that computation point.

***not on** the stream alignment*. This will make the diversion *unconnected* and indicates that the diverted water will be lost from the river system.

A **Name New Diversion** dialog (Figure 5.18) will appear. Enter a **Name** and **Description** for your diversion and click **OK** to complete the creation of the diversion.

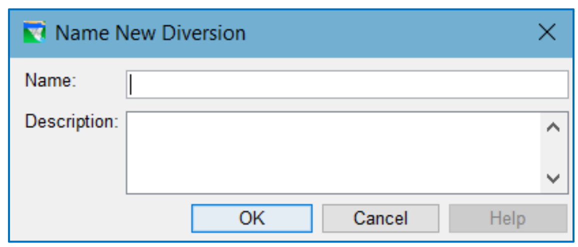


Figure 5.18 Name New Diversion

If the outflow of your diversion is *connected* to a computation point, the diversion will be drawn in the Map Display with a **blue** arrowhead. If the diversion is *unconnected*, the diversion will be drawn with a **black** arrowhead. Figure 5.19 illustrates two diversions—*Irrigation Diversion* is a connected diversion and *WS Diversion* is unconnected. If you cannot see a difference between the arrowheads of a connected and an unconnected diversion, you can change the draw properties of the diversions in the **Drawing Properties Editor.** Select **Drawing Properties…** from the **Edit** menu to open the editor. For more details, refer to Appendix B.

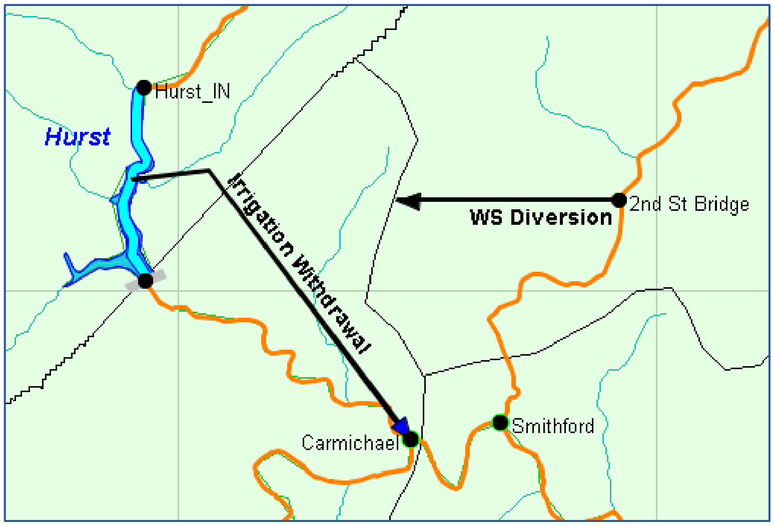


Figure 5.19 Connected and Unconnected Diversions

To draw a diversion from a reservoir:

- Using the Diversion Tool , position your pointer *along the storage reach* of a reservoir—away from the dam or inflow junctions, then press the **Ctrl** key and click. This will connect the diversion's inflow to the reservoir.
- Finish drawing the diversion as described above.

ResSim will not generate a computation point at the inflow of a diversion from a reservoir. The *Irrigation Withdrawal* diversion illustrated in Figure 5.19 is a diversion from a reservoir. Note that ResSim did not generate a computation point at the inflow of a diversion from a reservoir.

5.3.2 Disconnecting a Diversion

Connected diversions are diversions whose outflow point is connected to a computation point in the watershed schematic. To disconnect the outflow point of the diversion:

- Using the **Diversion Tool**  or the **Pointer Tool** , *right-click* on the diversion.
- Select **Disconnect Outflow** from the bottom of the diversion's context menu (Figure 5.20)

(NOTE: the inflow point of a diversion cannot be disconnected. If you connected the upstream end of a diversion to the wrong computation point, you must delete the diversion and re-create it.

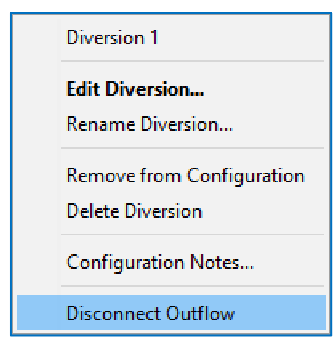


Figure 5.20 Diversion Context Menu

5.3.3 Reshaping a Diversion

A diversion is represented by a polyline that ends with an arrowhead. To reshape the diversion polyline:

- Using the **Diversion Tool**, *double-click* on the diversion. The polyline will highlight (turn yellow) and the vertices (points) that describe the polyline will display as tiny black dots as illustrated in Figure 5.21.

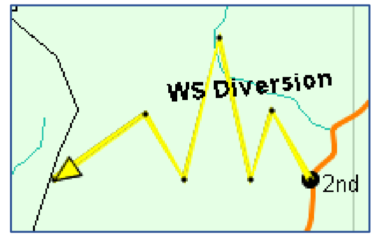


Figure 5.21 Highlighted Diversion

- Drag each of the black dots to a new position in the Map Display until the diversion polyline is shaped as you desire. If the diversion is unconnected, the only point that cannot be moved is the first one at the diversion inflow. If the diversion is connected, neither the first (inflow) point nor the last (outflow) point can be moved.
  - To **add** points to the polyline, hold down the **Ctrl** key and click anywhere along the diversion's polyline. A new black dot will appear where you clicked.
  - To **delete** points from the pool polygon, hold down both the **Ctrl** and **Shift** keys while clicking on a point you want to delete from the polygon. The point will disappear.

5.3.4 Editing Diversion Data

Use the Diversion Properties Editor to edit configuration data for the diversion elements. To open the **Diversion Properties Editor** (Figure 5.22):

- Using the **Diversion** or **Pointer** **Tool**, *right-click* on the diversion.
- Select **Edit Diversion** from the diversion's context menu.

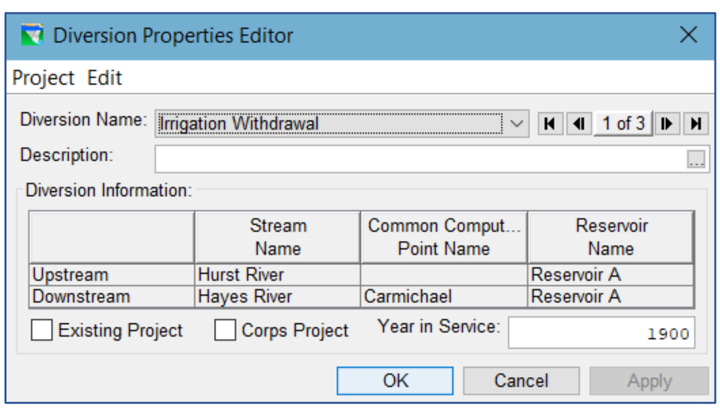


Figure 5.22 Diversion Editor

Use the **Diversion Properties Editor** to edit the description of the diversion, specify whether the diversion is an **Existing Project** and/or a **Corps Project**, and enter the **Year in Service** (when the diversion first began operating). You can also verify which schematic elements the diversion is connected to by reviewing the content of the **Diversion Information** table.

5.3.5 Renaming a Diversion

To rename a diversion:

- Using the **Diversion**  or **Pointer**  **Tool**, *right-click* on the diversion.
- Select **Rename Diversion…** from the diversion's context menu.
- Enter a new **Name** (and **Description**) for the diversion in the **Rename Diversion** dialog then click **OK**. (**Cancel** will abort the rename process.)
- An "are you sure" warning message will appear asking you to confirm that you really want to rename the diversion.
    - Select **Yes** to complete the rename process
    - Select **No** to abort the rename process
    - Select **Cancel** to return to the Rename Diversion dialog so that you can revise the new name for the diversion.

5.3.6 Removing a Diversion from a Configuration

To remove a diversion from the current configuration:

- Using the **Diversion**  or **Pointer**  **Tool**, *right-click* on the diversion.
- Select **Remove from Configuration** from the context menu
- The **Confirm Removal** dialog will appear (Figure 5.23). Click **Yes** to remove the diversion from the *active* configuration.

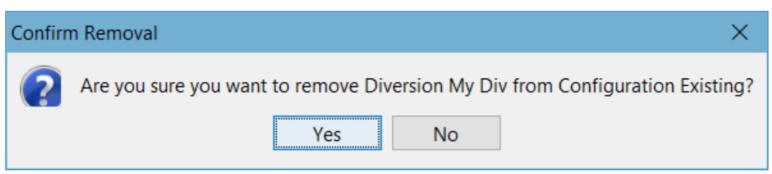
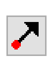


Figure 5.23 Confirm Removal of Diversion

You can also remove a diversion from a configuration by opening the **Configuration Editor** and using the **Project Selector** to remove the diversion. To open the **Project Selector**, select **Edit Project List…** from the **Project** menu of the **Configuration Editor**.

5.3.7 Deleting a Diversion

To delete a Diversion from the watershed:

- Using the **Diversion**  or **Pointer**  **Tool**, *right-click* on the diversion.
- From the context menu, select **Delete Diversion**. A **Confirm Deletion** dialog will open asking "Are you sure…?"

- Click **Yes** to delete the diversion and all data associated with the diversion from the watershed. Click **No** to cancel the delete.

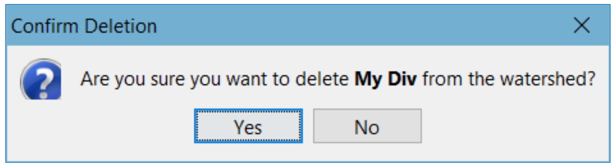


Figure 5.24 Confirm Deletion Dialog

5.3.8 Adding Configuration Notes for a Diversion

Although ResSim does not use Configuration Notes, other models that share the watershed may. To open the Configuration **Notes** editor for a diversion:

- Using the **Diversion** or **Pointer** **Tool**, *right-click* on the diversion.
- From the diversion's context menu, select **Configuration Notes**.
- In the **Configuration Notes** textbox of the **Notes** editor (Figure 5.25), enter information about the selected diversion relevant to the current configuration.

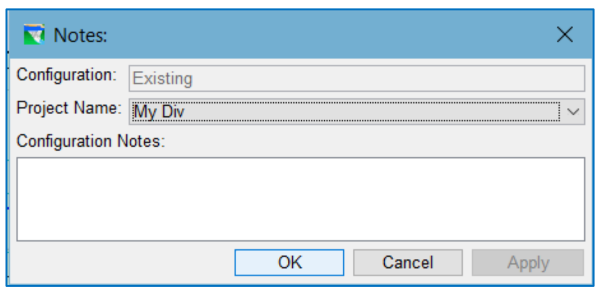


Figure 5.25 Configuration Notes for Diversion

You can also access the diversion's configuration **Notes** editor by selecting **Notes…** from **Edit** menu of the **Diversion Properties Editor**.

5.4 Drawing a Levee

A levee is (typically) an earthen barrier built along a stream to provide protection from flooding. A levee can reduce flood damage by preventing flood stages from reaching a potential damage area. Although levees are not used by ResSim, it may be useful information for the modeler when developing routing information.

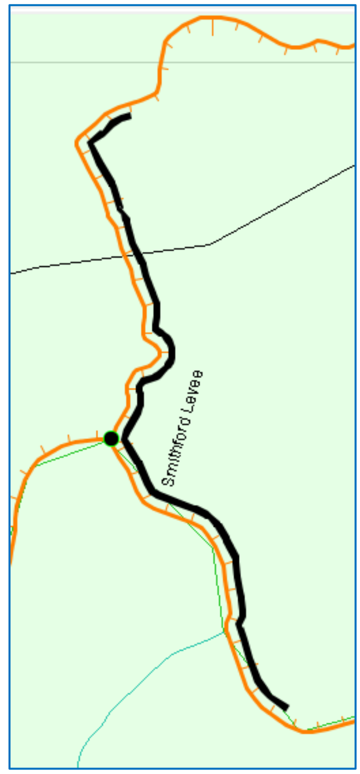


Figure 5.26 Levee

To add a levee to the watershed:

- Using elect the **Levee Tool** , move your cursor to the location of the *upstream* end of the levee along the Stream Alignment.
- While holding down the **Ctrl** key, click on the stream alignment to select the *upstream* end of the levee.
- Release the **Ctrl** key, move the cursor to the location of the downstream end of the levee and click again. A **Name** dialog will appear.
- Enter a **Name** and option **Description** for your levee, then click **OK** to complete the levee creation process. The levee will be drawn in the Map Display as a thick black polyline along one side of the stream alignment from the upstream to downstream points that you identified (Figure 5.26).
- You can use the **Levee Editor** to reposition the levee (Figure 5.27).

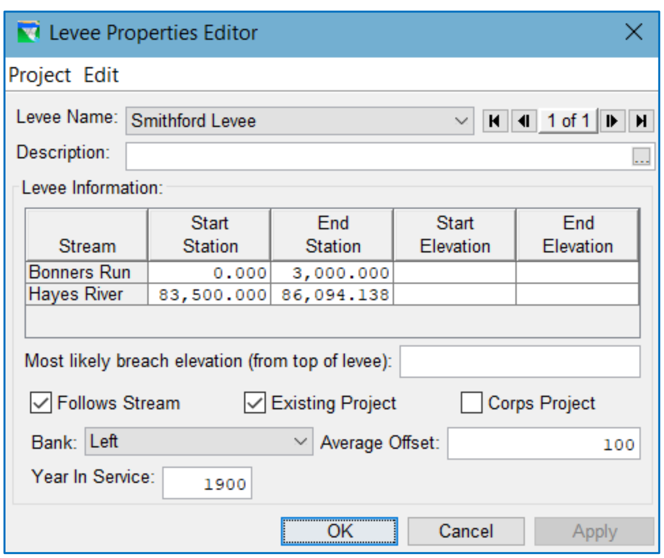


Figure 5.27 Levee Properties Editor

5.5 Drawing a Channel Modification

A channel modification is typically an attempt to improve conveyance in the natural river channel. The modification can be implemented in a number of ways, including debris and vegetation removal, armoring or lining of the channel, and even straightening of the channel. Although channel modifications are not used by ResSim, the presence of one may offer useful information to the modeler when developing routing information.

To add a Channel Modification to the watershed:

- Using the **Channel Modification** Tool, move your cursor to the location of the *upstream* end of the channel modification along the Stream Alignment.
- While holding down the **Ctrl** key, click on the stream alignment to select the *upstream* end of the channel modification.
- Release the **Ctrl** key, move the cursor to the location of the downstream end of the channel modification, and click again. A **Name** dialog will appear.
- Enter a **Name** and option **Description** for your channel modification, then click **OK** to complete the channel modification creation process. The channel modification will be drawn in the Map Display as a thick black polyline atop the Stream Alignment from the upstream to downstream points that you identified (Figure 5.28).

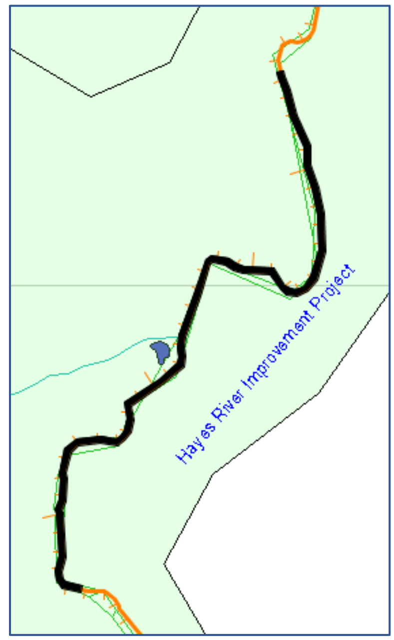


Figure 5.28 Channel Modification

- You can access the **Channel Modification Properties Editor** (Figure 5.29) by right-clicking on the channel modification in the Map Display and selecting Edit from the context menu. Useful options for editing the channel modification include revising the stationing of the upstream and downstream ends of the project and the position of the label relative to the midpoint of the project.

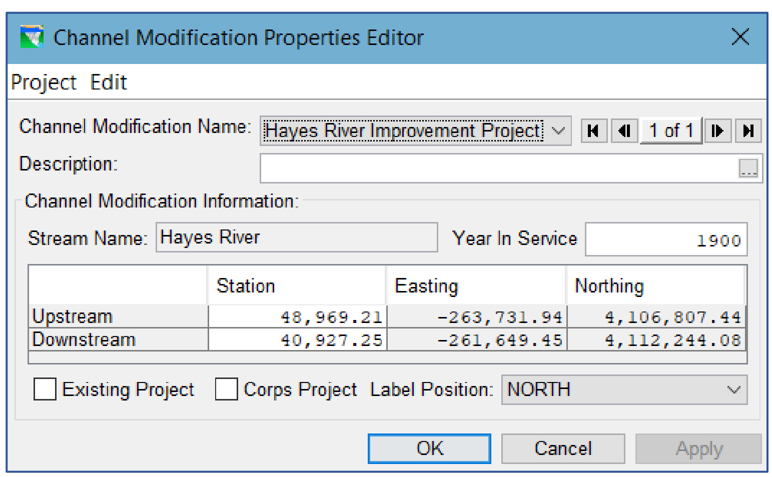


Figure 5.29 Channel Modification Properties Editor

5.6 Drawing an Off-Channel Storage Area

An off-channel storage area is a pond or ineffective flow area where water may collect. These project elements are not used by ResSim but that may be used by another model that shares the watershed (such as RAS). To model a constructed off-channel reservoir that ResSim *can* use, use a reservoir element, even if you have to create an off-channel stream in the Stream Alignment to place it on.

To add an Off-Channel Storage Area to the watershed:

- Using the Off-**Channel Storage Tool** , hold down the **Ctrl** key and click the location in the map where you want to start drawing the off-channel storage area.
- Continue to hold the **Ctrl** key while you click to add additional vertex points of the polygon that represents the off-channel storage area.
- Before placing the last vertex of the polygon, release the **Ctrl** key, then click to place the last point. A Name … dialog will appear.
- Enter a **Name** and optional **Description** for your off-channel storage area, then click **OK** to complete the off-channel storage area creation process. The off-channel storage area will be drawn in the Map Display as a closed polygon connecting the vertex points you placed (Figure 5.30).

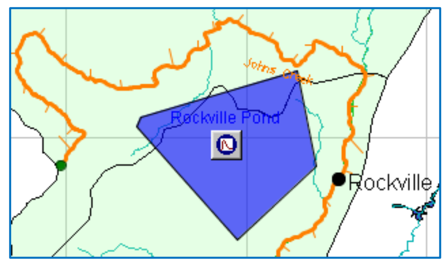


Figure 5.30 Channel Modification

5.7 Drawing an "Other" Project

Other project is a catch-all name for a man-make features in the watershed related to the movement of water. Possible *other projects* might include pump stations, in-line spillways, or even an ecosystem restoration project. ResSim does not use *other projects* but you should consider carefully whether the *other project* you are considering adding to the watershed could be modeled with a reservoir or a diversion—such as a pump station.

To add an "Other" Project to the watershed:

- Using the **Other Project Tool** , **Ctrl**-click at the location in the Map Display where you want the *other project* to appear. A Name… dialog will appear.
- Enter a **Name** and optional **Description** for your *other project*, then click **OK** to complete the *other project* creation process.
- The *other project* will be drawn in the Map Display as a computation point with an **Other Project** icon  attached (Figure 5.31).

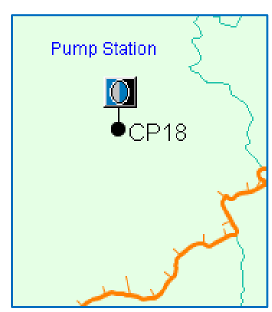


Figure 5.31 Other Project

- Use the **Computation Point** or **Pointer** tool to rename the computation point associated with the *other project*.
- You can use the **Other Project Properties Editor** to reposition the **Other Project** icon and its associated computation point (Figure 5.32).

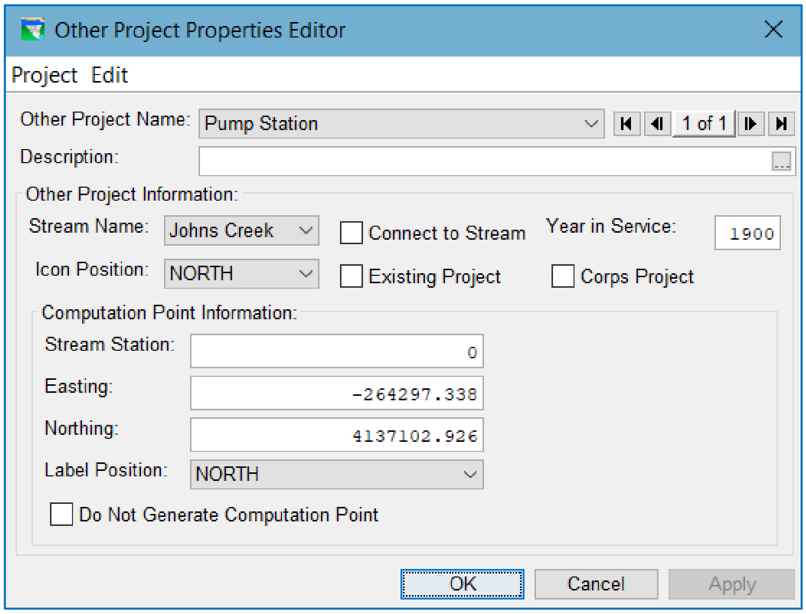


Figure 5.32 Other Project Properties Editor

5.8 Configuring Project Display Properties

You can configure the appearance of Reservoirs, Levees, Diversions, and Computation Points in the **Study Layer** of your watershed using the **Drawing Properties Editor** (Figure 5.33).

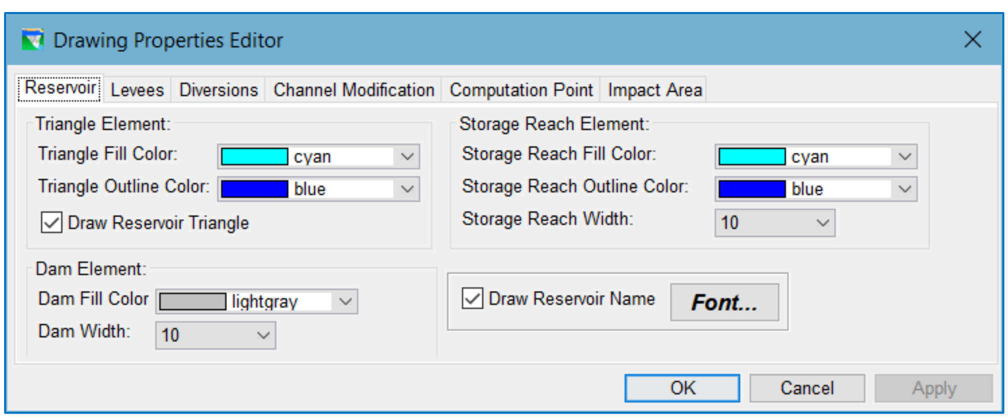


Figure 5.33 Drawing Properties Editor

To access the **Drawing Properties Editor**:

- Select **Drawing Properties...** from the **Edit** menu of the **Watershed Setup** module, or

- Select Properties from the context menu of the Study layer in the **Layer Selector**.

See Appendix B for details on how to change the drawing properties for your configuration elements.

Chapter 6
Developing a Reservoir Network

A *reservoir network* is the principal component of your ResSim model. It contains the model schematic, its elements, and all their physical and operational data. The model schematic elements include *reservoirs*, *routing reaches*, *junctions*, *diversions*, and *diverted outlets*.

Each model schematic element has its purpose in the reservoir network.

Reservoirs and **Diversions** are the operational elements—they make decisions about the movement of water through the network.

Junctions are connectors—they connect all other schematic elements to one another. Junctions also provide connection to inflow data.

Reaches route flow through the network, from one junction to another.

Diverted Outlets are a hybrid of a diversion and a reservoir outlet group. As an outlet group, they are effectively a sub-element of a reservoir. As a diversion, they take the water allocated to them and deliver it to wherever the diversion is connected.

This chapter provides an overview of the **Reservoir Network** module's user interface and describes the processes for creating a new reservoir network and completing its connectivity by adding reaches and other model schematic elements as needed.

Once you have established your reservoir network and its schematic, you will need to complete its definition by specifying the Junction, Reach, and Diversion data (Chapter 7). You will then use the Reservoir Editor (Chapter 8) to defining the physical (Chapter 9) and operational (Chapters 10, 11, and 12) components of the Reservoirs. You may also wish to develop Reservoir Systems (Chapter 13) in your network.

After completing the definition of your reservoir network, you can start creating alternatives (Chapter 14). And, with your network(s) and alternative(s) defined, you will be ready to create and run Simulations (Chapter 15).

After running a simulation, you can review results using default or custom Plots, Tables (Chapter 16), and Reports (Chapter 17). You can also develop scripts to post-process results or automate the building of plots, tables, and reports (Chapter 18).

6.1 The Reservoir Network Module

The **Reservoir Network** module of ResSim is effectively *the editor of a reservoir network.* Almost every feature and tool in this module is involved in creating, editing, or managing some aspect or component of a reservoir network. This Section describes the various user interface components and tools (Figure 6.1) provided in the **Reservoir Network** module.

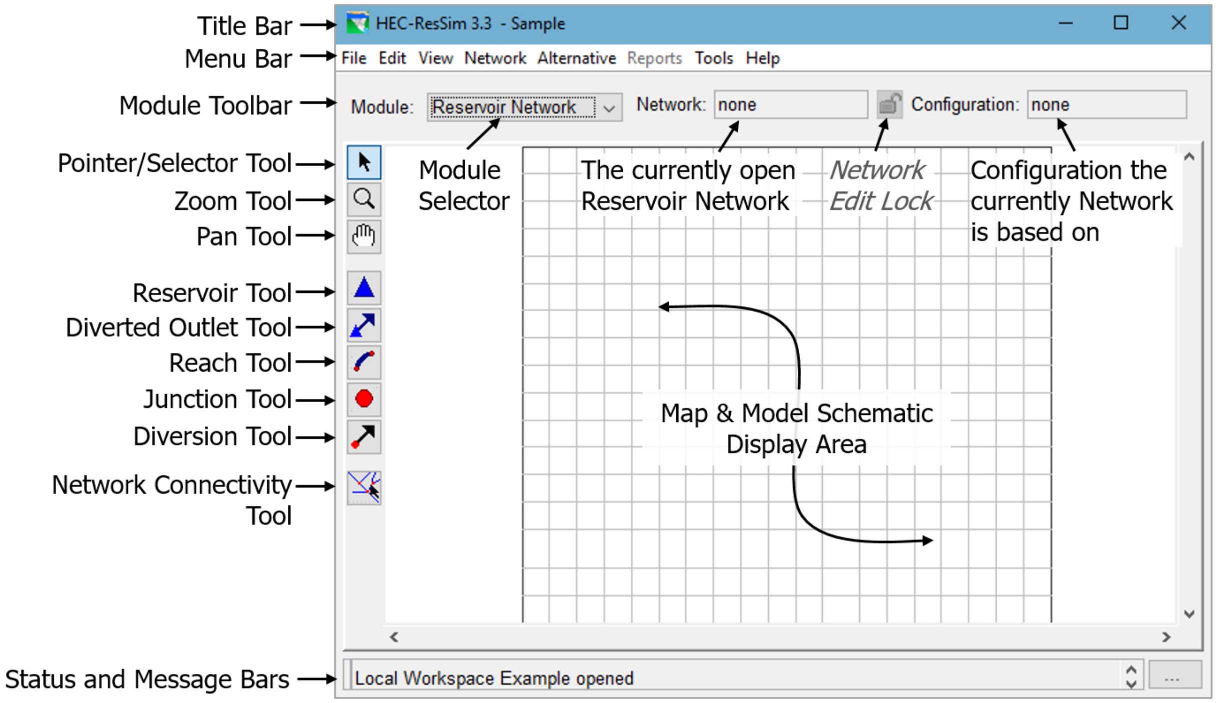


Figure 6.1 Reservoir Network Module

6.1.1 Menu Bar

The following is an overview of the menus unique to the **Reservoir Network** module which provide features for creating and editing reservoir networks and their elements. Detailed descriptions of the options in these menus will be provided in the context of specific network development tasks later in this and subsequent chapters. Refer to Chapter 2 for information about the **File**, **View**, **Tools,** and **Help** menus, which are common to all three modules.

Edit—The **Edit** menu (Figure 6.2) provides access to the ResSim editors for **Reservoirs**, **Reaches**, **Junctions**, **Diversions**, **Reservoir Systems**, **State Variables**, and **Water Account Sets**, as well as the **Wizard** to **Import Element Properties…** Also in the menu is the **Allow Network Editing** option to *lock* the module for editing. Since the **Reservoir Network** module is now used only by ResSim, the multi-user locking requirement is no longer necessary; so, the lock is now set by default whenever there is a network open in the module. All edit locks will be removed in the next version of ResSim.

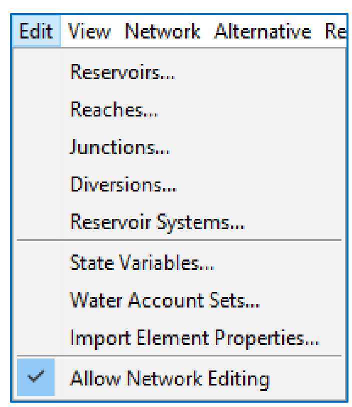


Figure 6.2 Edit Menu

Network—The **Network** menu (Figure 6.3) is unique to the Reservoir Network Module. You can think of it as a secondary File menu since it provides most of the same functions but applied specifically to reservoir networks. The Network menu options allows you to create a **New** network, **Open** an existing network, **Edit** the description of the current network, **Save**, copy (**Save As**), and **Rename** the current network, and **Delete Networks**. **Rename** a network. You can also **Update Network from Configuration** and **Import Networks**.

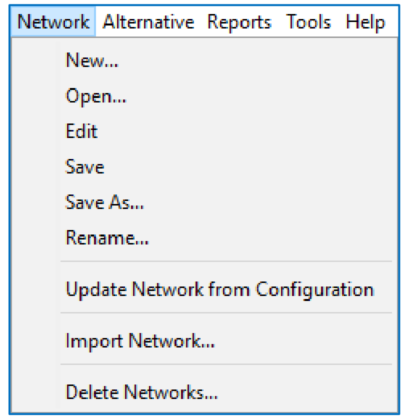


Figure 6.3 Network Menu

Alternative—The only option, **Edit**, in the **Alternative** menu (Figure 6.4) opens to the **Alternative Editor**. This menu is also available in the **Simulation** module.

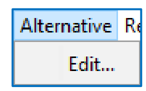


Figure 6.4 Alternative Menu

Reports—The **Reports** menu (Figure 6.5) provides access to a variety of reports that highlight different aspects of reservoir networks; these include the **Reservoir List**, **Reach List**, **Junction List**, and **Diversion List** as well as two **Advanced** reports—the **Network Connectivity** report (for **All Elements** or for **Selected Elements**) and the **Node List**. The last option, **Alternative Input**, lets you select an alternative and generate a structured listing of the features that make up the selected alternative and its associated network.

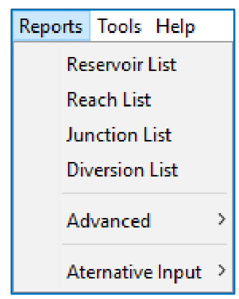


Figure 6.5 Reports Menu

6.1.2 The Module Toolbar

In the **Reservoir Network** module, the Module Toolbar (Figure 6.6) contains the Module selection as well as two fields that identify the currently open **Network** and the **Configuration** the open network is based upon.

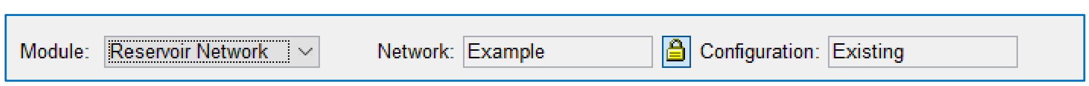


Figure 6.6 Network Module—Module Toolbar

Between the **Network** and **Configuration** fields is an **Edit Lock**  button. Since the multi-user locking requirement is no longer necessary, the lock is now set by default whenever there is a network open in the **Reservoir Network** module. All edit locks will be removed in a future version of ResSim.

6.1.3 Map Display Area

In the **Reservoir Network** module, the Map Display area shows the model schematic of the open reservoir network, along with the background maps and the stream alignment you established in the Watershed Setup Module.

By using the Map Tools to access context menus directly from the schematic in the Map Display, you can open data editors as well as Rename, Delete, and Import Properties for the elements in your reservoir network.

6.1.4 Map Tools

The **Map Tools** allow you to create and edit the model schematic elements from in the Map Display area. The **Reservoir Network** module has three *standard* Map Tools (Pointer, Zoom, and Pan), five network (model schematic) element tools (Reservoir, Diverted Outlet, Reach, Junction, and Diversion), and a tool for visualizing the connectivity of your reservoir network.

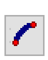

Pointer Tool—right-click with the **Pointer Tool** to select and access a context menu for any schematic element within your map display. The Pointer Tool is available in all ResSim modules.

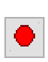

Zoom Tool—the **Zoom Tool** allows you to zoom in and out of the display area in all modules. To zoom in, hold the left mouse button down and outline the area you want to enlarge. To zoom out, click the right mouse button. Zooming out using the right click button zooms out by a factor of two, positioning the clicked location at the center of the screen. The Zoom Tool is available in all ResSim modules.


Pan Tool—after you have zoomed in with the Zoom Tool, you can use the Pan Tool to drag the content of the Map Display, effectively scrolling the viewable portion of the watershed. The Pan Tool is available in all ResSim modules.


Reservoir Tool—with the Reservoir Tool, you can add reservoirs to your reservoir network. You can also edit reservoir properties, rename, and delete reservoirs using commands in the context menu.


Diverted Outlet Tool—the Diverted Outlet Tool allows you to create a diverted outlet from a reservoir in your network. The context menu for this tool provides access to the Reservoir Editor and allows you to rename and delete a diverted outlet.


Reach Tool—use the Reach Tool to draw routing reaches (from *upstream* to *downstream*) to connect the junctions on the stream alignment. You can also edit reach properties, rename, break, and delete reaches using commands in the context menu.


Junction Tool—the Junction Tool allows you to manually insert junctions in your reservoir network. The context menu for this tool allows you to edit junction properties, rename and delete junctions.

 **Diversion Tool**—with the Diversion Tool you can add diversions to your reservoir network. You can also edit diversion properties, rename, delete, and disconnect outflow of diversion reaches using commands in the context menu.

 **Network Connectivity Tool**—using the Network Connectivity Tool you can click on any element in the map display area to see which elements are connected (connected elements will be highlighted). If you right-click on an element, you can choose to show the upstream or downstream connectivity.

6.2 Creating and Managing Reservoir Networks

ResSim provides three ways to create a reservoir network:

1. create a *new* network,
2. make a *copy* of an existing network, and
3. *import* a network from another watershed.

But, before you start creating reservoir networks, it is important to understand how they relate to watershed configurations and the watershed schematic(s) you created in the Watershed Setup module.

ResSim was designed to fully implement the *shared schematic* concepts that are the basis of the **Watershed Setup** module and watershed configurations. To that end, ResSim's reservoir networks were designed to be created using a watershed configuration as a template. *Using a configuration as a template* means that ResSim will create model elements in the reservoir network for each watershed schematic element it recognizes in the selected configuration. Thus…

For each **reservoir** in the configuration, ResSim will create a **reservoir** with the same name and position on the stream alignment in the new network.

For each **diversion** ***from a stream*** in the configuration, ResSim will create a **diversion** with the same name, shape, location, and connectivity in the new network.

For each **diversion** ***from a reservoir*** in the configuration, ResSim will create a **diverted outlet** the associated reservoir in the new network. The diverted outlet will have the same name, shape, location, and connectivity as the diversion in the selected configuration.

In addition, for each **computation point** defined in the watershed schematic, ResSim will create **junctions** in the reservoir network when a reservoir network is created based on a configuration.

Each of the *network* elements that were created based on a *watershed* schematic or configuration element stores a reference or link to the element it was based on. By maintaining this information, a network can be updated to reflect any changes that have been made to the watershed schematic or configuration since the network was first created.

Once the preliminary model schematic for a *new* network appears in the Map Display, you will need to add routing reaches (and possibly other network elements) to complete the connectivity of your reservoir network schematic.

6.2.1 Creating a New Network

When you create a new reservoir network, you can (and should) identify which *projects* will be included in your network by selecting an appropriate watershed configuration. After creating the network (based on a configuration), you *will* need to add additional elements to complete the connectivity and to represent any additional features that were not identified by the configuration.

To create a new reservoir network:

- From the **Network** menu, select **New**.

  The **Create New Reservoir Network** dialog will appear (Figure 6.7).

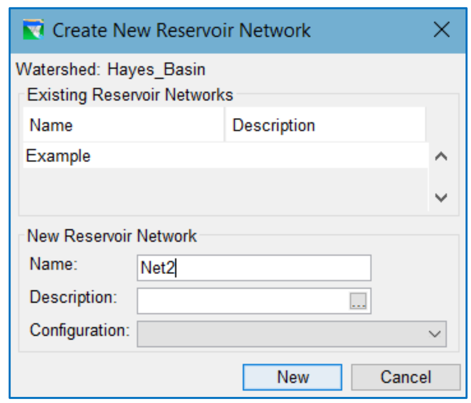


Figure 6.7 Create New Reservoir Network

- Enter a **Name** and a **Description** for the network.
- From the **Configuration** list, select the configuration on which this network will be based.
- Click **New**.

  If no configuration was selected, a warning dialog will ask you to confirm that the network will not be based on a configuration (Figure 6.8).

  - Click **Yes** to proceed without a configuration. If you do not use a configuration when creating your network, you will have to create *all* the model schematic elements for your network.
  - Click **No** to return to the **Create New Reservoir Network** dialog so that you can select a configuration.

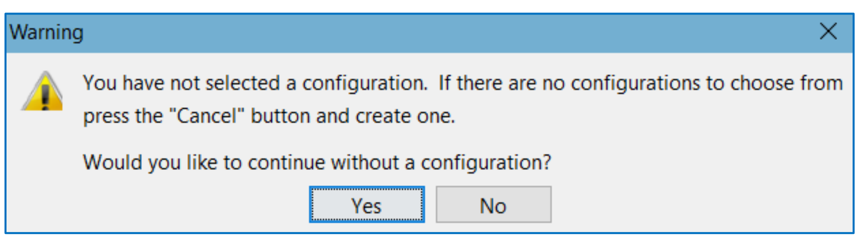


Figure 6.8 Creating a New Network with no Configuration Selected

After you (and ResSim) complete the process of creating a new network, you will see the name of your new network and the configuration it was based on displayed below the Module Toolbar. Also, the reservoirs, junctions, and diversions that ResSim created in your network based on the configuration will appear in the Map Display area.

Note: A reservoir network *can* be created without specifying or selecting a configuration. If you do so, you will have to add each model schematic element into your network by drawing them, one by one, in the Map Display area. And, networks created without a configuration cannot be updated based on changes in made to a configuration nor can a configuration be applied to or imposed on them.

6.2.2 Importing a Reservoir Network

The **Import Network Wizard** guides you through the process of importing a reservoir network from another watershed. For each step of the process, the Import Network Wizard displays a different dialog panel. Each panel identifies the current Step in the title bar and tells you what the objective of the step is with a label in the upper left corner of the panel (below the title bar). The process involves six steps:

1. **Select the Watershed**—select the *source* watershed that contains the network you want to import into your current watershed.
2. Select the Network to Import Elements from—select the *source* network.
3. **Set New Network Name**—the process of importing a network creates a new network in your current watershed, so you are given the option to give the *new* network a new name and description.
4. **Assign Stream Names**—associate streams in your current watershed's stream alignment to the schematic elements in the network you are importing.
5. **Resolve Network Computation Points**—associate computation points in your current watershed schematic to the junctions in the source network.
6. Review the **Import Summary** and **Finish**—this step shows you a listing of the current stream or computation point associations you made for the various elements of the network you are importing. It also shows what elements, if any, have not been associated to elements in your current watershed. Be sure to review the summary carefully.

To import a network:

- Select **Import Network…** from the **Network** menu.
- The **Import Network Wizard** will open and display the **Step 1 of 6** panel (Figure 6.9). This panel shows two tables.

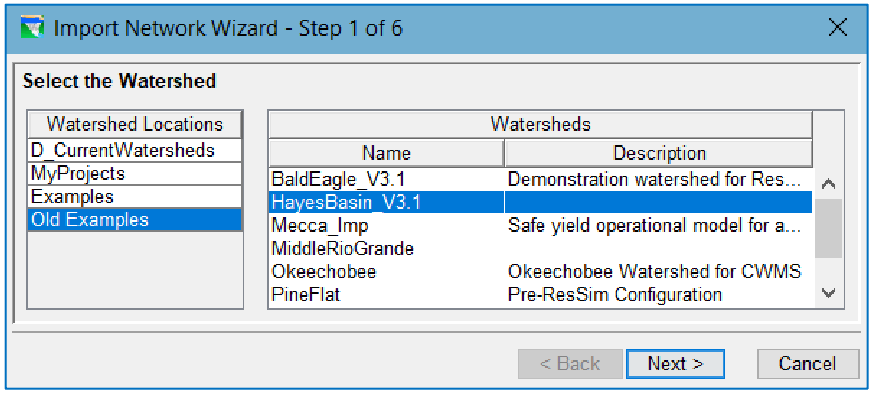


Figure 6.9 Import Network Wizard—Step 1—Select the Watershed

- The first table lists the **Watershed Locations** (shortcuts) you currently have defined in ResSim. Select the appropriate watershed location.
- The second table lists the **Watersheds** that ResSim found at the selected watershed location. Select the watershed that contains the network you want to import.

Note: The Import Network Wizard is only capable of identifying watersheds in known ***watershed locations***. If you do not have a watershed location (or *shortcut*) already defined in ResSim for the location where your source watershed resides, you ***must*** create one before you can proceed with the network import process. Instructions for creating a shortcut/watershed location are provided in Chapter 3, Section 3.2.1 Defining a Watershed Location and in Appendix A.

- Click **Next>** to proceed to the next step (or **Cancel** to abort the import process and exit the Import Wizard).
- The **Import Network Wizard** will display the **Step 2 of 6** panel (Figure 6.10). The **Select the Network to Import Elements from** panel shows the name of the *source* **Watershed** you selected and a list of the **Available Networks** it contains.

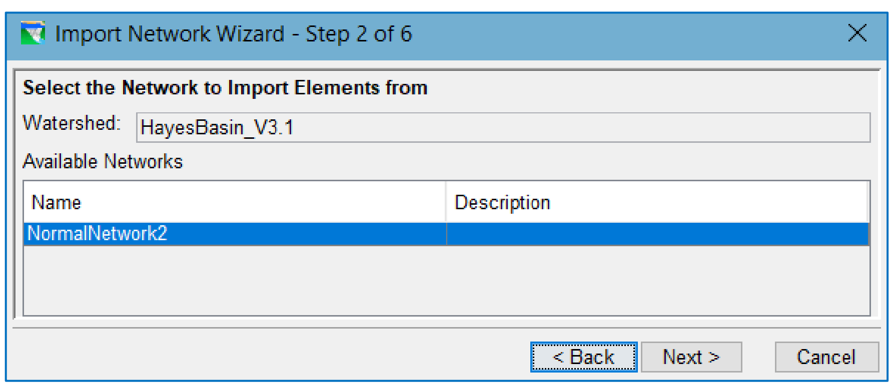


Figure 6.10 Import Network Wizard—Step 2—Select the Network to Import Elements from

- Select the *source* network (*from which* you would like to import all its elements).
- Click **Next>** to proceed to the next step or **<Back** to return to the previous step.

- The **Import Network Wizard** will display the **Step 3 of 6** panel (Figure 6.11). The top half of the **Set New Network Name and Description** panel shows the name of the *current* watershed and a list of the networks that exist in it. The bottom half has two textboxes which contain the name and description of the *source* network.

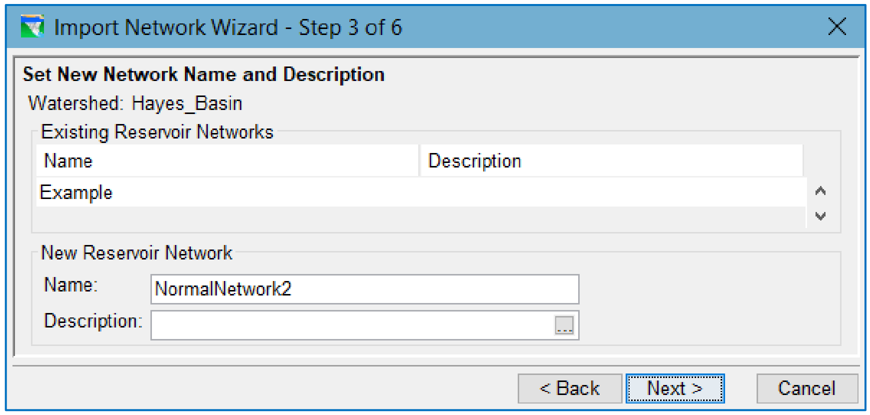


Figure 6.11 Import Network Wizard—Step 3—Set New Network Name and Description

- You can let the **New** (imported) **Reservoir Network** have the same name and description as its source or you can enter a new **Name** and **Description**.
- Click **Next>** to proceed to the next step (or **<Back** to return to the previous step.)
- The **Import Network Wizard** will display the **Step 4 of 6** panel (Figure 6.12). The **Assign Stream Name** panel shows the name of the current **Watershed** as well as the name you specified for the **New Network** and the name of the **Import From** (source) **Network**. This panel also contains a table showing a list of the network **Elements** to be **Imported**, their **Element Type**, and the (**From**) **Stream** they are associated with in the *source* watershed.

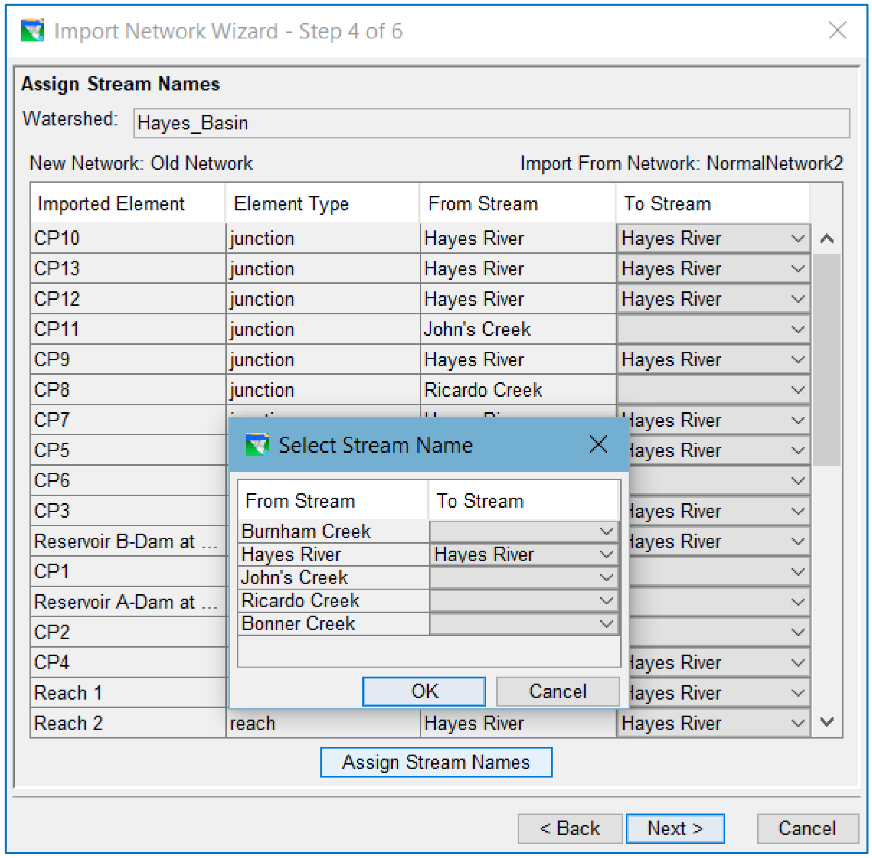


Figure 6.12 Import Network Wizard—Step 4—Assign Stream Names

- In the last column of the table, you will associate a stream in the current watershed to each element in the list. If any of the streams in the current watershed have the same name as a stream in the source watershed, ResSim will assume they are the same stream and will fill in those cells of the table for you. You have two options for assigning streams to each element:
    1. Assign the streams, one at a time, to each element in the list by selecting an appropriate steam from the drop-down list in each cell. *Or...*
    2. Assign the streams all at once by clicking the **Assign Stream Names** button, then, for each stream listed in the **From Stream** column in the **Select Stream Name** dialog, select a stream from your current watershed from the selection list in each cell in the *To Stream* column. When done, click **OK**. The importer will use the associations you made to complete the **To Stream** column of the **Assign Stream Names** table.
- Click **Next>** to proceed to the next step (or **<Back** to return to the previous step).
- The **Import Network Wizard** will display the **Step 5 of 6** panel (Figure 6.13). The **Resolve Network Computation Points** panel starts by identifying the current watershed which is followed by a selection list of the available configurations. Select the configuration that you want the new (imported) network to be associated with.

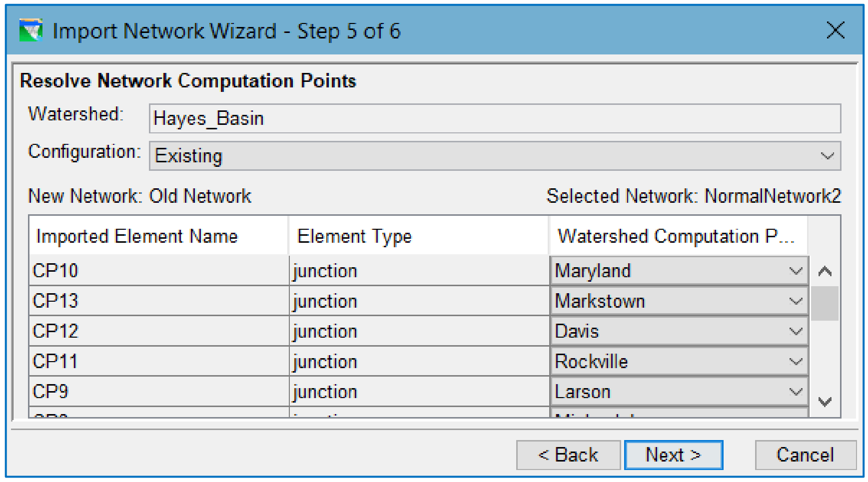


Figure 6.13 Import Network Wizard—Step 5—Resolve Network Computation Points

- The **Resolve Network Computation Points** panel also includes a table containing a list of the model schematic elements to be imported from the source network. Your objective is to associate each element with a watershed schematic element from the selected configuration in the current watershed. Use the drop-down list in each cell to make your selections. This can be a tedious process so take your time.
- Click **Next>** to proceed to the next step (or **<Back** to return to the previous step).
- The **Import Network Wizard** will display the **Step 6 of 6** panel (Figure 6.14). This Import Summary panel
- Review the **Import Summary** carefully. This is your last chance to double-check all the work you put in mapping (associating) the imported elements to streams and configurations.

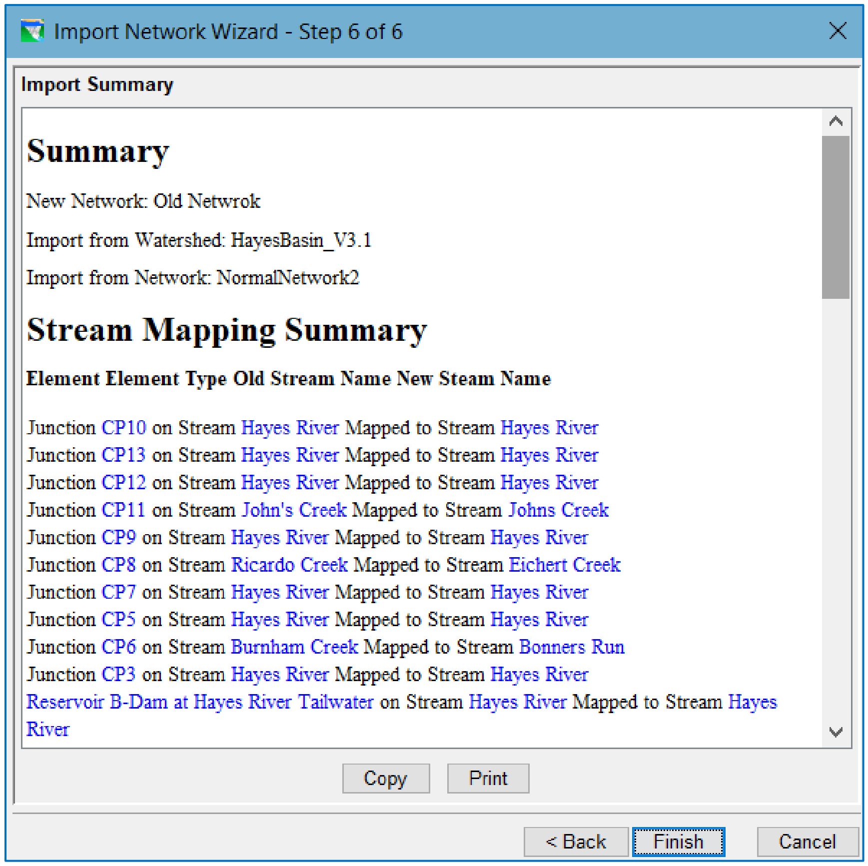


Figure 6.14 Import Network Wizard—Step 6—Import Summary

- Select **Finish** to carry out the Network import, or select **Back** to make changes in previous steps. A confirmation dialog will open…
  - Select **Yes** in the **Continue with Import** dialog (Figure 6.15) to perform the import. *Or…*
  - Select **No** to return to **Step 6 of 6** of the **Import Network Wizard**. From there, you can go **<Back** to a previous step to revise one or more entries or **Cancel** the process and exit the Wizard.

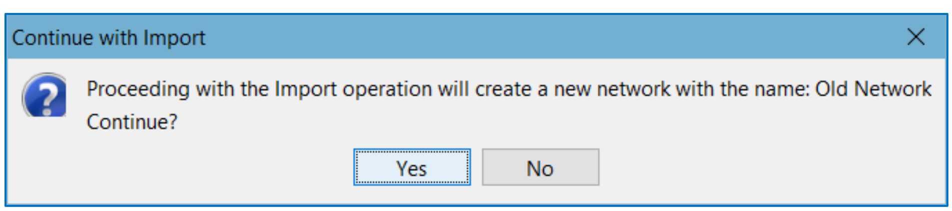


Figure 6.15 Continue with Import

- Review the imported network carefully. The Importer is good but not perfect. Areas for careful review include:
  - Any rule, constraint, or condition that is a function of a time series (model, state, or external variable)—the links may look okay, but it wouldn't hurt to reset them.

- o Downstream connectivity—any rule or feature that identifies another element in its setup should be reviewed. These features include downstream control rules, tandem rules, pump rules, and tailwater definitions.
- o Any state variable or scripted rule—scripts often reference other model elements, but the importer does not have the ability to revise the names of those elements *in the scripts* during the import process.

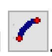
6.3 Completing the Network Connectivity

Routing reaches are elements that establish the connectivity of the reservoir network schematic. By adding a routing reach between two junctions, you can connect the reservoir network so that water will transverse through the routing reach from *upstream* to *downstream*.

6.3.1 Drawing Routing Reaches

Routing Reaches are drawn from *upstream* to *downstream*. You connect two adjacent junctions (typically, computation points added to the network's Configuration in the Watershed Setup Module) by drawing a routing reach.

To draw routing reaches:

- Select the **Reach Tool** .
- Point to the *upstream* junction at the upstream end of the reach.
- Hold down the **Ctrl** key and click on the junction to start the *upstream* end of the reach.
- Release the **Ctrl** key and move the mouse pointer along the stream alignment, then click on the *downstream* junction.

Routing reaches automatically conform to the stream alignment. You can connect routing reaches to existing junctions; otherwise, if you begin and/or end a reach elsewhere on the stream alignment, ResSim will create new junctions at both/either end of the reach.

ResSim will automatically constrain a new reach between existing junctions (e.g., will not allow a reach to be drawn past an existing junction). If the reach appears to draw past a junction, then the junction is not on the stream you think it's on. This can occur if the computation point (that the junction is based on) is not initially placed on the appropriate stream. To solve this situation, go back to the Watershed Setup Module and revise the location of the computation point (you may need to delete and recreate the computation point to be on the appropriate stream). After saving your watershed configuration, change to the Reservoir Network Module and select **Update Network from Configuration** from the **Network** menu. See Section 6.4.6, "Updating a Reservoir Network" for additional information for updating networks.

Additionally, you may not connect tributaries to the middle of a reach; tributaries must connect to a junction.

6.3.2 Renaming Routing Reaches

ResSim automatically names routing reaches according to the names of the upstream and downstream junctions. To rename a reach component:

- Using the **Reach Tool** , *right-click* on the routing reach to be renamed.
- Select **Rename** from the context menu. The **Rename Reach** dialog will appear (Figure 6.16), allowing you to type in a new name.

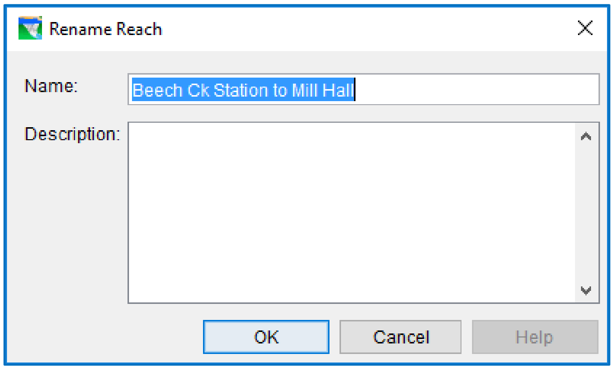
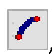


Figure 6.16 Rename Reach

The new name will now appear as a label in the map display.

6.3.3 Deleting Routing Reaches

To delete routing reaches:

- Using the **Reach Tool** , *right-click* on the routing reach to be deleted.
- Select **Delete** from the context menu. A **Confirm Delete** dialog will appear.
- Click **Yes** if you are sure about the deletion.

The deleted routing reach will no longer appear in the map display.

6.4 Managing Reservoir Networks

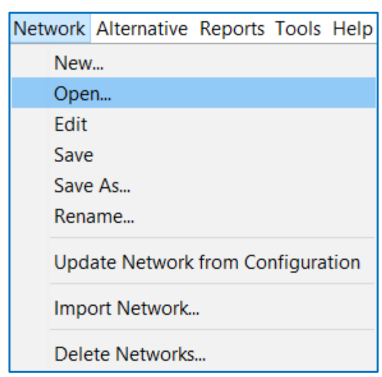


Figure 6.17 Network Menu

The tools for managing your reservoir networks are provided through the **Network** menu of the module (Figure 6.17).

Although you will rarely, if ever, need to work directly with the files that make up a ResSim watershed, it may be useful to know how the reservoir network data is stored.

Reservoir networks are not stored to a single file in the watershed; they are, in fact, stored to 8 separate files. The primary network file *network*.rsys file—where *network* is the name you gave the reservoir network. The .rsys file contains the definition of all the network objects that make up the reservoir, their connectivity, and most of the data that describes them. The second file is the *network*.dss file which stores all of the tabular data that describes the physical data of the reservoirs and any rating curves for the junctions.

6.4.1 Opening an Existing Reservoir Network

To open an existing Reservoir Network for editing:

- Select **Open** from the **Network** menu. The **Open Reservoir Network** dialog will appear (Figure 6.18).
- Select the **Existing Reservoir Network** you want to open and click **Open**.

  The **Open Reservoir Network** dialog will close, and you will see the name of your network and its configuration displayed in the Module Toolbar.

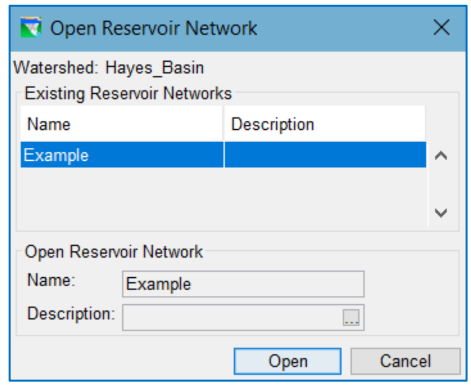


Figure 6.18 Open Reservoir Network

6.4.2 Editing a Network

Since the whole **Reservoir Network** module is effectively the editor for reservoir networks, the **Edit** option in the **Network** menu does not open an editor with a wide variety of parameters for you to define or change. Instead, it opens a small dialog (Figure 6.19) whose sole purpose is to allow you to edit the **Description** of the network. This may seem trivial, but experienced modelers, as well as the developers of ResSim, consider the descriptions for most elements of a model to be important and significant. Descriptions allow you to internally document each aspect of the model as it is created and as it evolves. These descriptions will be invaluable to you and to anyone else who wants to use or review your model.

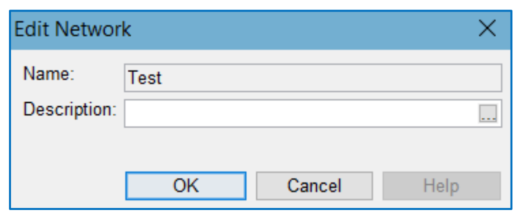


Figure 6.19 Edit Network Dialog

6.4.3 Saving a Network

ResSim maintains a lot of information about your network in memory. And each time you make any changes to the components of your network and click **OK** or **Apply** in an editor, those changes are stored in memory. And, ResSim will give you a visual clue when the network in memory is different from what is stored in files in the watershed. That visual clue is an asterisk * at the end of the **Network** name in the Module Toolbar (Figure 6.20).

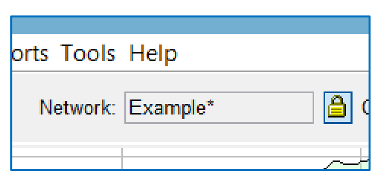


Figure 6.20 Network has changes

To save the open network to disk:

- Select **Save** from the **Network** menu.

Since ResSim knows when changes have been made to the open network, if you try to close ResSim or change modules, ResSim will ask you to save with the query dialog shown in Figure 6.21.

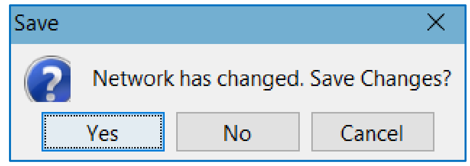


Figure 6.21 Save Network Query

Click **Yes** to save the network and proceed with activity that you initiated.

Click **No** to discard your changes and proceed with the activity.

Click **Cancel** to abort the activity and return to the **Reservoir Network** module. Your changes will not be saved to disk nor will they be discarded.

6.4.4 Renaming Networks

Renaming a network seems like a simple thing to do. And it is. All you have to do to rename a network is:

- Select **Rename...** from the Network menu to open the Rename Network dialog (Figure 6.22).
- Select the network you want to rename from the list of **Existing Reservoir Networks**.
- Enter a new **Name** (and **Description**) for the network.
  - Click **Rename** to rename the selected network.
  - Click **Cancel** to abort the rename process.

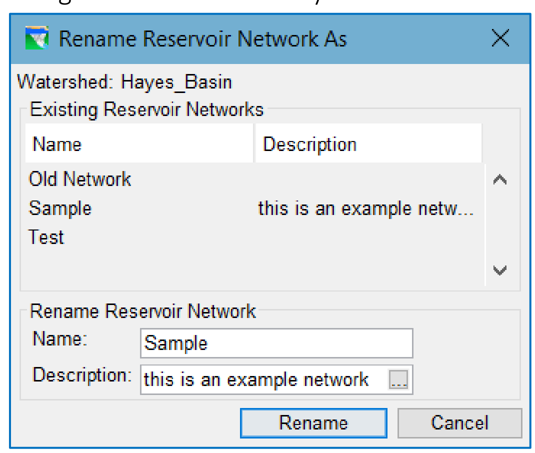


Figure 6.22 Rename Network As Dialog

Renaming a network may have unintended consequences:

- A reservoir network may be referenced by one or more alternatives. Since the alternatives are tied to the network by its ID number, not its name, the connection between the alternatives and the network are not broken when you rename a network. However, things can get a bit more complicated in your simulations. If you have a simulation that was created before you renamed the network that one or more of the simulation's alternatives are based on, the alternatives will continue to compute and function as they did before, but the new name of the network will not migrate into the simulation, not even with the **Replace From Base** feature.
- And, when you rename a network, the eight files that contain your network's data on disk do not actually get renamed. Instead, a new set of files are created with the new network name and the primary network file of the old network is

deleted, leaving behind seven files that still have the original name of the network cluttering-up your watershed.

6.4.5 Deleting Networks

You will occasionally find it necessary to delete unnecessary networks from your watershed. To delete one or more networks:

- Select **Delete Networks…** from the **Network** menu. The **Delete Networks** dialog will open (Figure 6.23).
- Select the network that you want to delete from the list of **Existing Reservoir Networks**. You can select more than one. The names of the selected networks will appear in the **Name** field below.
- Click **OK** to proceed (or **Cancel** to abort the delete process).
- An "are you sure…?" dialog listing the networks you selected to delete will appear (Figure 6.24).
    - Click **Yes** to delete the selected network(s).
    - Click **No** to abort the delete process.
    - Click **Cancel** to return to the **Delete Networks dialog.**

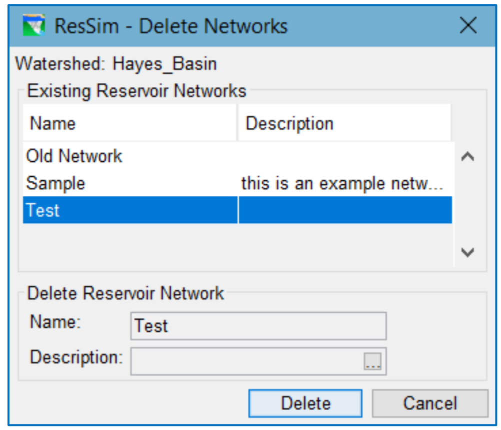


Figure 6.23 Delete Networks Dialog

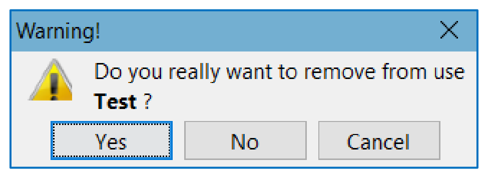


Figure 6.24 Delete Networks Query

Although deleting reservoir networks is simple, there may be unintended consequences if you are not careful:

- *Alternatives reference networks.* If you delete a network that is referenced by an existing alternative, that gets broken. When you select a select a network that has been broken in this manner, an Error Message dialog will appear (Figure 6.25) and the alternative will not display any of its settings in the editor. Since there is no way to attach another network to your alternative, your only option with a broken alternative is to delete it.
- *Simulations contain copies of alternatives*. If you have a simulation that contains a copy of one or more alternatives that have been broken by the delete of their network, the alternatives in the simulation become orphans. They will continue to function correctly in the simulation but their connection to the original alternative is broken so **Save to Base** and **Replace from Base** will no longer work for those alternatives.

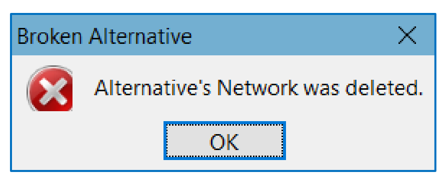


Figure 6.25 Broken Alternative Message

6.4.6 Updating a Network from its Configuration

In the **Watershed Setup** Module, if you make changes (e.g., add, delete, move or rename elements) to the watershed configuration that your network is based on, you may need to update your Reservoir Network to include those changes.

To update the active network from its configuration:

- Select **Update Network from Configuration** from the **Network** menu.
- A confirmation dialog will appear (Figure 6.26).

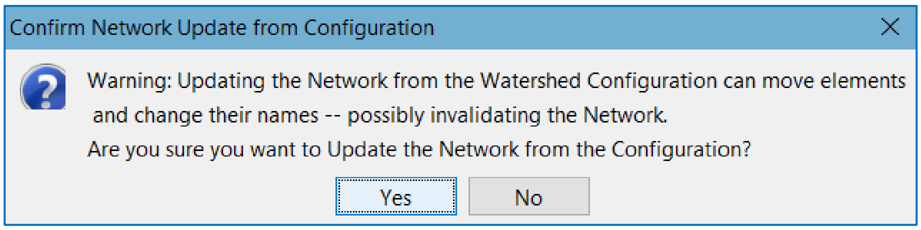


Figure 6.26 Confirm Network Update from Configuration

After the network has been updated from the revised configuration, you should see the revisions reflected model schematic in the Map Display area.

6.5 The Reservoir Network Reports

In the **Reservoir Network** module, the **Reports** menu provides access to summary reports pertinent to the elements in your active network. Each report includes a table containing the list of information pertinent to that report. In addition, **Advanced** reports describing your network connectivity are also available.

For information about printing and exporting options available from each report's **Report** menu, see Appendix E.

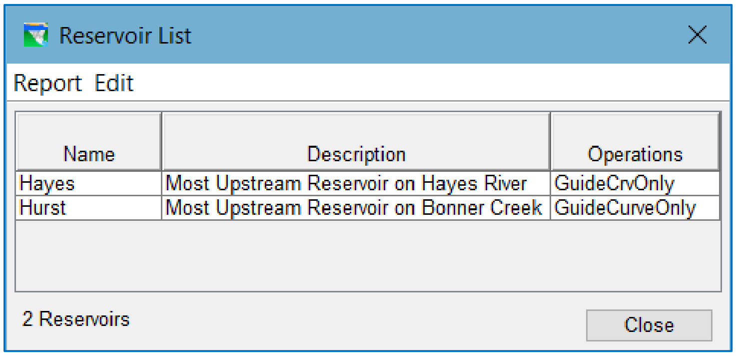
Tip

Sorting a Report Table:

In most reports, if you click on a column header of the table, the contents of the table will resort alpha-numerically based on the content of that column.

6.5.1 The Reservoir List

The **Reservoir List** (Figure 6.27) displays the names, descriptions, and operations of all reservoirs in your network. Additionally, the number of reservoirs is listed in the bottom right corner. For information about printing and exporting options available from the report's **Report** menu, see Appendix F. The **Edit** menu provides access to the *Reservoir Editor* (after selecting a reservoir in the list and then selecting **Edit** from the **Edit** menu).



Reservoir List

Report Edit

Name	Description	Operations
Hayes	Most Upstream Reservoir on Hayes River	GuideCrvOnly
Hurst	Most Upstream Reservoir on Bonner Creek	GuideCurveOnly

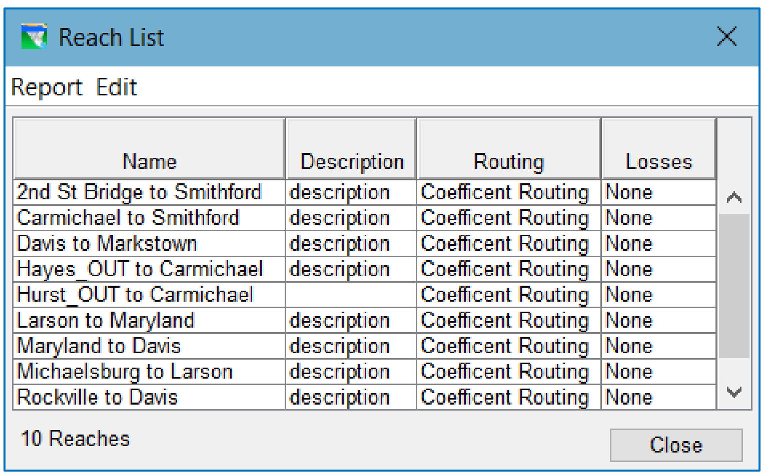
2 Reservoirs

Close

Figure 6.27 Reservoir Network Reports—Reservoir List

6.5.2 The Reach List

The **Reach List** (Figure 6.28) displays the names, descriptions, routing method, and losses of all reaches in your network. For information about printing and exporting options available from the report's **Report** menu, see Appendix F. The **Edit** menu provides access to the *Reach Editor* (after selecting a reach in the list and selecting **Edit** from the **Edit** menu).



Reach List

Report Edit

Name	Description	Routing	Losses
2nd St Bridge to Smithford	description	Coefficent Routing	None
Carmichael to Smithford	description	Coefficent Routing	None
Davis to Markstown	description	Coefficent Routing	None
Hayes_OUT to Carmichael	description	Coefficent Routing	None
Hurst_OUT to Carmichael		Coefficent Routing	None
Larson to Maryland	description	Coefficent Routing	None
Maryland to Davis	description	Coefficent Routing	None
Michaelsburg to Larson	description	Coefficent Routing	None
Rockville to Davis	description	Coefficent Routing	None

10 Reaches

Close

Figure 6.28 Reservoir Network Reports—Reach List

Tip

Finding Schematic Elements:

In large or unfamiliar models, it can be difficult to locate a specific model element. The network summary reports are a great tool for this. Find the element you are looking for in the report list and click on it to select it. When an element is selected in the report listing, that element is highlighted in the Map Display area.

6.5.3 The Junction List

The **Junction List** (Figure 6.29) displays the names and descriptions of all junctions in your network. For information about printing and exporting options available from the report's **Report** menu, see Appendix F. The **Edit** menu provides access to the

Junction Editor (after selecting a junction in the list and selecting **Edit** from the **Edit** menu).

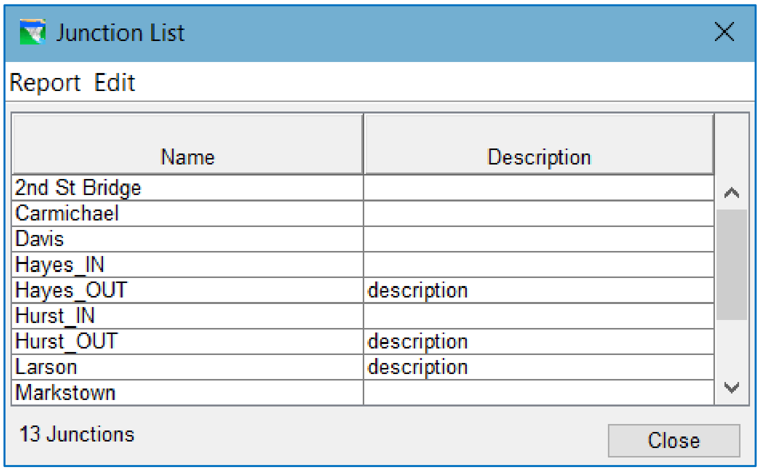


Figure 6.29 Reservoir Network Reports—Junction List

Tip *Deleting Schematic Elements:*

If you encounter a circumstance in which you want to delete an element that you either cannot find or select in the Map Display area or you want to delete several element at once, the network summary reports can come in hand for this too. Select the element(s) you want delete in the table, then select Delete from the report's Edit menu.

6.5.4 The Diversion List

The **Diversion List** (Figure 6.30) displays the names, descriptions, and method of all diversions in your network. For information about printing and exporting options available from the report's **Report** menu, see Appendix F. The **Edit** menu provides access to the *Diversion Editor* (after selecting a diversion in the list and selecting **Edit** from the **Edit** menu).

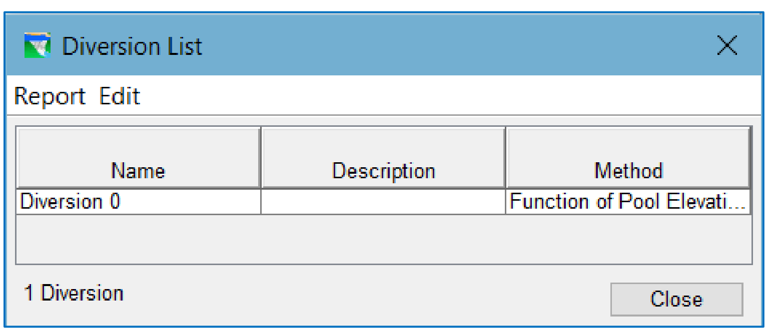


Figure 6.30 Reservoir Network Reports—Diversion List

6.5.5 The Advanced Reports

Two **Advanced Reports** are available from the **Reports** menu—**Network Connectivity** and **Node List**. These reports each describe how the model schematic is connected.

6.5.6 The Network Connectivity Report

The **Network Connectivity** report provides a detailed list of the elements (and sub-elements) that make up the model schematic of the active reservoir network. This report is complicated and requires that you understand 1) the concepts of elements, sub-elements, and the nodes that connect them and 2) how the report is arranged to identify the elements, their sub-elements, the connecting, and the connections.

This report uses row *shading* to indicate grouping of sub-elements within other elements. The rows identifying parent or primary schematic elements have a grey background. If a primary element contains sub-elements, those sub-elements will have a light grey background. The connection nodes of an element or sub-element have a white background and immediately follow the element or sub-element they pertain to.

The headers for each column identify their contents, however, some of the headers include two labels separated by a slash (/). The first label applies to the data in the grey or light grey rows (the elements and sub-elements). The second label applies to the data in the white rows (the nodes). The column headers include:

Element/Node—the element or node type. These include:

JunctionElement—a Junction element; a primary schematic element. As far as the network connectivity is concerned, junctions do not have physical sub-elements. However, they *do* have virtual sub-elements—local inflows—which show up in the network connectivity report as inflow nodes at the junction.

ReachElement—a Reach element; a primary schematic element. As far as the network connectivity is concerned, reaches do not have sub-elements and, when connected *properly*, they have only one inflow and one outflow node.

DiversionElement—a diversion; a primary schematic element. Diversions are a bit more complex than junctions or reaches; they *do* contain sub-elements. Every diversion has one sub-element named Cntrl which identifies flow entering the diversion. If the diversion is routed or returned to the system, there will also be a sub-element named Rch, which represents the flow leaving the diversion.

ReservoirElement—a reservoir; a complex element in the network that contains other sub-elements.

If a sub-element of the reservoir is a complex element with sub-elements of its own, it may be one of the following:

ReservoirDamElement—the dam. The dam contains at least two sub-elements by default—Tailwater and L&O.

DivertedOutletElement—a diverted outlet that was added to the reservoir. Like the dam, it will have a "tailwater" sub-element and, if connected back into the network at its downstream end, it will also have a "reach" element.

OutletGroupElement—an outlet group added to the dam, diverted outlet, or another outlet group.

If a sub-element of the reservoir or of one of its complex sub-elements is a simple element with no sub-elements of its own, it will be labeled **Element** and may represent one of the following:

The **Pool**—the storage element of the reservoir.

The **Tailwater** junction—the collector of the various release elements of the dam or diverted outlet.

The **L&O**—the *default* release element of the dam that represents **L**eakage (if specified) and **O**verflow.

An **Outlet**—a release element added to the dam or to an outlet group.

ID—the number of the element or node.

Name—the name of the element or node. Elements will show either the name you gave the element or the name ResSim assigned to it—like *Pool* for the reservoir pool element. Nodes that represent a local inflow at a junction will show the name you gave to the local inflow. All other nodes are assigned a name in the form *~Enn* where *nn* usually relates to the element number of the parent element of the node.

Function/Upstrm Elem—Elements will indicate the purpose of the element, e.g., the *function* of a controlled outlet is *AdjustableFlow*.

Nodes will identify the ID number of the element upstream of the node.

Parent/Dnstrm Elem —Sub-elements will indicate the name of the parent element to which the sub-element belongs. If blank, the element is a primary element.

Nodes will identify the ID number of the element that is downstream of the node.

Note: the purpose of the Network Connectivity Report is to show how all the network elements are connected through the inflow and outflow nodes listed with each element. But to fully understand the report you need to know a few things:

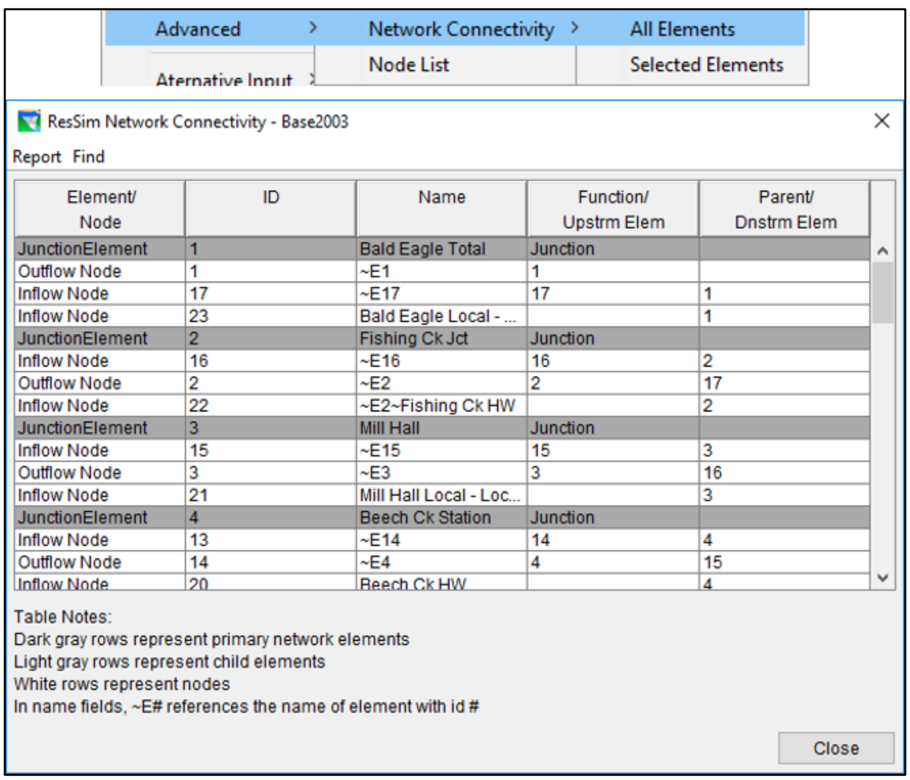
1) Every element has an outflow node that "belongs" to that element; that node will always show the ID number of the element it belongs to (or one of that element's sub-elements) as the upstream element.
2) Local inflows appear as inflow nodes at junctions; these nodes display blank for the upstream element.
3) Inflow nodes almost always "belong" to an element other than the element in which they are listed as an *inflow* node. The exceptions to this are:
    a. Diversions are treated as negative inflows at junctions. Thus, Diversion elements place an *inflow* node at the junction *from which*

they remove water and that node appears as an *outflow* node in the diversion element. That's bad enough but this oddity also shows up in the upstream and downstream element ids. The inflow node that represents the flow **going to** the diversion has the diversion element id as the **upstream** element; and that same node, listed as an outflow node at the diversion, has the ID number of the junction element it is taking water from as the downstream element.

b. Reservoir sub-elements that represent a group of outlets (dam, outlet group, or diverted outlets) act like a *passthrough* element for the elements they contain; thus, the inflow nodes listed in the dam are also inflow nodes to the outlets they contain, even if those outlets are contained in an outlet group. In addition, each group-type element contains a junction sub-element that collects the outflow nodes of the outlets into a single outflow that then appears as the outflow node of the (internal) junction and of the group.

To generate a **Network Connectivity** report for **all elements** in the active network:

- from the Reports menu, select Advanced → **Network Connectivity** → **All Elements** (Figure 6.31)



Element/ Node	ID	Name	Function/ Upstrm Elem	Parent/ Dnstrm Elem
JunctionElement	1	Bald Eagle Total	Junction	
Outflow Node	1	~E1	1	
Inflow Node	17	~E17	17	1
Inflow Node	23	Bald Eagle Local - ...		1
JunctionElement	2	Fishing Ck Jct	Junction	
Inflow Node	16	~E16	16	2
Outflow Node	2	~E2	2	17
Inflow Node	22	~E2~Fishing Ck HW		2
JunctionElement	3	Mill Hall	Junction	
Inflow Node	15	~E15	15	3
Outflow Node	3	~E3	3	16
Inflow Node	21	Mill Hall Local - Loc...		3
JunctionElement	4	Beech Ck Station	Junction	
Inflow Node	13	~E14	14	4
Outflow Node	14	~E4	4	15
Inflow Node	20	Beech Ck HW		4

Figure 6.31 Reservoir Network Reports—Advanced—Network Connectivity—"All Elements"

To generate a Network Connectivity report for a set of selected elements:

- First, select elements you want in the report by holding down the Shift key and clicking on the desired elements in the Map Display area (holding the Shift key allows you to select multiple schematic elements as indicated in the map shown in Figure 6.32).

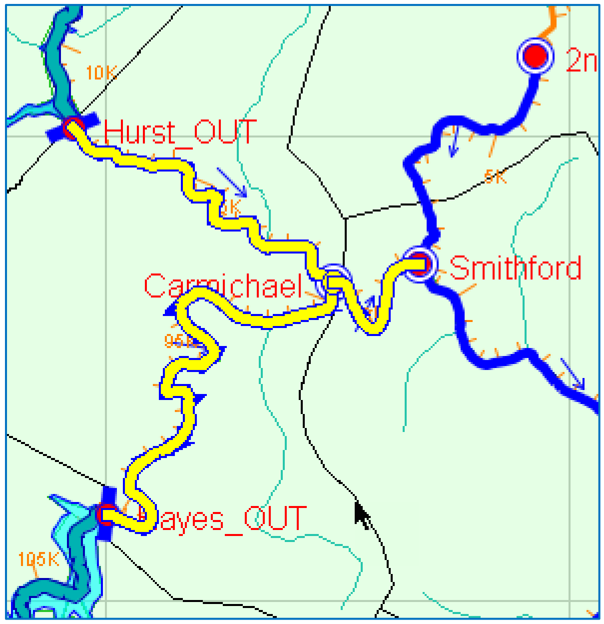


Figure 6.32 Selection of Multiple Schematic Elements

- Then, from the **Reports** menu, select: **Advanced → Network Connectivity → Selected Elements.** The report will look just like the one shown in Figure 6.31 but will only contain information for those elements you selected in the Map Display area.

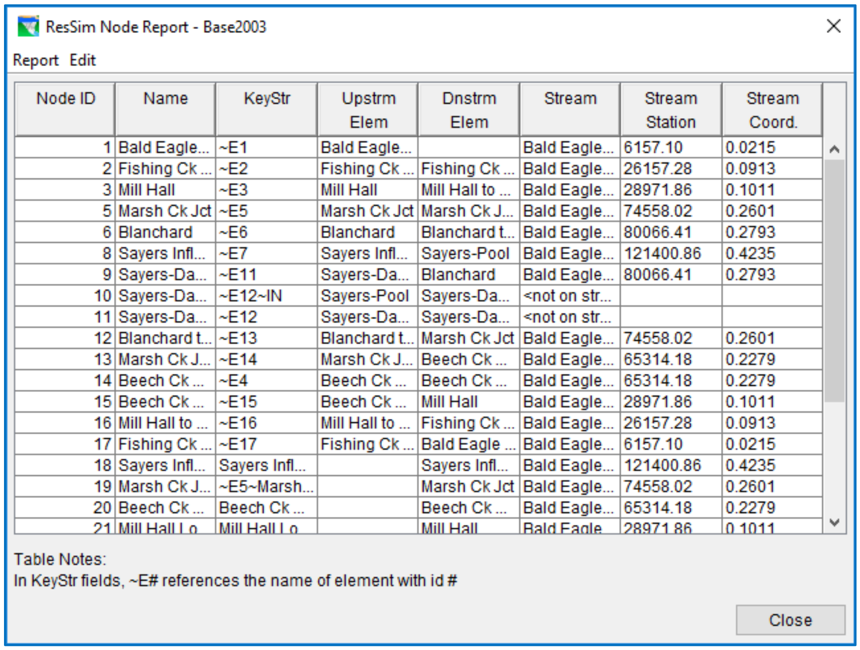
The **Find** menu of the **Network Connectivity** report allows you to search for text strings within the report.

6.5.7 The Node List

The second advanced report is the **Node List** (Figure 6.33). This report provides a summary of all nodes in your reservoir network. Details include the Node ID, Name, Key String, Upstream Element, Downstream Element, Stream, Stream Station, and Stream Coordinate (i.e., a normalized position on the stream).

To generate a **Node List** report:

- from the **Reports** menu, select Advanced → **Node List**



ResSim Node Report - Base2003

Report Edit

Node ID	Name	KeyStr	Upstrm Elem	Dnstrm Elem	Stream	Stream Station	Stream Coord.
1	Bald Eagle...	~E1	Bald Eagle...		Bald Eagle...	6157.10	0.0215
2	Fishing Ck ...	~E2	Fishing Ck ...	Fishing Ck ...	Bald Eagle...	26157.28	0.0913
3	Mill Hall	~E3	Mill Hall	Mill Hall to ...	Bald Eagle...	28971.86	0.1011
5	Marsh Ck Jct	~E5	Marsh Ck Jct	Marsh Ck J...	Bald Eagle...	74558.02	0.2601
6	Blanchard	~E6	Blanchard	Blanchard t...	Bald Eagle...	80066.41	0.2793
8	Sayers Infl...	~E7	Sayers Infl...	Sayers-Pool	Bald Eagle...	121400.86	0.4235
9	Sayers-Da...	~E11	Sayers-Da...	Blanchard	Bald Eagle...	80066.41	0.2793
10	Sayers-Da...	~E12~IN	Sayers-Pool	Sayers-Da...	<not on str...		
11	Sayers-Da...	~E12	Sayers-Da...	Sayers-Da...	<not on str...		
12	Blanchard t...	~E13	Blanchard t...	Marsh Ck Jct	Bald Eagle...	74558.02	0.2601
13	Marsh Ck J...	~E14	Marsh Ck J...	Beech Ck ...	Bald Eagle...	65314.18	0.2279
14	Beech Ck ...	~E4	Beech Ck ...	Beech Ck ...	Bald Eagle...	65314.18	0.2279
15	Beech Ck ...	~E15	Beech Ck ...	Mill Hall	Bald Eagle...	28971.86	0.1011
16	Mill Hall to ...	~E16	Mill Hall to ...	Fishing Ck ...	Bald Eagle...	26157.28	0.0913
17	Fishing Ck ...	~E17	Fishing Ck ...	Bald Eagle ...	Bald Eagle...	6157.10	0.0215
18	Sayers Infl...	Sayers Infl...		Sayers Infl...	Bald Eagle...	121400.86	0.4235
19	Marsh Ck J...	~E5~Marsh...		Marsh Ck Jct	Bald Eagle...	74558.02	0.2601
20	Beech Ck ...	Beech Ck ...		Beech Ck ...	Bald Eagle...	65314.18	0.2279
21	Mill Hall Lo	Mill Hall Lo		Mill Hall	Bald Eagle	28971.86	0.1011

Table Notes:
In KeyStr fields, ~E# references the name of element with id #

Close

Figure 6.33 Reservoir Network Reports—Advanced—Node List

The **Edit** menu provides two options: **Clean Network** and **Delete Node**. These options can potentially damage a working network and should only be used when recommended by a ResSim technical expert.

Chapter 7
Junctions, Reaches, & Diversions

Once you have created a reservoir network and established its connectivity (as described in Chapter 6), you will need to specify the properties of the elements in the network. To do so, you will use ResSim's specialized network element editors. This chapter explains the **Junction**, **Reach**, and Diversions **Editors** and how to enter and edit data for these network elements. Chapters 8-12 describe the **Reservoir Editor** and how to edit the physical and operational data for your reservoirs.

As explained in Chapter 6, the **Reservoir Network** module of ResSim is effectively *the editor of a reservoir network.* As such, the Reservoir Network module is where you will create your reservoir network and definite the data for its elements. However, when you begin to compute your alternatives in the **Simulation** module, you will learn that the same editors you used to define the elements of you network are accessible there as well.

There are two methods for accessing the element editors from the ResSim interface:

From the **Edit** menu—select the element type you want to edit. Figure 7.1 shows the **Edit** menu and its list of **Editors** that you can use to edit the elements of your network.

From the **Map Display**—Using either the **Pointer Tool** or the specific element's **Tool**, *right-click* on the element in the **Map Display** area and select **Edit ... Properties** from the schematic element's context menu

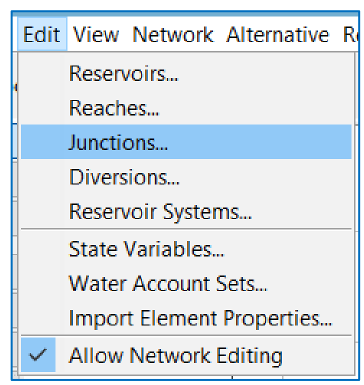


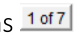
Figure 7.1 Reservoir Network Module—Edit Menu

All the reservoir network element editors (e.g., the **Junction Editor**, shown in Figure 7.2) share some common characteristics. These include:

Name Field—The **Name** field at the top of the editor displays the name of the element that you are currently editing. However, in addition to identifying the current element, the Name field has a second function—it is an element selector; it contains a list of all the elements (of this editor's type) in your network, allowing you to *select* any element in the list to edit without leaving the editor.

Navigation Buttons—To the right of the **Name** field are a set of *Navigation Buttons* 1 of 7 that allow you to move forward and backward through the list of elements for editing.

The leftmost button will take you to the first element in the list.
The rightmost button will take you to the last element in the list.
The middle two buttons, and , will select the previous or next element, respectively, in the list.

The box in the middle of the navigation buttons  displays the index of the current element and the total number of elements of that type in the currently open dataset (in this case, the network).

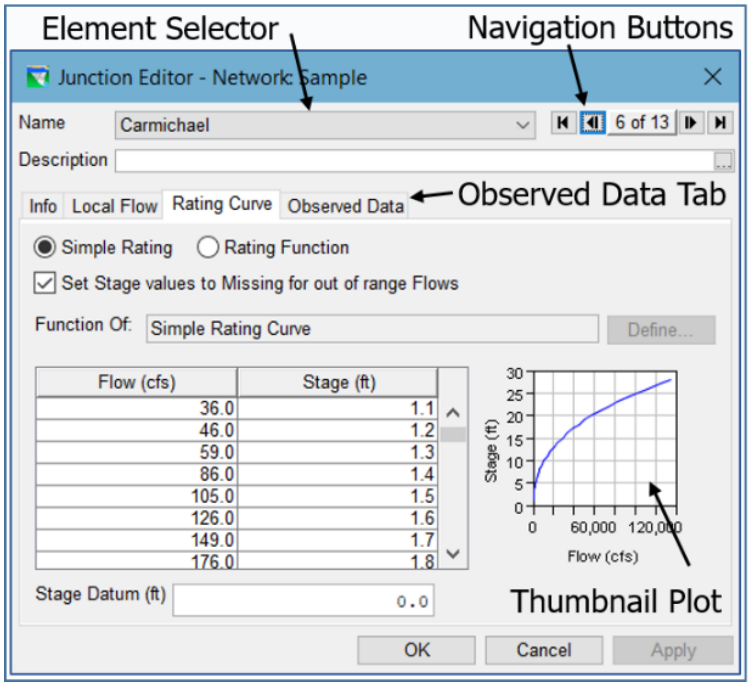


Figure 7.2 Element Editor—Common Features

Description Field—Below the **Name** field is the **Description**. If you provided a description when you named the network element (or its associated configuration element), that description will appear in this field. Almost all description fields throughout ResSim are editable, allowing you to change and add to them as you work on your model.

> ***Tip*** **Description** Fields accompany almost every namable object you will create in ResSim. They are included in the **Name** dialog when you first create an object and they are also included in all the editors where you define the data for the object. USE THEM!
> You'll find there's nothing more helpful to you, other modelers, and reviewers than a well internally-documented model.
> So, when you enter your descriptions, try to answer as many of the Who, What, Where, When, Why, and How questions as you can; describe what the object represents, the sources of the data or parameters, why you chose to represent the physical or operational element the way you did, and what result or behavior you were expecting.

Thumbnail Plot—Wherever a table of data appears in a ResSim editor, a small plot called a **thumbnail plot** will accompany it, as illustrated in Figure 7.2. These plots are included to help you quickly notice bad data in the table. To get a full-size plot of the data, double-click on the thumbnail plot.

Tab Panels—The Tabs provide access to various parameters for defining the current element.

Observed Data tab—The **Observed Data** tab (Figure 7.3) contains a table listing the computed parameters for the element. If you *may* want to associate an observed time series with one or more of the listed variables, place a checkmark in the checkbox to the right of the Variable(s) of interest by clicking anywhere in the checkbox's cell. For each Location and Variable that has a checkmark in the **Observed** box in the table, a corresponding entry will appear in the table on **Observed Data** tab of the **Alternative Editor**. See Chapter 14 for details on associating observed time series data in the **Alternative Editor** with the selected locations and variables from the element editors.

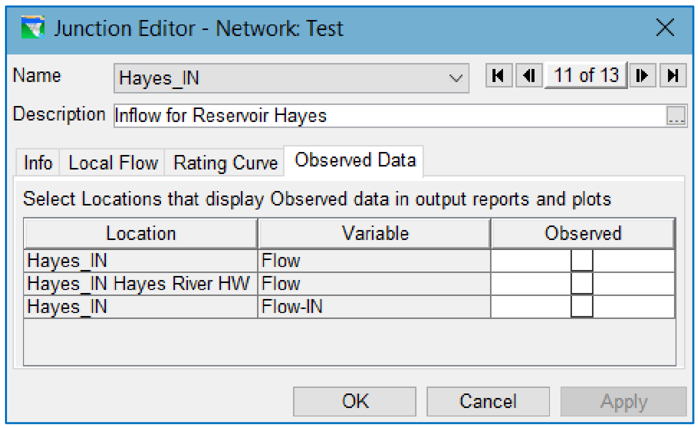


Figure 7.3 Element Editor—Observed Data Tab

Providing observed data is optional. ResSim uses observed data for comparison purposes only — and then, only in its standard element plots accessible from the active schematic in the Simulation module. ResSim does not use observed data in its computations.

OK, Cancel, and **Apply Buttons**—Use these buttons to save or discard your changes and close the Editor:

OK—will *save* your changes and close the Editor.
Cancel—will *discard* your changes and close the Editor.
Apply—will *save* your changes without closing the Editor. You should use the **Apply** button before changing tabs or navigating to another element in the Editor. Even if you **Cancel** after an **Apply**, your changes up to the last **Apply** action are saved.

7.1 The Junction Editor

Use the **Junction Editor** (Figure 7.4) specify the properties of the junctions in your network.

The **Junction Editor** has four tabs on which you will define your junction data:

- Info
- Local Flow
- Rating Curve, and
- Observed Data.

These tabs are described in the following sections.

7.1.1 Junction Editor—Info Tab

The **Info** tab of the **Junction Editor** (Figure 7.4) displays the **Stream Station**, **Stream**, and **Tributaries** associated with your junction. These properties describe the junction's relationship to the stream alignment.

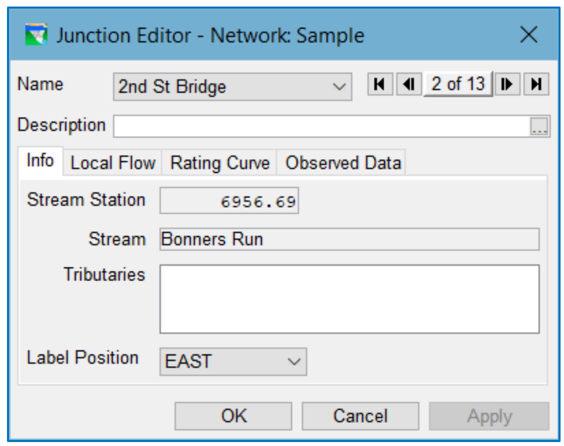


Figure 7.4 Junction Editor—Info Tab

Stream Station—the position of the junction is described as its distance (in map units) from the downstream end of the stream it's on in the stream alignment.

If the junction is not associated with a computation point in a watershed configuration, the **Stream Station** can be edited. However, by changing the stream station of a junction, you are changing the position of the junction along the stream alignment, so be careful with this option, you can get unintended consequences and there are restrictions — for example, you should not attempt to move a junction above or below another junction on the current stream by changing its stationing.

Stream and **Tributaries**—These fields are provided so that you can verify that the junction is located where you intended when you placed it in your schematic from the **Reservoir Network** or **Watershed Setup** module.

Label Position—Near the bottom of the Info tab is a **Label Position** field. This field is a dropdown list that contains the 8 points on the compass (**North, South, East, West, Northeast, Northwest, Southeast, and Southwest**). Use this field to control where the label for the current junction will display in the **Map Display** area. The junction's **Name** will be drawn horizontally in the position you selected with respect to the center of the junction (plus a small offset).

7.1.2 Junction Editor—Local Flow Tab

The **Local Flow** tab (Figure 7.5) of the **Junction Editor** contains a table where you can identify one or more inflows entering the river system at this junction.

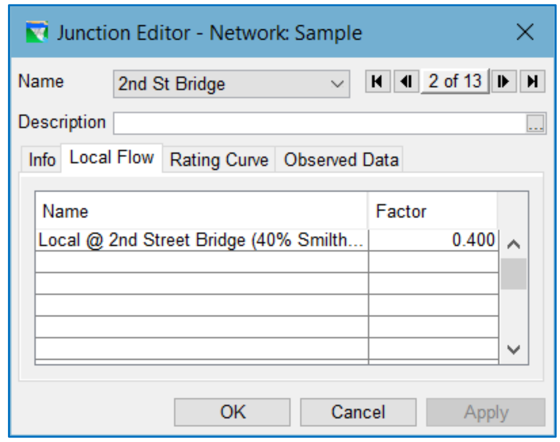


Figure 7.5 Junction Editor—Local Flow Tab

You must identify at least one inflow *at each headwater junction* in your watershed; inflows at all other junctions are optional. Each inflow is identified by a unique **Name** and **Factor.**

Name—The name of each **Local Flow** should identify the source of the inflow or contain some other uniquely identifying information so that it will be clear what is required when it comes time to associate a time series of flow data with this inflow in the **Time-Series** tab of the **Alternative Editor**. (See Chapter 13 for more on creating Alternatives.)

There is limit on length of the local flow name, but it should be unique within the first 24-32 characters. And, since this name will be used in the B Part of a DSS pathname, do not use the following characters: \ ‘ / ` “ and |.

NOTE: Local flows cannot have the same name as a junction (or any other network element).

Factor—The factor value is provided to enable you to provide a weighting of the values in the associated inflow hydrograph. Use this **Factor** for "basin weighting" of a single inflow hydrograph across multiple junctions. The default value for the **Factor** is 1.0.

When you identify one or more local inflows on the **Local Flow** tab of a junction, then a visual clue will appear in your model schematic in the map display. This visual clue appears as a white circle or halo around the junction, as illustrated in Figure 7.6. The white halo makes it easy to tell at a glance where inflows are coming into your reservoir network.

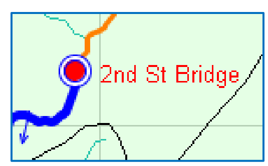


Figure 7.6 Junction with Local Inflow

7.1.3 Junction Editor—Rating Curve Tab

Most junctions do not require a rating curve, but for those that do or for any junction where you would like ResSim to output stage associated with is computed junction flow, you can use the **Rating Curve** tab (Figure 7.7) to enter a rating curve for that junction. Two options are available for entry of a rating curve — **Simple Rating** and **Rating Function**.

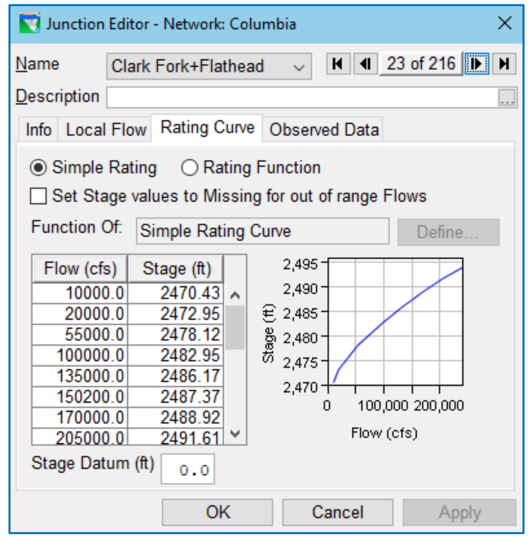


Figure 7.7 Junction Editor—Rating Curve Tab

Simple Rating—A Simple Rating can be used where you have a relationship between **Flow** and **Stage** at the junction. ResSim considers **Flow** to be the independent variable and **Stage** to be the dependent variable of a rating curve so be sure to enter **Flow** in the first column of the table and **Stage** in the second column. Figure 7.7 shows an example of a Simple Rating.

Rating Function—A Rating Function can be used when **Stage** is a function of a variable *other than **Flow** at the junction* or when **Stage** is influenced by backwater effects. If you select the **Rating Function** option, you will be able to specify a stage lookup table as a function of:

- **Model Variable**,
- **External Variable**,
- **State Variable**, or
- **Two Variables** (such as flow and downstream pool elevation). Figure 7.8 illustrates an example of a family of rating curves (stage as a function of *two* independent variables). For more information on setting the independent variable(s) of a function (lookup table), refer to Appendix C.

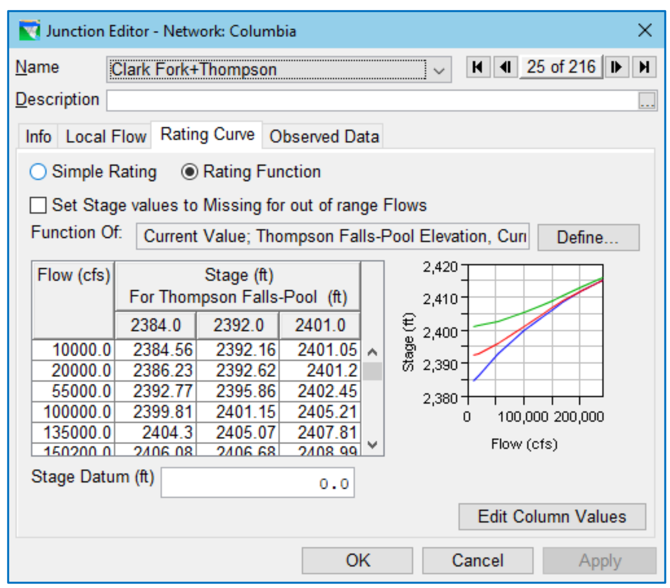


Figure 7.8 Junction Editor—Rating Curve—Function of Two Variables

7.2 The Reach Editor

Reaches route water through your reservoir network by employing a hydrologic routing method. Use the **Reach Editor** (Figure 7.9) to specify routing method, the routing parameters, and the loss rates for each of your reaches.

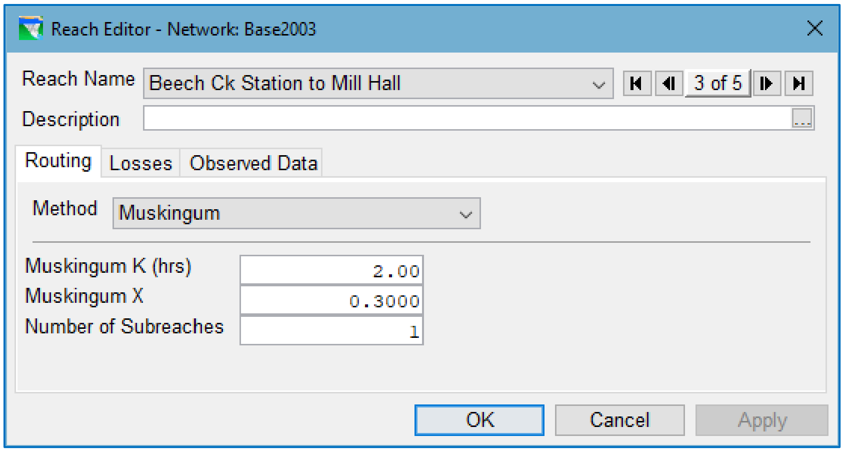


Figure 7.9 Reach Editor

The **Reach Editor** has three tabs with which to define your reach data:

- **Routing**,
- **Losses**, and
- **Observed Data**.

These tabs and their options are described in the following sections.

7.2.1 Reach Editor—Routing Tab

The Reach Editor's **Routing** tab (Figure 7.10) is where you will select the reach's hydrologic routing method and enter the parameters for the selected method. There are nine routing methods to choose from:

- **Null Routing**
- **Coefficient**
- **Muskingum**
- **Muskingum-Cunge 8-pt Channel**
- **Muskingum-Cunge Prismatic Channel**
- **Modified Puls, SSARR, Working R&D Routing**
- **Variable Lag & K**.

Each method requires a different set of parameters.

Select the routing method for the reach from the dropdown list of the **Method** selector. The panel below the **Method** selector will fill with the appropriate fields for defining the parameters for the routing method you selected.

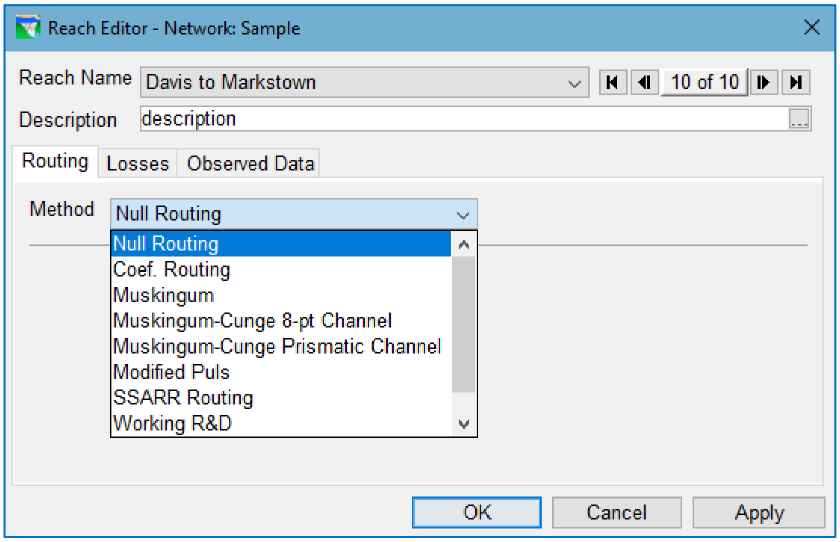


Figure 7.10 Reach Editor—Routing Tab

NOTE: If the flow entering a reach is negative, only the Null and Coefficient Routing methods will "route" that negative flow to the outflow of the reach. All other routing methods will not route negative flow and will instead produce a zero outflow. This "zero-ing out" of the negative inflow has the net effect of "creating water" and could be considered a violation of the conservation of mass of the system.

7.2.1.1 Null Routing

The **Null Routing** method produces an instantaneous translation of the inflow to the outflow of the reach. Since this method applies no lag or attenuation to the inflow, it requires no parameters as you can see in Figure 7.11. If you have a very short reach or a reach that exhibits no lag or attenuation of the hydrograph, you can use **Null Routing**.

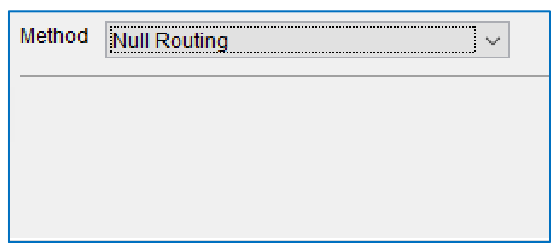


Figure 7.11 Null Routing Edit Panel

7.2.1.2 Coefficient Routing

The **Coefficient Routing** method is a linear hydrologic routing method that describes the routing with a series of timesteps and coefficients (or factors). The coefficient method uses the following equation to compute outflow from the reach:

$$O_t = \sum_{i=1}^{n} C_i * I_{t-(i-1)}$$

Where:

O = Outflow
I = Inflow
C = Coefficient
t = the current timestep
n = the number of rows in the table.

Each coefficient equates to the fraction of the flow entering the reach that will reach the downstream end at the end of each timestep in the table. The values in the table must sum up to 1. The coefficients represent the fraction of the flow entering the reach in the current timestep that will exit the reach in the current and subsequent timesteps.

Figure 7.12 shows the edit panel for the **Coefficient Routing** method. In the example, 75% of the flow entering the reach in the current timestep plus 25% of the previous timestep's inflow will exit the reach in the current timestep. The other 25% of the current timestep's inflow to the reach will exit the reach in the next timestep, along with 75% of the next timestep's flow into the reach.

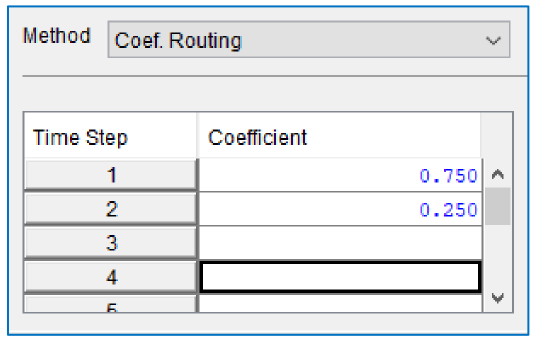


Figure 7.12 Coefficient Routing Edit Panel

Tip *If you enter a coefficient of 1.0 in the first row of the Coefficient Routing table and leave the rest of the table blank, you have defined a Null or instantaneous routing reach — all the flow entering the reach will exit the reach in the same timestep.*

7.2.1.3 Muskingum Routing

The **Muskingum** routing method (Figure 7.13) is a linear routing method that describes the routing with three parameters: **Muskingum K**, **Muskingum X**, and **Number of Subreaches**.

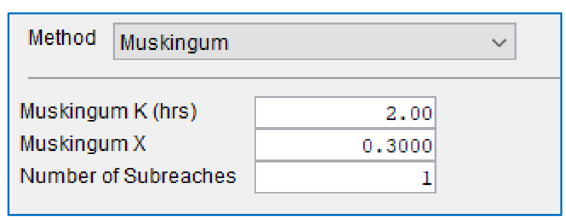


Figure 7.13 Muskingum Routing Edit Panel

The Muskingum method computes outflow from the reach using the following equation:

$$O_t = (A - B) * I_{t-1} + (1 - A) * O_{t-1} + B * I_t$$

Where:

$$A = \frac{2 * \Delta t}{2 * K * (1 - X) + \Delta t}$$

$$B = \frac{\Delta t - 2 * K * X}{2 * K * (1 - X) + \Delta t}$$

and:

O = Outflow
I = Inflow
t = the current timestep
Δt = the length of the timestep
K = Muskingum K
X = Muskingum X

Muskingum K—The travel time for the reach, in hours; i.e., how long it takes for a drop of water that enters the reach to exit the reach.

Muskingum X—The Muskingum weighting factor or attenuation coefficient, a value from 0.0 to 0.5. X= 0.0 indicates maximum attenuation of the hydrograph through the routing reach. X=0.5 indicates no attenuation; a "direct translation" of the hydrograph through the reach.

Number of Subreaches—The travel time through any reach should be at least as long as the timestep. If the travel time is greater than the timestep, the number of subreaches should approximately equal K/Δt. If the travel time of a reach is less than half the length of the timestep, consider using **Null** routing.

> *Tip*
>
> A Rule of Thumb…
> The Number of Subreaches should equal K/Δt (rounded to a whole number), where:
> K is the travel time and
> Δt is the length of the timestep (in hours).

7.2.1.4 Muskingum-Cunge 8-pt Channel Routing

The **Muskingum-Cunge 8-pt Channel** routing method is a non-linear hydrologic routing method that uses a representative cross section to apply a physical or hydraulic influence to the method.

The edit panel for the **Muskingum-Cunge 8-pt Channel Routing** method is shown in Figure 7.14. This panel includes a generalized illustration of an 8 point cross section to provide a guide for defining the cross section. Below the illustration is a thumbnail plot to show *your* cross section as defined in the **Cross Section Table**. Use the thumbnail plot to verify that your cross section points were entered correctly and adequately reflects the cross section template shown in the illustration.

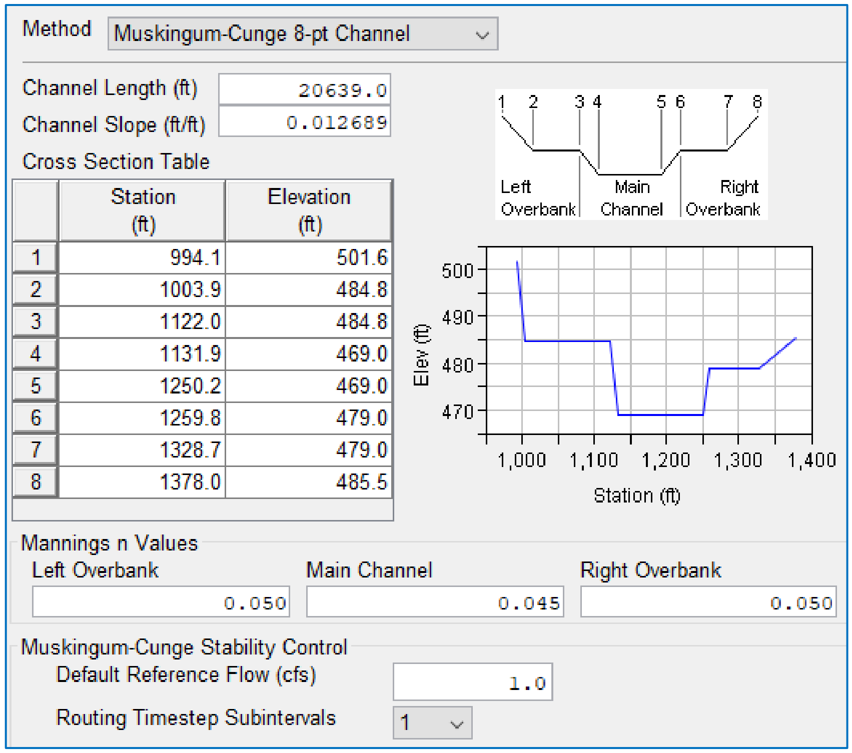


Figure 7.14 Muskingum-Cunge 8-pt Channel Routing Edit Panel

The **cross section illustration** in the edit panel for the Muskingum-Cunge 8pt Channel routing method shows the numbering of the eight cross section points and how they relate to the partitioning of the cross section into a Left Overbank, Channel, and Right Overbank. This illustration indicates that points 3 and 6 are the bank stations of the cross section, these points the mark the boundary between the channel and the overbanks.

Be sure that when you enter ***your*** cross section into the Cross Section Table that its bank stations are at points 3 and 6.

The parameters for defining the **Muskingum-Cunge 8-pt Channel** routing method include:

Channel Length—The length of the routing reach, measured along the channel centerline. This value is *not* computed by ResSim based on the stream alignment.

Channel Slope—The *average* bed slope of the routing reach, $\Delta y/\Delta x$.

Eight Station and Elevation pairs—The horizontal and vertical position (respectively) of each point used to describe the representative cross section for the reach.

Manning's n Values—The roughness coefficients for the **Left Overbank**, **Main Channel**, and **Right Overbank** are used in the normal depth calculations of the cross section.

Default Reference Flow—The reference flow is used to compute the celerity of a flood wave through the reach which is then used to compute the travel time for the reach. If the travel time is greater than the size of the computation interval (timestep), the reach is divided into subreaches for the computations. Travel time is usually greater for low flow; therefore, start with a default reference flow value that approximates the base flow for the reach.

Routing Timestep Subintervals—As its name suggests, this parameter is the number of increments to break the timestep into for purposes of numerical stability of the Muskingum-Cunge solution. Although some implementations of the Muskingum-Cunge method include a strategy for determination of the subintervals based on a review of the inflow hydrograph, this option is not feasible for ResSim because the flow hydrographs into the reaches downstream of the reservoirs are not "known" at the start of a ResSim simulation. As a result, ResSim requires you to select the number of subintervals as a whole number value between 1 and 12.

> *Tip* A Rule of Thumb…
> Use **fewer** subintervals for **constant or gradually** varying flow.
> Use **more** subintervals for **rapidly** varying flow.

7.2.1.5 Muskingum-Cunge Prismatic Channel Routing

The **Muskingum-Cunge Prismatic Channel** method (Figure 7.15) is the same basic method as the Muskingum-Cunge 8-pt Channel method but it uses one of two shapes to describe the representative cross section—a trapezoid or a circle. Use this method for reaches that have no overbanks and whose channel cross section can be approximated by a trapezoidal shape or for routing through pipes or culverts.

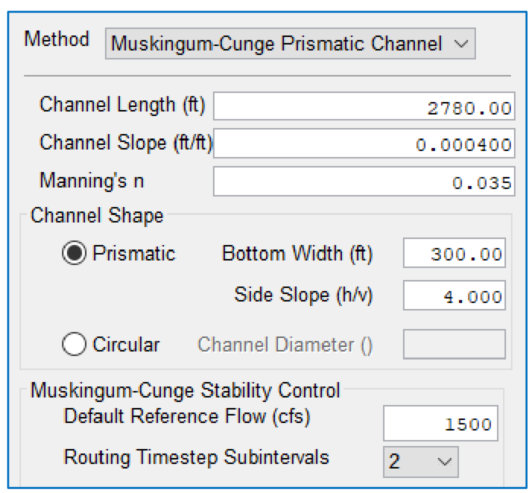


Figure 7.15 Muskingum-Cunge Prismatic Channel Routing

The parameters for defining the Muskingum-Cunge Prismatic Channel method include:

Channel Length—The length of the routing reach, measured along the channel centerline.

Channel Slope—The average bed slope of the routing reach, $\Delta y/\Delta x$ ("rise over run").

Manning's n—The roughness coefficient to be used in the normal depth calculations of the cross section.

Channel Shape—Select from the following channel cross section shapes:

Prismatic—the representative cross section is shaped like a trapezoid (whose bottom width is less than or equal to its top width — see Figure 7.16). The trapezoidal shape is defined by two parameters: a bottom width and a side slope; depth of the trapezoid is not required, ResSim will assume the channel is infinitely deep.

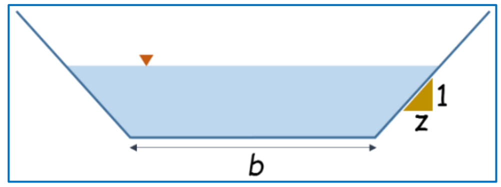


Figure 7.16 Prismatic Channel

Bottom Width—*b*, the width of the bottom of the trapezoidal section.

Side Slope—*z*, the slope of the sides of the trapezoid entered as $\Delta x/\Delta y$. ("run over rise". Yes, this is the inverse of the bed slope.) If the sides of your trapezoid are vertical, the slope is 0.0.

Circular—This cross section shape is a simple circle. Use this shape to represent a free-flowing pipe or culvert. This method and shape is valid for flow depths up to 0.77*diameter. For depths greater than this, the depth used by the method is held to 0.77*diameter and a warning message is generated. If you choose Circular as the Channel Shape, you also need to specify the Channel Diameter.

Channel Diameter—Define the size of the circle.

Muskingum-Cunge Stability Control—Enter the default reference flow and the routing timestep subintervals.

Default Reference Flow—The flow value is used to compute the celerity (and thus the travel time) for the reach. If the travel time is greater than the timestep size, the reach is divided into subreaches for the computation. Travel time (and thus the number of subreaches) is greater for low flow; therefore, the reference flow value should approximate the base flow for the reach.

Routing Timestep Subintervals—As its name suggest, this parameter is the number of increments to break the timestep into for purposes of ***numerical stability*** of the Muskingum-Cunge solution. Select the number of subintervals as a whole number value between 1 and 12.

7.2.1.6 Modified Puls Routing

The **Modified Puls** routing method (Figure 7.17) describes the reach as a series of cascading reservoirs, the number of which is specified by the **Number of Subreaches**. The outflow from the reach is defined as a function of storage in the reach. The parameters for defining the **Modified Puls** routing method are described below:

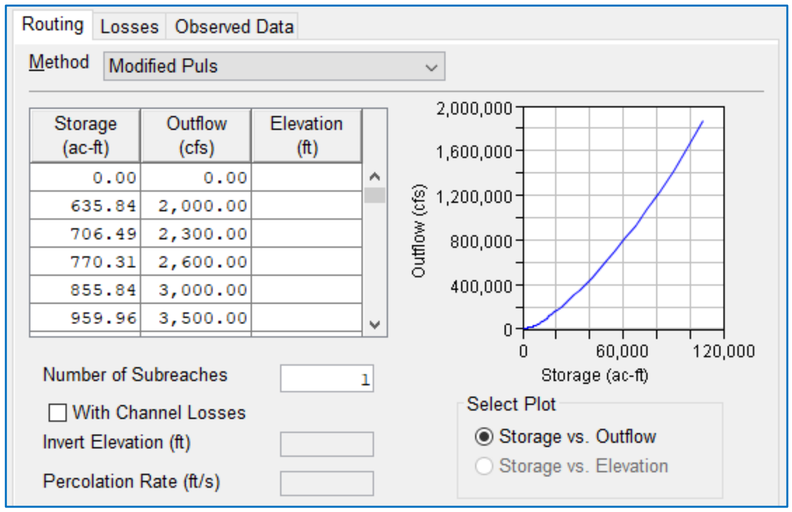


Figure 7.17 Reach Editor—Modified Puls Routing Method

Storage and **Outflow**—Enter a monotonically increasing set storage and outflow values to describe the storage-outflow relationship needed for this routing method. HINT: An HEC-RAS steady-flow model can be used to develop a Storage-Outflow relationship for use in this method.

Number of Subreaches—This parameter is similar to that used in the Muskingum routing method. Enter the reach travel time divided by the computation interval.

Channel Losses—An option to compute **Channel Losses** due to seepage within the routing computation is included in the edit panel for the Modified Puls routing method. The data required to use this option includes **Elevation** data associated with the **Storage** values in the Storage-Outflow table, an average Invert Elevation for the reach, and a Percolation (or Seepage) Rate. *However, the **Channel Losses** option and all the associated data entry fields have been disabled in this version. See the Losses tab of the Reach Editor to learn how to define channel losses that are computed independently of the routing method.*

7.2.1.7 SSARR Routing Method

The SSARR routing method (Figure 7.18) is an implementation of the channel routing method used in the computer program *Streamflow Synthesis & Reservoir*

Regulation (SSARR) developed the Corps' Northwestern Division (USACE, 1991) and NOAA's North West River Forecast Center.

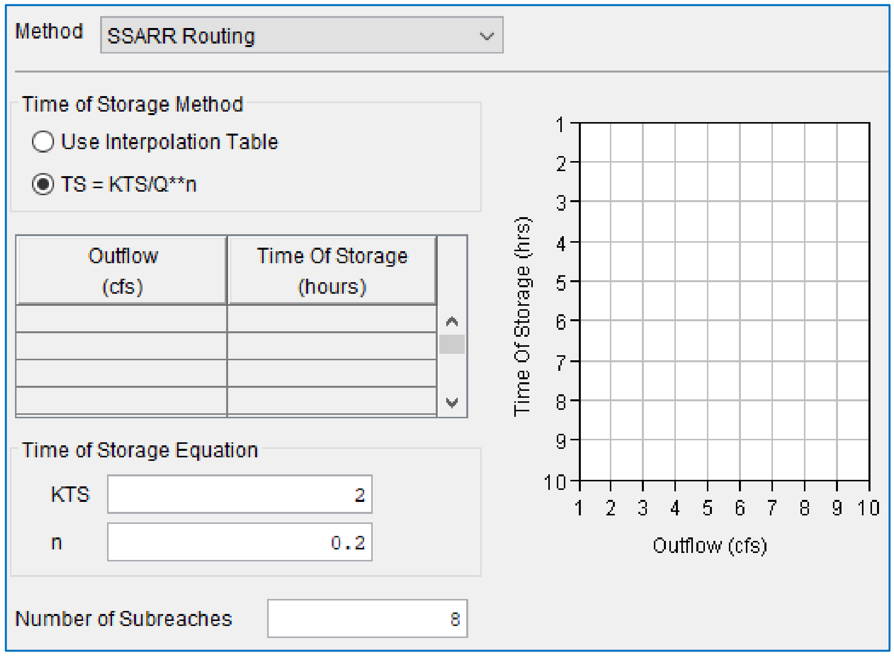


Figure 7.18 Reach Editor—SSARR Routing Method

SSARR is conceptually similar to Modified Puls in that it represents the reach as a "chain of lakes" and the **Number of Subreaches** defines the number of "lakes in the chain. The routing of each lake in the chain is described by the equation:

$$O_t = O_{t-1} + \Delta t * \frac{I_m - O_{t-1}}{T_s + \frac{\Delta t}{2}}$$

Where:

O_t = Outflow for this timestep
O_{t-1} = Outflow for the previous timestep
Δt = length of the timestep
T_s = time of storage
I_m = the mean inflow

Two options are available for defining the parameters for this routing method:

Outflow vs Time of Storage (Interpolation table)—In this form of the SSARR method, the outflow vs. reach storage is defined by Time of Storage values, T_s in units of hours.

Time of Storage equation—this equation defines T_S as a power function of outflow:

$$T_s = \frac{KTS}{Q^n}$$

Where:

T_s = Time of Storage (per increment, in hours)
Q = Outflow

KTS = Coefficient, determined empirically
n = The power coefficient on outflow; usually between -1.0 and 1.0.

As evident from the above equation, T_S is a nonlinear function of discharge except when n = 1. A negative value of n is used when time of storage increases as discharge increases. According to the SSARR User's Manual, a value of n = 0.2 is reasonable for most streams in the Columbia River Basin.

7.2.1.8 Working R&D Routing

The Working R&D routing method (Figure 7.19) uses a nonlinear storage-outflow relation, like the Modified Puls method, but it adds the concept of wedge storage, like the Muskingum method. For a linear storage-outflow relation, the Working R&D method produces results identical to the Muskingum method. For routing with no wedge storage (Muskingum X = 0), the Working R&D method produces results identical to the Modified Puls method (USACE, 1994).

This method is not widely used. In fact, it has been so long since anyone attempted to apply it in ResSim that no one noticed that the Storage-Outflow table is nonfunctional. This will be corrected in or removed from future versions of the software.

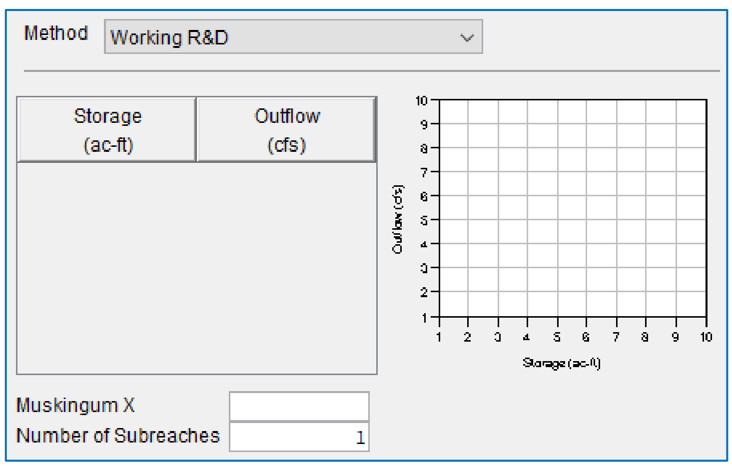


Figure 7.19 Reach Editor—Working R&D Routing Method

Storage and **Outflow**—Use a monotonically increasing set of storage and outflow values to describe the storage-outflow relationship needed for this routing method.

Muskingum X—The attenuation coefficient used in the wedge storage computations (from 0.0 to 0.5). A value of 0.0 indicates maximum attenuation of the hydrograph through the routing reach. A value of 0.5 indicates a "direct translation" of the hydrograph through the reach.

Number of Subreaches—The number of steps (subreaches) applicable for the routing reach. This parameter should be approximately equal to travel time divided by the computation interval.

7.2.1.9 Variable Lag & K

The **Variable Lag & K** method (Figure 7.20) is a hydrologic routing method for use between flow points that is based on a graphical routing technique. This method was added to ResSim at the request of the National Weather Service. **Lag & K** is a flexible method of storage routing, where both Lag and K values can be either constant or variable. **Lag & K** works by delaying the inflow hydrograph in time causing a *Lag* in the graph. The **K** value is used to attenuate the newly shifted curve.

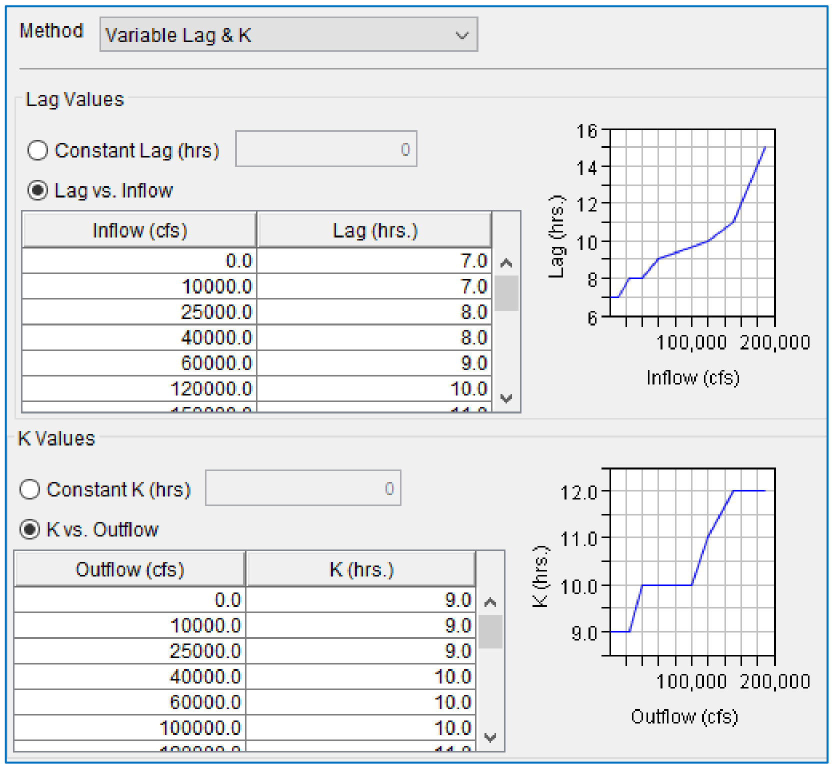


Figure 7.20 Reach Editor—Variable Lag & K Method

Constant Lag (hrs)—The travel time of the reach in hours.

Lag vs. Inflow—The **Lag vs Inflow** table is why the method is called "Variable". Use the **Lag vs Inflow** table to describe the relationship between Inflow and travel time.

Constant K (hrs)—the attenuation factor, in hours. The derivation of **K** is often given as a ratio of channel storage to discharge. It is related to the **K** value used in the **Muskingum** method.

K vs. Outflow—**K** can vary as a function of Outflow. Use the **K vs Outflow** table to enter the function; the Outflow values must be monotonically increasing. The thumbnail plot to the right of this table displays the data you entered in the table.

7.2.2 Reach Editor—Losses Tab

The **Losses** tab of the **Reach Editor** (Figure 7.21) provides two options for computing losses in the routing reach:

Constant Seepage—The flow (in cms or cfs) lost to the reach through the soil. This value will be constant throughout the simulation.

Seepage as a function of Flow—This option lets you define the loss as a function of the routed flow of the reach; flow values should be monotonically increasing. The thumbnail plot will reflect the values you enter in the table and can be viewed in full size when you double-click on it.

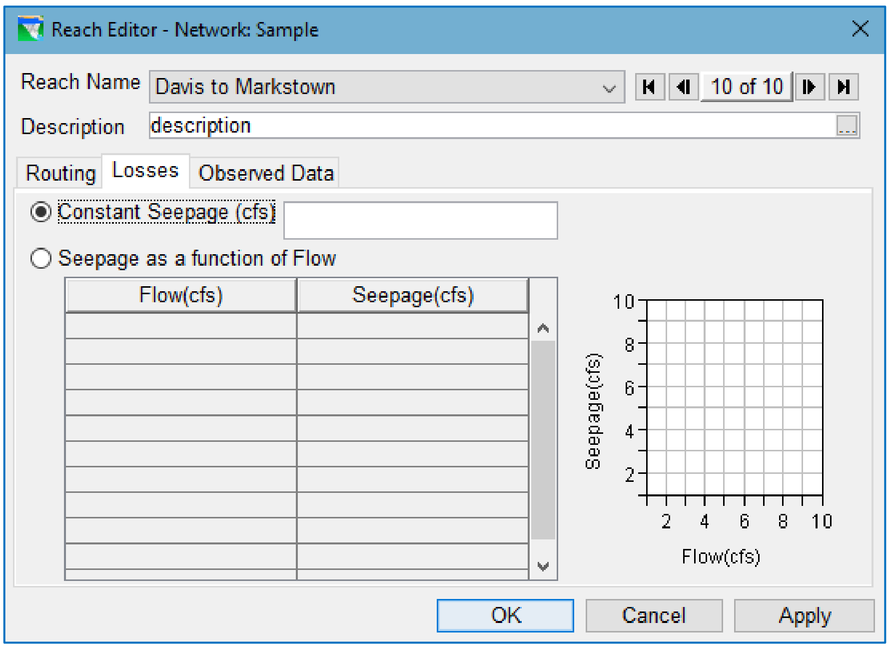


Figure 7.21 Reach Editor—Losses Tab

Reach losses are computed after the flow is routed through the reach; therefore, the final outflow from the reach is the routed flow minus the losses.

7.3 The Diversion Editor

Diversions remove water from your river system at junctions in your reservoir network.

If a diversion is connected at its downstream end to another junction in the network, that diversion is said to be "connected" and the downstream junction will receive the routed diversion flow as inflow. Connected diversions have a reach component to allow you to specify the routing and channel losses through the diversion reach. If a diversion is *not* connected at its downstream end, it is said to be "unconnected", and it removes the diversion quantity of flow from the network completely.

Diversions take priority over a reach or reservoir that may be connected to the outflow of the junction, so if there is not enough flow entering the diversion's source junction to meet the specified diversion quantity, the diversion will get all there is, and the downstream element will receive zero inflow. Although unusual, a diversion is allowed to have a negative diversion quantity; a negative diversion will be seen as inflow to the junction.

Use the **Diversion Editor** (Figure 7.22) to specify the diversion flow (or demand) for each diversion in your network.

The **Diversion Editor** has four tabs for defining the data related to your diversion:

- **Diversion**
- **Losses**
- **Routing**
- **Observed Data**

These tabs are described in the following sections.

NOTE: the **Losses** and **Routing** tabs are available only for connected diversions (diversions that connect to another junction at their outflow).

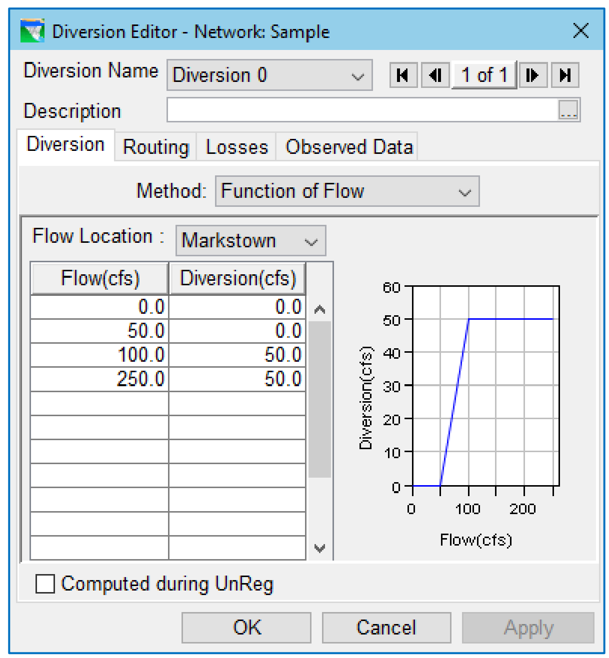


Figure 7.22 Diversion Editor

7.3.1 Diversion Editor—Diversion Tab

The Diversion Editor's **Diversion** tab (Figure 7.23) is where you will specify the diversion *demand*—the quantity of flow that the diversion will try to withdraw from its source (upstream) junction. This tab has two data entry fields: **Method** and **Computed during UnReg**, and an edit panel in between them.

Method—The **Method** selector offers seven options for specifying the diversion quantity:

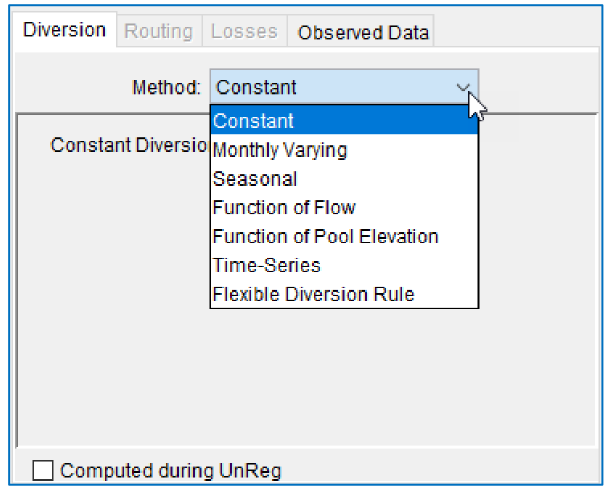


Figure 7.23 Diversion Editor—Diversion Tab

- Constant
- Monthly Varying
- Seasonal
- Function of Flow
- Function of Pool Elevation
- Time Series
- Flexible Diversion Rule

Select the diversion method from the **Method** list. The edit panel below the **Method** selector will fill with the appropriate fields to enable you to specify the parameters for the diversion method you have selected. The default method is **Constant**. The diversion methods are described in the following sections.

Computed during UnReg—Activate the **Computed during UnReg** checkbox if you want the diversion to be reflected in the unregulated flow computations. If checked, the diversion *will* operate during the unregulated flow calculations; if unchecked, the unregulated flow will reflect flow conditions *without* the diversion.

7.3.1.1 Diversion Method—Constant

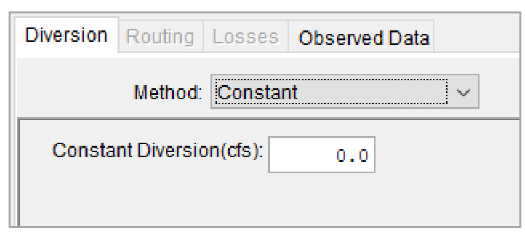


Figure 7.24 Constant Diversion Method

Use the **Constant** diversion method (Figure 7.24) when the desired withdrawal from the river system is always the same amount. The only parameter for this method is the flow rate of the diversion. This constant value will apply to each period of your simulation.

7.3.1.2 Diversion Method—Monthly Varying

Use the **Monthly Varying** diversion method (Figure 7.25) when the withdrawal from the river varies on a monthly basis. For this method, you will need to enter the diversion flow value corresponding to each month of the year. These monthly values will apply to each year in your simulation.

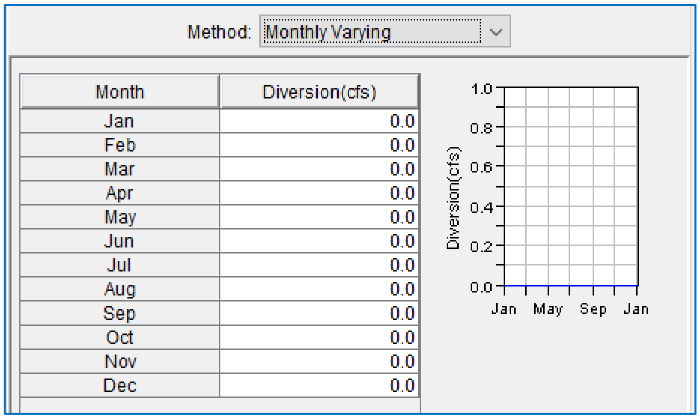


Figure 7.25 Monthly Varying Diversion Method

7.3.1.3 Diversion Method—Seasonal

Use the **Seasonal** method (Figure 7.26) when the withdrawal pattern varies as a function of date but not on a monthly basis. For each season, you will need to enter:

- a **Date**, in the format ddMMM (e.g., 01Jan), for the beginning of the season
- an associated **Diversion** flow value.

The seasonal table always starts on the first of January (01Jan), so if you have a season that spans across the calendar year boundary, be sure to specify the start of that season as the last entry in your table (as well as the first). Like the monthly varying diversion, the seasonal diversion is a pattern that repeats for each year of your simulation and each season's desired value applies until the start of the next season (i.e., step interpolation will be used between values in the table).

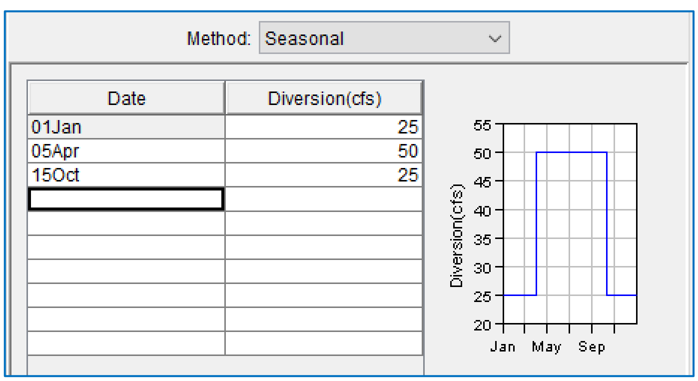


Figure 7.26 Seasonal Diversion Method

7.3.1.4 Diversion Method—Function of Flow

Use the **Function of Flow** diversion method (Figure 7.27) to describe the diversion demand as a function of flow in the river. Although for most function-of-flow diversions the location of the flow is usually the diversion's source junction, you can identify *any* junction in the network as the flow location for the function.

When specifying the data for this method, first select the **Flow Location** from the list of junctions in your network. Then, define the diversion function—the relationship between the Flow Location's **Flow** and the **Diversion** flow. Linear interpolation will be used between values in the table.

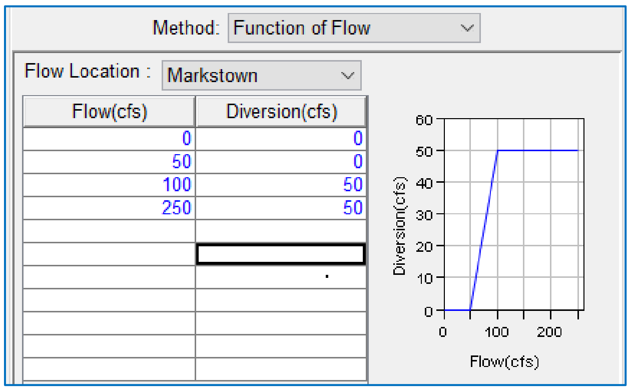


Figure 7.27 Function of Flow Diversion Method

7.3.1.5 Diversion Method—Function of Pool Elevation

Use the **Function of Pool Elevation** diversion method (Figure 7.28) to describe a diversion demand that varies with the pool elevation of a reservoir in your network. For this method, you will need to select the reservoir from the **Reservoir** list and then specify the relationship between that reservoir's pool **Elevation** and the **Diversion** flow. Linear interpolation will be used between values in the table.

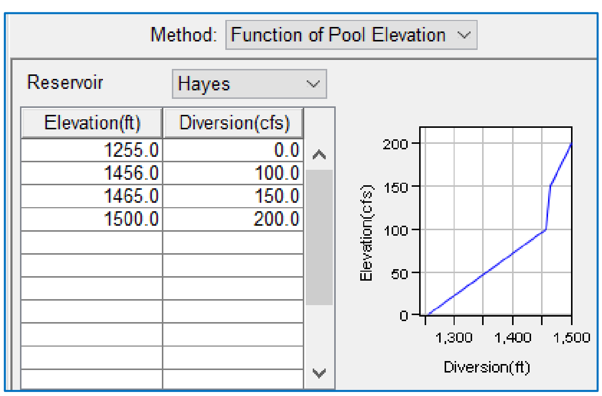


Figure 7.28 Function of Pool Elevation Diversion Method

7.3.1.6 Diversion Method—Time-Series

Use the **Time-Series** diversion method (Figure 7.29) for a diversion whose demand varies through time without a repeating pattern and without any relationship to flow or pool elevation.

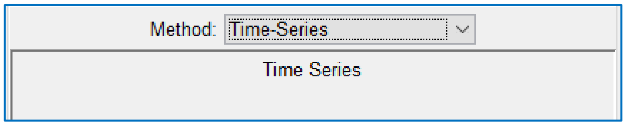


Figure 7.29 Time-Series Diversion Method

The edit panel for the **Time-Series** method has no data entry fields; instead you will be required to identify a time-series for this diversion when you setup your alternatives; an entry for this diversion will appear in the Time-Series mapping table when you create an Alternative (Chapter 13). The time-series you provide must be in standard units of flow (cfs or cms).

Tip You can also use the Time-Series diversion method when you want to change the diversion demand per alternative without having to change the network.

7.3.1.7 Diversion Method—Flexible Diversion Rule

The **Flexible Diversion Rule** diversion method (Figure 7.30) was added to allow the diversion demand to be specified as a function of any model variable or state variable and thus, make specification of the diversion demand as "flexible" as a reservoir release rule (see Chapter 11, Section 11.2). Although virtually every other diversion method could be *replaced* by this method, the older methods with their simpler interfaces are easy to understand and use.

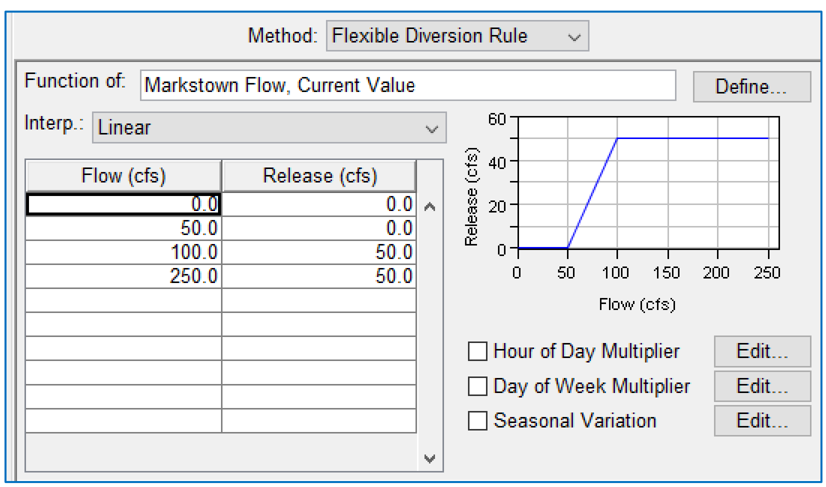


Figure 7.30 Flexible Diversion Rule Method

The data required to specify a "flexible diversion rule" depends on how you define the *function*. The common options for defining the rule are described below.

Function of—The first step in specifying a **Flexible Diversion Rule** is to identify the independent variable of the "function" (the diversion quantity is the dependent variable). To do so, select the **Define** button to the right of the **Function of:** field. The **Independent Variable Definition** editor will open (Figure 7.31); use this editor to select the type of variable your "Release is a function of". Options include: **Date**, **Date and Time**, **Model Variable**, **External Variable**, and **State Variable**. See Appendix C for a description the Independent Variable Definition editor and of each of the variable types.

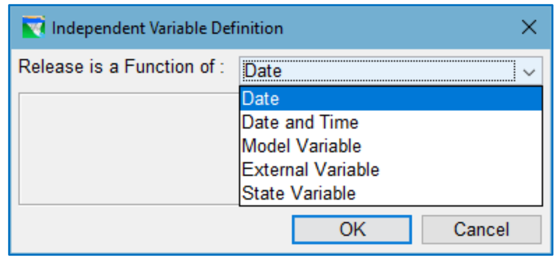


Figure 7.31 Select Independent Variable

Interp—Select an *interpolation method* to be used between values in the function table. Options include **Linear**, **Step**, and **Cubic**.

Hour of Day Multiplier—The primary purpose of the **Hour of Day Multiplier** is to turn the diversion on and off at specific times of the day, but it can also be used to vary the diversion demand throughout the day.

To specify a set of hourly multipliers to be applied to the diversion demand, select the **Hour of Day Multiplier Edit** button. The **Hour of Day Multiplier** dialog will appear (Figure 7.32); this dialog has a table listing each hour of the day and an associated multiplier. The default multiplier is 1.0.

For example, if you have a diversion that only requires flow from 6 a.m. to 4 p.m., but the morning hours require twice as much flow as the afternoon hours, you could modify the multipliers as shown in Figure 7.32. The multiplier is 0.0 for the hours from 0000 to 0600 and from 1600 to 2400 (to turn off the diversion during those hours) and 0.5 for the hours from 1200 to 1600. Alternatively, you could enter 2 for the hours from 0600 to 1200 —the choice between applying 0.5 for the afternoon or 2 for the morning is dependent on the values you specified in the diversion function that the multiplier will be applied to.

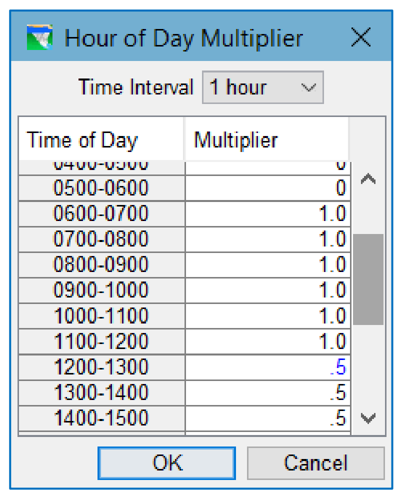


Figure 7.32 Hour of Day Multiplier with Example Pattern

Click **OK** to apply your entries and close the **Hour of Day Multiplier** dialog. A checkmark will appear in the checkbox in front of the Hour of Day

Multiplier label in the diversion edit panel when the default set of multipliers is modified.

Time Interval—A selector for **Time Interval** is available in the **Hour of Day Multiplier** dialog. This selector defaults to **1 hour**, but options for intervals of **1**, **2**, **3**, **4**, **6**, and**12 hours** are available; if you pick another interval, the **Hour of Day Multiplier** table will change to reflect your interval selection. For example, if you chose **12 hours**, the Multiplier table would have only two rows, 0000-1200 and 1200-2400. If you placed 0.0 in the first row and 1.0 in the second, your diversion would divert flow only between noon and midnight.

Day of Week Multiplier—The purpose of the **Day of Week Multiplier** is to turn the diversion on or off on specific *days of the week*, but it can also be used to vary the diversion demand throughout the week.

Select the **Day of Week Multiplier Edit** button to specify a set of daily multipliers to be applied to the diversion demand. The **Day of Week Multiplier** dialog will appear (Figure 7.33). This dialog contains a table where you can specify a multiplier for each day of the week. The default value is 1.0.

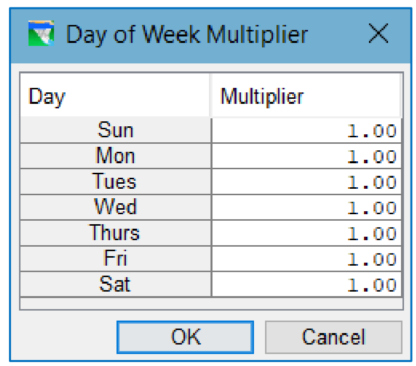


Figure 7.33 Day of Week Multiplier

So, if you have a diversion that diverts only on weekdays, you can set the multiplier for Saturday and Sunday to 0.0 and leave all other days set to 1.0.

Click **OK** to close the **Day of Week Multiplier** dialog. If the table contains any non-default values, a checkmark will appear in the **Day of Week Multiplier** check box in the diversion edit panel

Seasonal Variation—The **Seasonal Variation** option allows you to add **Date** (or **Season)** as a *second* independent variable to your **Function of** relationship, but it is only active when your first independent variable is **Model Variable**, **External Variable**, or **State Variable** (i.e., a time-series).

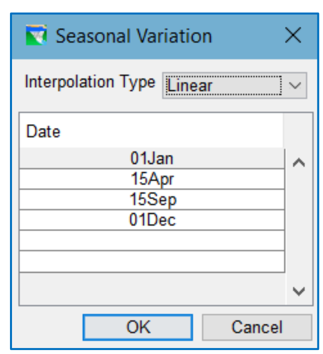


Figure 7.34 Seasonal Variation

So, if your diversion demand varies as a function of a time-series variable *and* seasonally, select the **Seasonal Variation Edit** button to specify the seasons. The **Seasonal Variation** dialog (Figure 7.34) will appear. This dialog contains a seasonal table in which you can enter dates that represent the start of each "season". Seasonal tables always start on **01Jan**. If you have a season that crosses the calendar year boundary, it will be entered as two seasons, one at the beginning of the year and one at the end of the year (the first and last seasons in the table).

The **Seasonal Variation** dialog also provides an option for **Interpolation Type**. You can select from: **Linear**, **Cubic**, and **Step**. *This* interpolation selection applies to the seasonal variation of your function (the columns of your table).

Click **OK** to close the **Seasonal Variation** dialog. If you created more than one season, a checkmark will appear in the **Seasonal Variation** check box in the diversion edit panel and a column will be added to the function table for each season specified in the Seasonal Variation dialog.

7.3.1.7.1 Flexible Diversion—Function of: Date

If the diversion demand *varies seasonally* (as a function of date), select **Date** from the **Release is a Function of:** list in the **Independent Variable Definition** editor for your **Flexible Diversion Rule**'s function. A seasonal function table will appear in the diversion's edit panel as illustrated in Figure 7.35. All seasonal tables in ResSim start on **01Jan**. You can enter a single demand value for **01 Jan** to describe a constant diversion amount throughout the year or enter the appropriate dates and diversion flows to specify the relationship between time of year and the desired diversion flow (release). Be sure to specify the **Interp**olation type to indicate how to determine the value to be returned when the value of the independent variable is between values in the table.

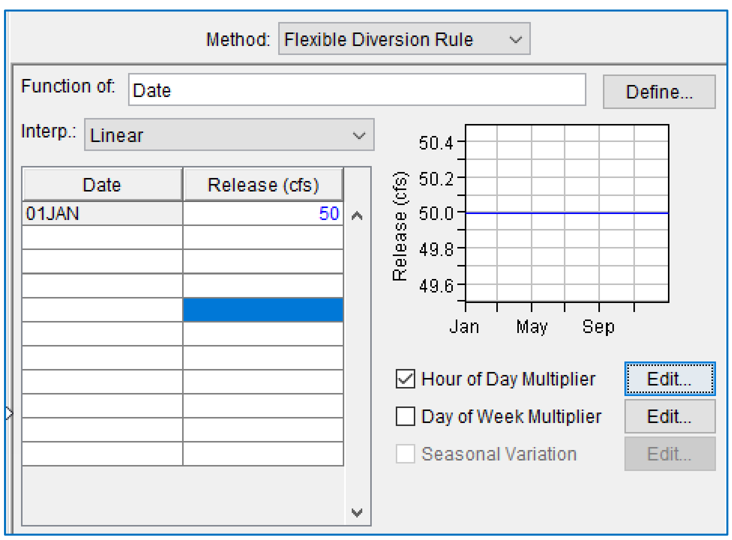


Figure 7.35 Flexible Diversion—Function of Date

You can use the ***Flexible* Function of Date** diversion method to represent a **Constant** (Section 7.3.1.1), **Monthly Varying** (Section 7.3.1.2) and **Seasonal** (Section 7.3.1.3) diversion methods by selecting **Step** for the **Interp**olation method. However, by using **Linear** interpolation and/or **Hour of Day** and **Day**

of Week Multipliers, your flexible diversion function can be defined in ways the original methods could not.

All seasonal tables in ResSim treat each date entered as the beginning of the day, i.e., as applying at 0000 hours. If you would prefer that it applied at the end of the day, use the next day or use a table that allows you specify Time as well as Date.

7.3.1.7.2 Flexible Diversion—Function of: Date and Time

The ***Flexible*** **Function of Date and Time** diversion function is almost exactly the same as the ***Flexible*** **Function of Date** diversion function — with one difference—the addition of a **Time** column in the function table — see Figure 7.36. Use this function type when the demand varies with *both* date and time. When entering time values, use a 24-hour clock, e.g. 6 pm is 1800.

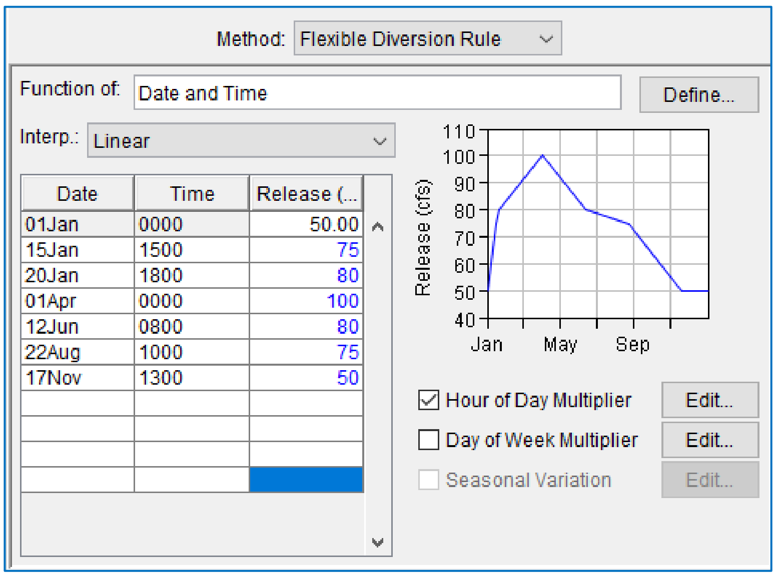


Figure 7.36 Flexible Diversion—Function of Date & Time

7.3.1.7.3 Flexible Diversion—Function of: Model Variable

If the diversion demand *varies as a function of a standard variable computed by ResSim*, you can use the Model Variable form of the Flexible Diversion rule. To set up:

- Select **Model Variable** from the **Release is a Function of:** selector in the **Independent Variable Definition** dialog. The features of this dialog are described in detail in Appendix C.

  The edit panel of the **Independent Variable Definition** dialog (Figure 7.37 will fill with three **Filter** fields above a table containing all the model variable time-series computed by ResSim *for the current network*. To the right of the model variables table is a section labeled **Time Series Options**.

- Find and highlight the appropriate model variable time-series for your function in the table, then click the **Select** button (Figure 7.37). Your selection will appear in the **Selected Model Time-Series** field at the bottom.

> Note: double-clicking on the variable you want will NOT perform the selection; you must use the Select button.

- Next, specify the **Time Series Options** to indicate which value to use from the selected time series at each timestep of the simulation.
- Click **OK** when you have selected the Model Variable and Time Series Options for your function. The **Flexible Diversion Rule** edit panel will now contain a table for you to enter the relationship between the specified External Variable and the diversion demand.

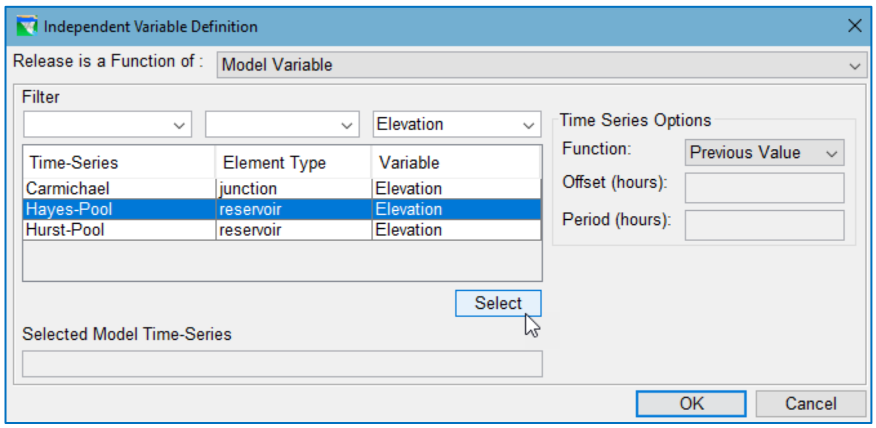


Figure 7.37 Flexible Diversion—Function of Model Variable

Be sure that the model variable time-series you selected appears in the **Selected Model Time-Series** field at the bottom of **Independent Variable Definition** dialog before clicking the **OK** button in the to apply your settings, close the dialog, and return to the **Diversion Editor.** The **Flexible Diversion Rule** edit panel will now contain a table for you to enter the relationship between the model variable and the diversion demand.

7.3.1.7.4 Flexible Diversion—Function of: External Variable

When the diversion demand *varies as function of an external time-series, select* **External Variable** from the **Release is a Function of:** list in the **Independent Variable Definition** dialog (Figure 7.38).

When **External Variable** is selected, the edit panel of the **Independent Variable Definition** dialog will fill with a **Variable Name** text field and the **Time Series Options**. The name you provide will appear in the **Alternative Editor's Time Series** tab where you will be expected to associate an HEC-DSS time-series dataset with this variable. Enter an appropriately descriptive variable name so that you know what time-series to use for each External Variable to specify in your model.

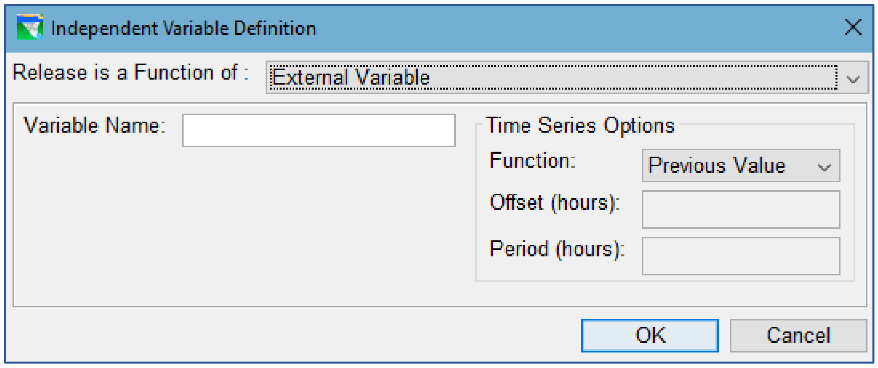


Figure 7.38 Flexible Diversion—Function of External Variable

The **Time Series Options** apply to the **External Variable** time-series in exactly the same way as they do for a **Model Variable** time-series. See Appendix C for details on these options.

Click the **OK** button in the **Independent Variable Definition** dialog to apply your settings, close the dialog, and return to the **Diversion Editor.** The **Flexible Diversion Rule** edit panel will now contain a table for you to enter the relationship between the specified External Variable and the diversion demand.

7.3.1.7.5 Flexible Diversion—Function of: State Variable

When the diversion demand *varies as a function of a user-defined (scripted)* ***State Variable***, select **State Variable** from the **Release is a Function of:** list in the **Independent Variable Definition** dialog (Figure 7.39).

When **State Variable** is selected, the edit panel of the **Independent Variable Definition** dialog will fill with a selection list showing all the state variables defined in the current network and the **Time Series Options**. Select the appropriate state variable from the selection list. Note: the state variable must be defined prior to creating a flexible diversion function that uses it (see how to create **State Variables** in Section 12.2).

The **Time Series Options** apply to the **External Variable** time-series in exactly the same way as they do for a **Model Variable** time-series. See Section 7.3.1.7.3 for details on these options.

Click the **OK** button in the **Independent Variable Definition** dialog to apply your settings, close the dialog, and return to the **Diversion Editor.** The **Flexible Diversion Rule** edit panel will now contain a table for you to enter the relationship between the specified **State Variable** and the diversion demand.

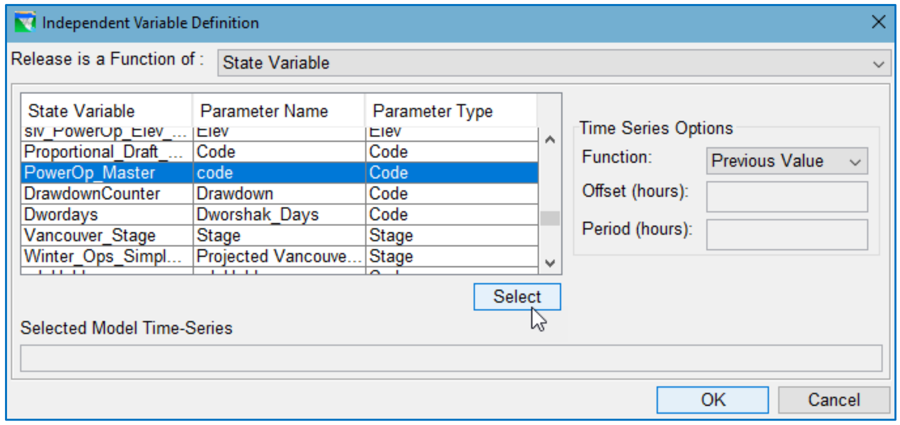


Figure 7.39 Flexible Diversion—Function of State Variable

7.3.2 Diversion Editor—Routing Tab

The **Diversion Editor's Routing** tab (Figure 7.40) allows you to define the routing for the diversion channel. The **Routing** tab is only active for diversions that are connected to a downstream junction. The routing methods available to a diversion are the same as those available to a reach; they are: **Null**, **Coefficient**, **Muskingum**, **Muskingum-Cunge 8-pt Channel**, **Muskingum-Cunge Prismatic Channel**, **Modified Puls**, **SSARR**, Working R&D Routing, and **Variable Lag & K**; each with its own set of parameters. If there is no lag or translation of the hydrograph through the diversion reach, you can select **Null Routing** (which is the default routing method).

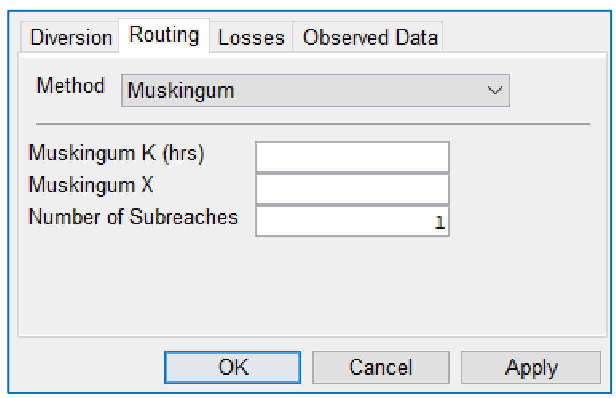


Figure 7.40 Diversion Editor—Routing Tab

The **Routing** tab of the **Diversion Editor** is identical to the **Routing** tab of the **Reach Editor**. For details about using specifying the routing method on the **Routing** tab, refer to Section 7.2.1.

7.3.3 Diversion Editor—Losses Tab

The **Diversion Editor's Losses** tab (Figure 7.41) allows you to define losses from the diversion channel. Like the Routing tab, the Losses tab is only active for diversions that are connected to a downstream junction. Three options for specifying the channel losses are provided: **Return Ratio**, **Constant Seepage**, or **Seepage as a function of Flow**. Whichever loss method you used, the loss is applied to the *routed* flow from the diversion channel, right before it enters the downstream junction.

Return Ratio—to describe a consumptive use diversion where only a fraction of the diverted water actually returns to the network, specify a **Return Ration** between 0.0 and 1.0 (the default is 1.0) to indicate how much of the diverted flow reaches through the diversion to the downstream junction.

Constant Seepage—Enter a seepage value (in cfs or cms) to indicate how much diversion flow will "seep into" the ground and therefore be lost from the system.

Seepage as a function of Flow—Enter a Flow vs. Seepage relationship (in cfs or cms) In the Flow vs. Seepage table to indicate how much of the diversion flow is lost before reaching the downstream junction. The thumbnail plot to the right of the table will reflect the values you enter and can be viewed in full size when you double-click on it.

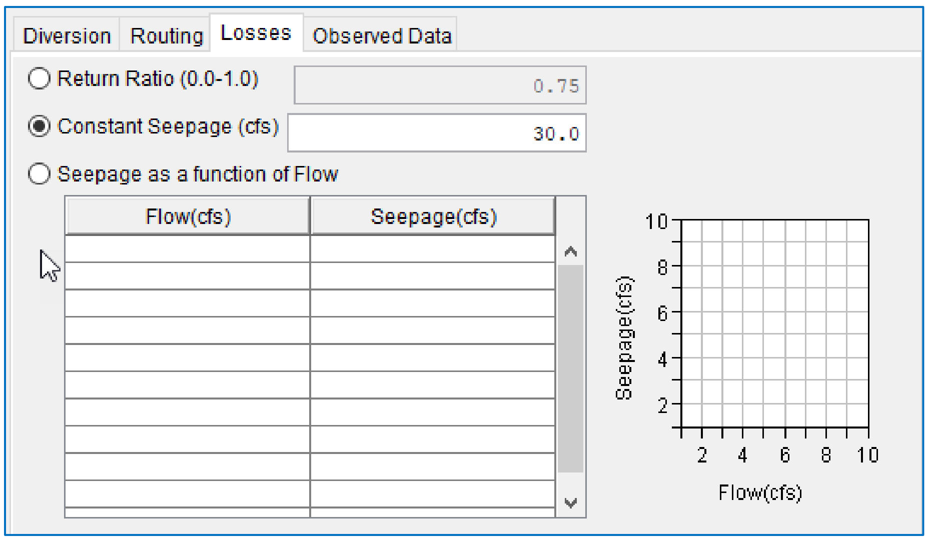
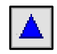


Figure 7.41 Diversion Editor—Losses Tab

Chapter 8
The Reservoir Editor

The **Reservoir Editor** (Figure 8.1) is used to specify both the physical and operational data for a reservoir through the use of two tabs—a **Physical** tab and an **Operations** tab.

To open the **Reservoir Editor**:

- Select **Reservoirs...** from the **Edit** menu (of the **Reservoir Network** or **Simulation** module). OR
- Using the **Pointer**  or **Reservoir Tool** , *right-click* on a reservoir in the Map Display area, then select **Edit Reservoir Properties** from its context menu.

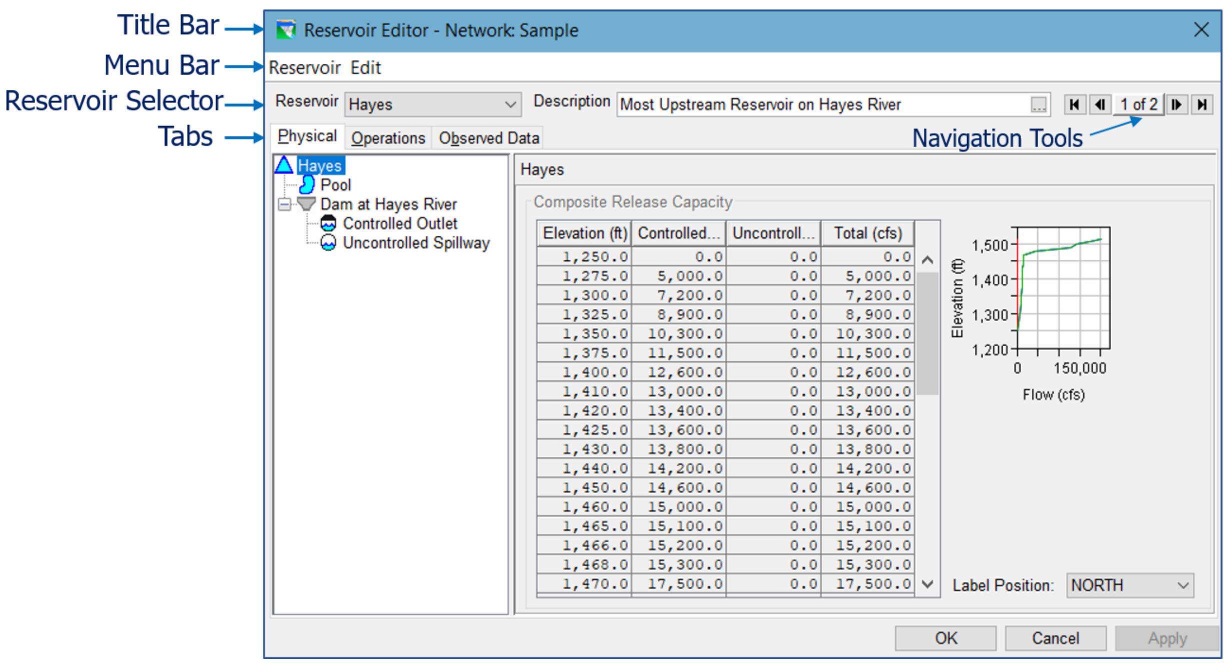


Figure 8.1 Reservoir Editor—Physical Tab—Annotated

Figure 8.1 shows an annotated version of the **Reservoir Editor**. From top down, the **Reservoir Editor** is made up of:

A **Title Bar**—which displays the name of the editor followed by the name of the current reservoir network.

A **Menu Bar**—which has a **Reservoir** (File) menu and an **Edit** menu. The Menu Bar may also display one or more additional menus depending on your most recent action or selection in the editor.

A **Navigation Bar**—which provides features to selecting the reservoir for editing. The **Navigation Bar** includes:

A **Reservoir Selector** that you can use to select the reservoir you want to edit

A **Description** textbox that displays the description you entered when you created the reservoir. The ellipsis button at the end of the text box will open a description edit dialog.

A set of **Navigation Tools** for switching between reservoirs in the Editor.

A set of **Tabs** including:

Physical tab—for defining the physical properties of a reservoir. The Physical Tab divided into two parts: the **Reservoir Tree** is at the left and an **Edit Panel** is to the right.

Operations tab—for defining the zones and rules and other features that describe the reservoir's operating constraints

Observed Data tab—for selecting the computed model variables that you may want to compare to an observed or external time series when reviewing results through ResSim's standard reservoir plots.

A set of **Save & Close Buttons** including:

Ok—will save the changes made to the current reservoir and close the Editor.

Cancel—will close the Editor and discard (not save) any changes made to the current reservoir.

Apply—will save the changes made to the current reservoir but will not close the Editor. It is a good practice to press **Apply** before switching to another tab or reservoir.

8.1 Reservoir Editor—Physical Tab

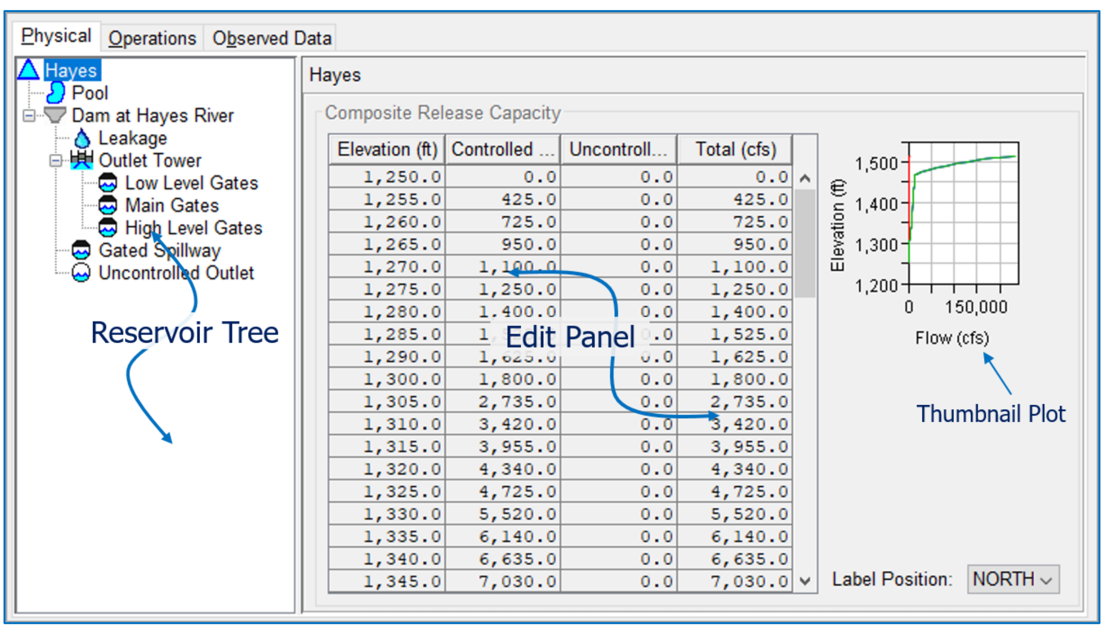


Figure 8.2 Reservoir Editor—Physical Tab

8.2 Reservoir Editor—Operations Tab

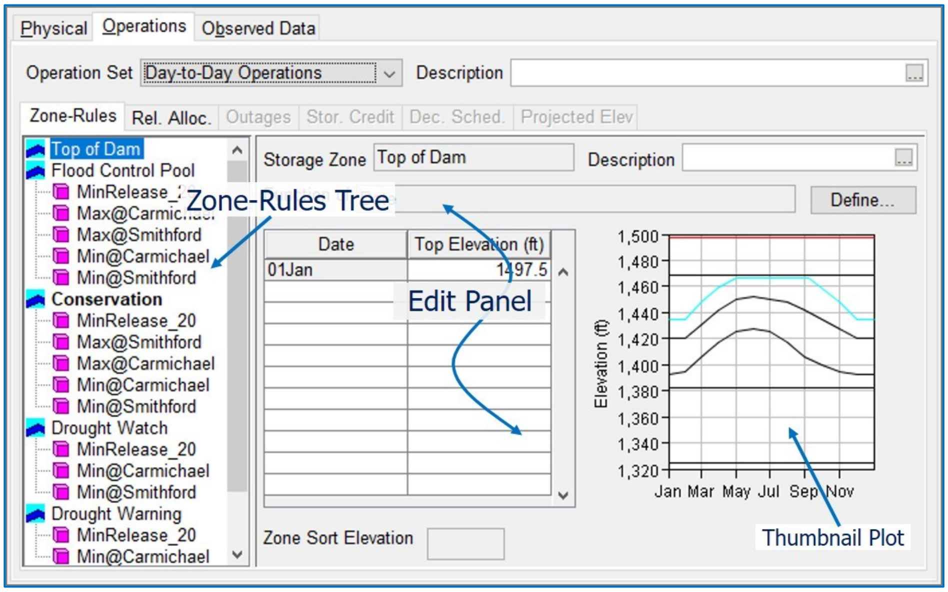


Figure 8.3 Reservoir Editor—Operations Tab

8.3 Reservoir Editor—Observed Data Tab

Use the **Observed Data** tab (Figure 8.4) to indicate that observed data is available for comparison purposes in ResSim's standard plots. If the **Observed** box in the table is checked for a given model variable, then there will be a corresponding entry in the Time-Series mapping table on the Observed Data tab of the Alternative Editor (Section 14.10) when you create or edit an alternative using this network.

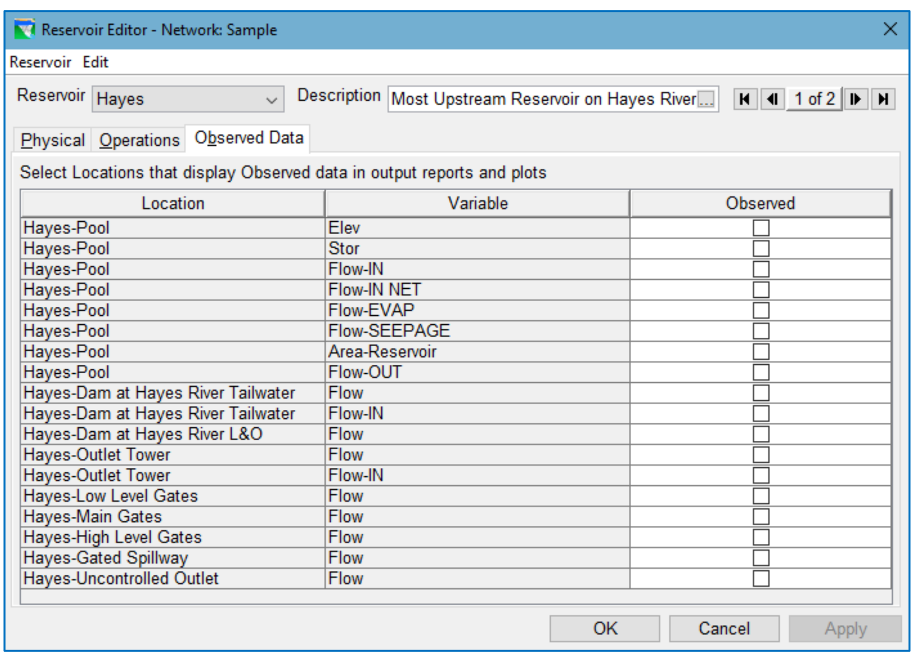


Figure 8.4 Reservoir Editor—Observed Data Tab

Chapter 9
The Physical Properties of Reservoirs

Once you have created a reservoir network and established its connectivity (as described in Chapter 6), you will need to specify the properties of the elements in the network. To do so, you will use ResSim's specialized network element editors. The four major element editors all share some common features; see the introductory section of Chapter 7 for a description of these common features. Chapter 7 then goes on to describe the **Junction**, **Reach**, and **Diversions Editors** and how to enter and edit data for their respective network elements.

The data needed to define a reservoir element is divided into two types: Physical and Operational. This chapter describes the physical components of a reservoir and how to use the **Physical** tab of the **Reservoir Editor** (Figure 9.1) to define them. Subsequent chapters will describe the options for defining the reservoir operating objectives and constraints. Chapter 10 describes the basics for defining reservoir operating objectives and constraints, and Chapter 11 describes operational rules. Chapter 12 describes the advanced operational features such as IF_Blocks and scripts. Two or more reservoirs can be made to operate in conjunction by defining them as a Reservoir System. Reservoir Systems are covered in Chapter 13.

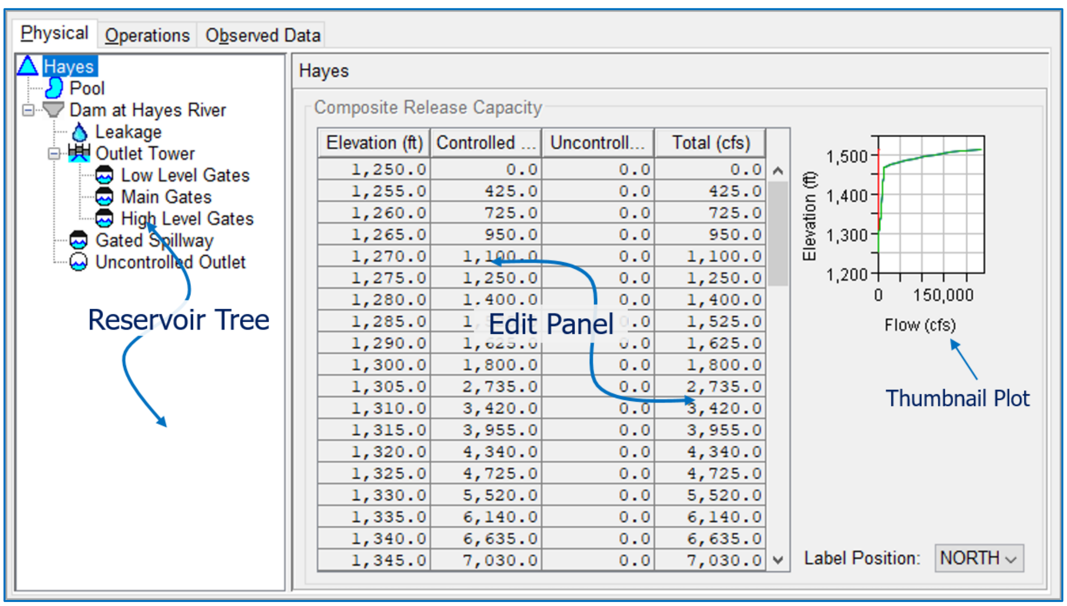


Figure 9.1 Reservoir Editor—Physical Tab

9.1 The Reservoir Tree

In ResSim, reservoirs are complex elements that are made up of a variety of other elements (or components). The elements that make up a reservoir fall into two categories — release elements and property elements. A Release Element is a physical feature of a dam or reservoir that can take water away from the reservoir pool and put it somewhere else in the network, usually into the downstream river system. The *release elements* include the reservoir itself, the dam, all the outlet types, outlet groups, and diverted outlets. A property element describes some physical aspect of the reservoir, such as the size and shape of the reservoir pool. The *property* components include the pool, evaporation, seepage, leakage, tailwater elevation, and forebay head loss.

The **Physical** tab of the **Reservoir Editor** uses a "tree structure" to represent the hierarchical relationship between the reservoir and its elements. This tree structure is called the **Reservoir Tree** and is shown at the left side of the **Physical Tab** of the **Reservoir Editor** (see Figure 8.1).

The **Reservoir** node is the root of the **Reservoir Tree**. By default, each reservoir has two *elements*:

the **Pool** and

the **Dam**

which appear as nodes immediately below the **Reservoir** in the **Reservoir Tree**.

Other *Elements* that can be added to the **Reservoir Tree** include:

Controlled Outlets,

Uncontrolled Outlets,

Power Plants ,

Pumps,

Outlet Groups,

Diverted Outlets,

Evaporation, **Seepage**, and **Leakage** losses, and

Tailwater Elevation and **Forebay Head Loss** specifications.

When an element is added to a node in the Reservoir Tree, the receiving node spawns a branch in the tree (if it doesn't already have one) and the new element becomes a node on the branch. Almost all components of a reservoir are added to the **Pool**, the **Dam**, an **Outlet Group**, or a **Diverted Outlet** branch of the **Reservoir Tree**. Element nodes cannot be added to the reservoir node itself — with one exception — a diverted outlet. See Section 9.5.1 for details on adding a diverted outlet to the reservoir.

But the **Reservoir Tree** is more than just an illustration of the reservoir hierarchy; it is the user interface mechanism you will use to access the data for the various components of a reservoir.

When you select an element (node) in the **Reservoir Tree**…

The **Edit Panel** to the right of the Reservoir Tree will display the *edit pane* for the selected component. At the top of each edit pane is the fully-qualified name of the selected element. The rest of the edit pane will contain the data entry fields for specifying the data needed to define the selected element.

An *element menu* will appear in the **Menu Bar** of the **Reservoir Editor.** For example, when you select the **Dam** in the reservoir tree, a **Dam** menu appears in the menu bar. Each element menu contains options pertinent to the selected element in the Reservoir Tree.

And, when you *right-click* on any element in the Reservoir Tree, a context menu will appear, giving you access to the same options that are available from the element menu that will appear in the Menu Bar.

The order in which you add elements to the reservoir (tree) and edit their properties is flexible. You can add all the physical elements to the reservoir then go back and define each element's properties, or you can define the properties of each element as you add it, before going on to add and edit the next element. Choose the style that works best for you. Just be sure to finish defining the physical representation of your reservoir before going on to define its operational data (Chapter 10), as the definition of the operational zones depends on the physical definition of the pool and the rules depend on the description of the outlets.

9.2 The Reservoir

When you select the **Reservoir** node in the Reservoir Tree, the **Edit Panel** displays the Reservoir pane (Figure 9.2).

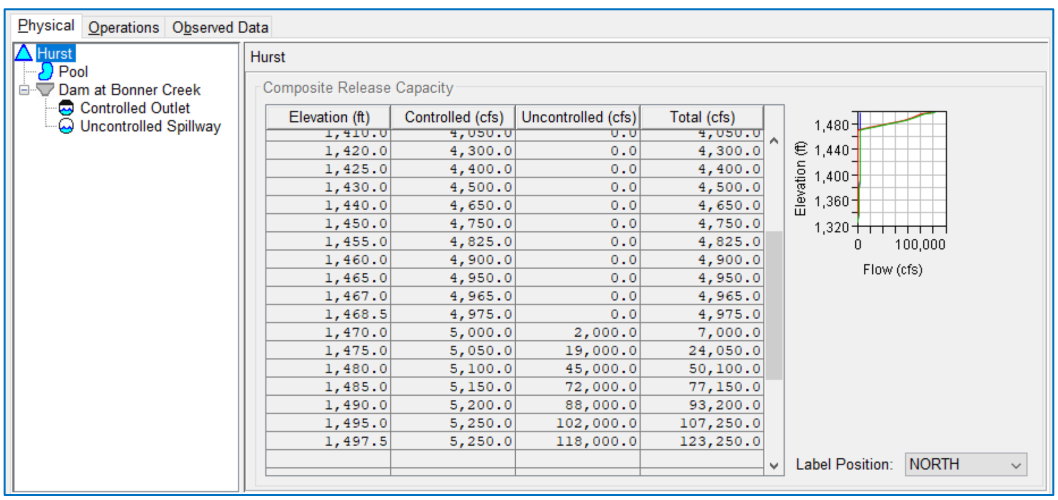


Figure 9.2 Reservoir Editor—Physical Tab—Reservoir Pane

Below the selected reservoir's name, the Reservoir pane contains a non-editable table listing the release capacity of the reservoir. The thumbnail plot to the right of the **Composite Release Capacity** table displays three curves, one for each release capacity column shown in the table. The **Controlled** capacity curve is blue, the **Uncontrolled** capacity curve is red, and the **Total** capacity curve is green.

Only one *editable* field is available on the Reservoir pane—it is the **Label Position** selector in the bottom right corner. Use this selector to specify where you want the reservoir label will to display in the Map Display area relative to the current reservoir's label handle. Refer to Section 5.2.2 to learn how to identify the Reservoir's label handle.

9.3 The Pool

The **Pool** is the reservoir element in which you specify the characteristics of the reservoir's storage though an elevation-storage-area relationship.

When you select the **Pool** node in the Reservoir Tree, the **Edit Panel** displays the Pool pane (Figure 9.3). This pane contains a table in which you must enter the reservoir's elevation-storage-area (ESA) relationship. Before entering data in the storage table, select one of the two options provided above the table to indicate the type of ESA relationship you will enter; your options are **Linear Interpolation** or **Conic Interpolation**.

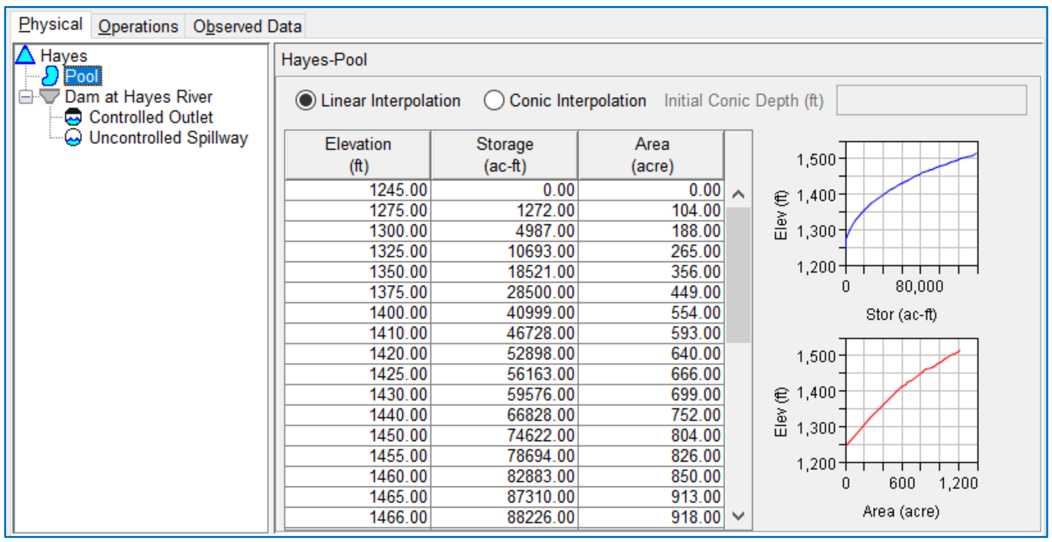


Figure 9.3 Reservoir Editor—Pool Pane

Linear Interpolation—By default, the interpolation method ResSim uses to compute values for elevations and storages between those specified in the table is linear. With this method, area data is optional unless you add evaporation losses to the pool.

Conic Interpolation—This method requires that you specify a series of values for **Elevation** and pool surface **Area** to describe the reservoir pool. Also required is a value for starting storage at the lowest elevation in the table and the **Initial Conic Depth** (the depth from the first storage value down to zero storage). Using data you provide, ResSim will compute the storage between each sequential pair of elevations using the equation for the volume of a slice of a cone. Conic interpolation will also be used to obtain intermediate storage values for elevations between those explicitly entered in the table (using a linearly interpolated area at each elevation of interest).

The pool definition detailed in the regulation manual for most reservoirs was originally developed using conic interpolation. The area of the pool surface at a regular interval of elevation was computed based on the topography of the land.

The following are requirements when specifying the **Elevation, Storage**, and **Area** data:

- A minimum of two rows (values) must be entered.
- Values in each column must increase down the columns (i.e., lowest elevation in the top row, highest elevation in the bottom row).
- No duplicate values.

To enter data into the table, either copy-and-paste it from a spreadsheet or type it in manually. The two thumbnail plots to the right of the storage table illustrate the elevation-storage curve and the elevation-area curve represented by the data you provided in the table. You can open a full-size plot from any thumbnail plot by double-clicking on it.

NOTE: Elevation and Storage data are required for definition of the reservoir pool. Surface area data is *optional* unless you are defining storage using conic interpolation or you added evaporation losses to the pool.

9.3.1 Pool Losses

Two types of pool losses can be added to a reservoir pool—**Evaporation** and **Seepage.** Both types of losses remove water from the reservoir pool, never to be seen again in the model. In other words, ResSim does not continue to account for the water lost to evaporation or seepage nor does the water return to the system in any way.

When you add **Evaporation** and/or **Seepage** to the reservoir pool, they will appear as nodes in the **Pool** branch of the Reservoir Tree (Figure 9.4). Since only one **Evaporation** or **Seepage** node may be added to a reservoir pool, there is no need nor option to rename it.

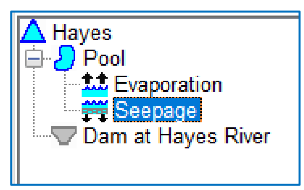


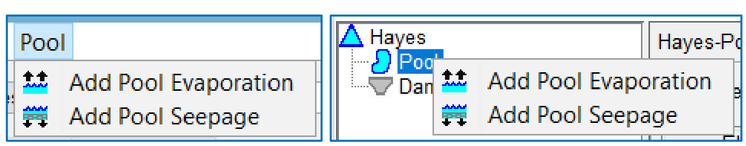
Figure 9.4 Reservoir Tree with Pool Losses

9.3.1.1 Evaporation

Evaporation is a loss of water from the reservoir pool to the atmosphere. It is entered as a rate in units of depth per period of time. ResSim gives you two ways to specify evaporation — as a monthly table where you specify the number of inches (or millimeters) per month or as a time series. Computed evaporation is output as a rate in units of flow (cfs or cms).

To add **Evaporation** to the reservoir pool:

- *Right-click* on the **Pool** in the **Reservoir Tree** on the **Physical** tab of the **Reservoir Editor** (Figure 9.5).



Pool Menu in Menu Bar Pool Context Menu from Reservoir Tree

Figure 9.5 Pool Menu Option

- Select **Add Pool Evaporation** from the **Pool**'s context menu (Figure 9.5).
- An **Evaporation** node will appear in the Reservoir Tree as the active node and the Edit Panel will display the **Evaporation** edit pane as shown in Figure 9.6.

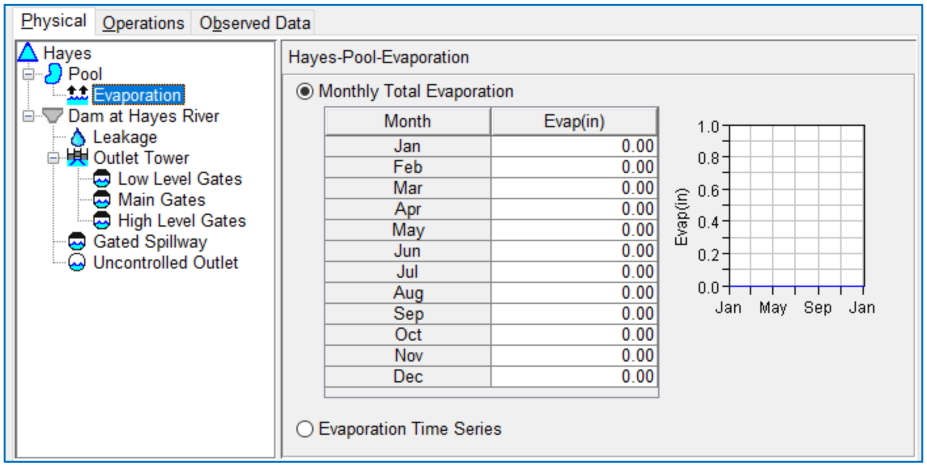


Figure 9.6 Reservoir Editor—Evaporation

- Select the type of **Evaporation** you want to specify.
  - For **Monthly Evaporation**, enter a value for the seepage rate in cfs (or cms) in the text box for each month. The thumbnail plot to the right of the table will reflect the values you enter in the table.
  - For **Evaporation Time Series**, no further information is required in the Evaporation edit pane. Instead, when you set up an alternative that uses this network, an entry for evaporation will appear in the Time Series table. There you must identify the evaporation rate time series for the reservoir.

NOTE: in addition to entering evaporate rate information, reservoir surface area is also required to model evaporation. Be sure to enter area information in the elevation-storage table for your reservoir pool.

9.3.1.2 Seepage

Seepage is loss of water from the reservoir pool to the ground. It is entered and reported as a rate in units of flow (cfs or cms). ResSim gives you two ways to specify Seepage—as a constant rate or as a function of pool elevation.

To add **Seepage** to the reservoir pool:

- *Right-click* on the **Pool** in the **Reservoir Tree**.
- Select **Add Pool Seepage** from the **Pool**'s context menu in the Reservoir Tree (Figure 9.5).
- A **Seepage** node will appear in the Reservoir Tree as the active node and the Edit Panel will display the **Seepage** edit pane as shown in Figure 9.7.

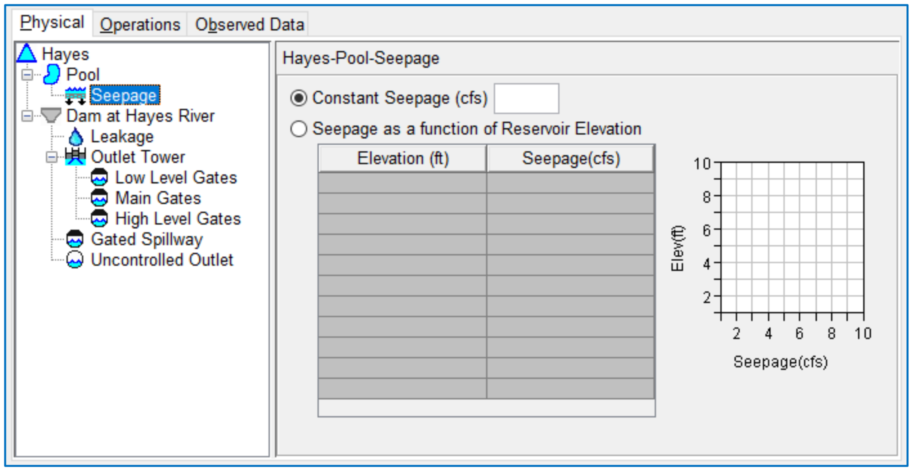


Figure 9.7 Reservoir Editor—Seepage

- Select the type of **Seepage** you want to specify.
  - For **Constant Seepage**, enter a single value for the seepage rate in cfs (or cms) in the text box.
  - For **Seepage as a function of Reservoir Elevation**, enter the relationship of **Elevation** vs. **Seepage** into the table. The data must be entered in increasing values of elevation.

9.4 The Dam

The **Dam** is the reservoir element that can move water from the reservoir pool to the downstream system and is the primary element to which you will add outlets and outlet groups. In the **Reservoir Tree**, the dam is a special type of outlet group. What makes the dam *special* are three features that standard outlet groups don't have:

1. The dam is attached directly to the reservoir
2. The dam can be assigned a leakage element
3. The dam is the default release element of a reservoir — an overflow weir. Even if you do not add outlets to it, the dam itself is a weir that ResSim models using the standard weir equation and a (hard-coded) weir coefficient of 3.0. To define the weir, you must provide:

   Elevation at top of dam—the elevation at which overtopping of the dam will occur (i.e., the weir crest).

   Length at top of dam—the length of the dam or the length of the section of the dam that will overtop (the weir length).

Tip If you do not want ResSim to compute dam overflow or if you want to specify the overflow capacity using a separate outlet, enter zero for the length of the dam.

The computed *overflow* from the dam will be combined with leakage and will be appear in the output as "Dam L&O".

When you select the **Dam** node in the Reservoir Tree, the **Edit Panel** displays the Dam pane (Figure 9.8)

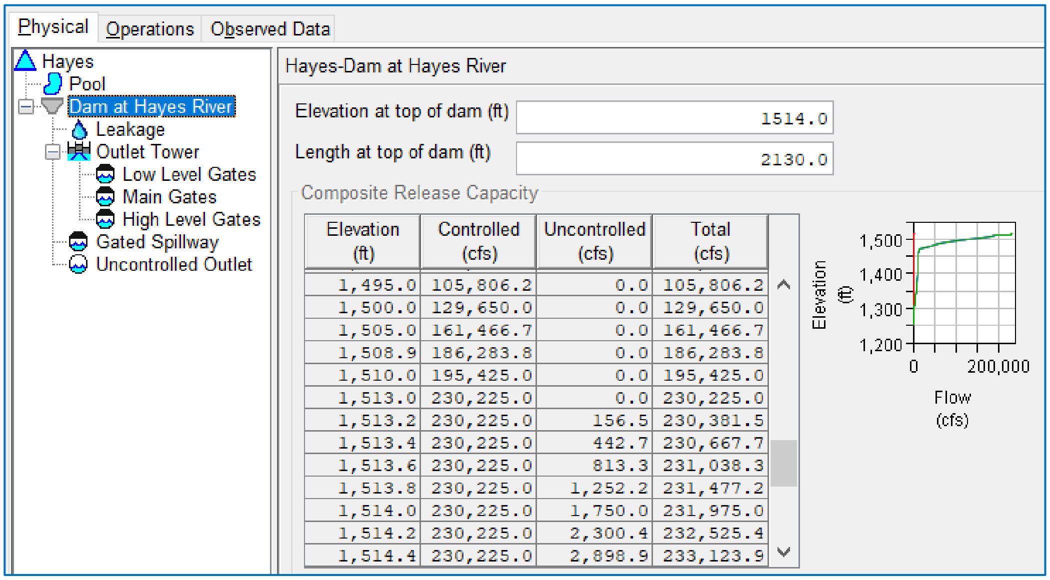


Figure 9.8 Reservoir Editor—The Dam

Below the name of the Dam, the Dam pane contains a non-editable table displaying the **Composite Release Capacity** of the dam. The thumbnail plot to the right of the **Composite Release Capacity** table displays three curves, one for each release capacity column shown in the table. The **Controlled** capacity curve is blue, the **Uncontrolled** capacity curve is red, and the **Total** capacity curve is green. Be aware that the uncontrolled capacity in the table *does not* include the overflow capacity of the dam itself.

9.4.1 Rename the Dam

The default name of the dam is "Dam at *stream name*", where *stream name* is the name of the stream on which the dam is located. In the example shown in Figure 9.8, the name of the dam is "Dam at Hayes River". Except in the Reservoir Tree, where ever the dam is referenced, the dam name is always appended to the reservoir name; this combined name can get long so it is highly recommended that you rename the dam to something short, like just "Dam".

To rename the dam:

- Select **Rename** from the **Dam**'s context menu. A **Rename** dialog will open.
- Enter a new **Name** for the dam and click **OK**.

In case you are wondering why the default name of the dam includes the stream name, it is because you can create a reservoir with more than one dam. (Yes, you really can.) By drawing the reservoir across two or more streams that flow away from the inflow(s), a dam will be created at each outflow junction of the reservoir and each dam requires a unique name in the reservoir. See Section 5.2.3 for details on creating a reservoir with more than one dam.

9.4.2 Adding Outlets and other Elements to the Dam

Unless you *only* want to model dam overflow, you will need to add one or more *outlets* to the dam to enable water to pass through it into the downstream system.

Outlet types available in ResSim include:

Controlled Outlets—can be used to represent any outlet that has some mechanism, like a gate or valve, which can be used to adjust the release capacity of the outlet.

Uncontrolled Outlets—can be used to represent any outlet whose release capacity cannot be adjusted; i.e. it is always fully open (e.g., gates, valves).

Power Plants—a special type of controlled outlet that can be used to represent a hydropower generation facility fed by a reservoir (e.g., weirs, culverts).

Pumps—another special type of controlled outlet that can be used to represent a pump or the "reverse" half of a reversible turbine in a pumpback system.

The other *release elements* that are available in ResSim include:

Outlet Groups—used to combine one or more outlets so that they can be operated together as a single unit.

Diverted Outlets—a combination of an Outlet Group and a Diversion. Diverted outlets are schematic elements. You create them from the map display area in the same manner as you create a diversion. You cannot add a diverted outlet to the reservoir through the Reservoir Editor, but you *can* edit it, including adding and removing outlets and other elements and defining the routing properties of the diversion.

Leakage—a loss of water from the reservoir pool that passes "through" the dam into the downstream system.

In addition to release elements, the following dam elements allow you to define certain properties of the dam, outlet, or group:

Tailwater Elevation—a feature you may need add to the dam or an outlet so that it can compute head, the difference between pool elevation and tailwater, for the purpose of computing power generation or determining the ability to pump.

Forebay Head Loss—a feature you can add to the dam or an outlet so that it can adjust the current pool elevation to reflect the elevation of water at the face of the outlets for the purpose of computing release capacity.

Like the Pool, there are two ways to add outlets and other elements to the Dam:

1. from the **Dam** menu or
2. from the **Dam**'s context menu in the Reservoir Tree.

Each menu shown in Figure 9.9 provides the same list of options for adding and renaming dam elements. For simplicity, we'll focus on use of the context (right-click) menu.

Reservoir Editor—Dam Menu Reservoir Tree: Dam Context Menu

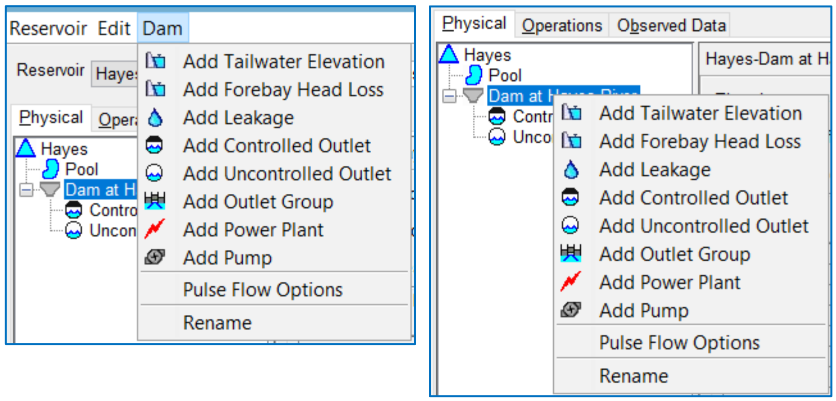


Figure 9.9 The Reservoir Editor's Dam Menus

To add an outlet or other element to the dam:

- *Right-click* on the **Dam** to which you want to add the outlet or other feature.
- Select the outlet type or property you want to add from the context menu. The menus illustrated Figure 9.9 show the various outlet types and dam elements that you can add to a dam.
- The new outlet or feature will appear in the Reservoir Tree as a branch beneath the dam or outlet group you selected.

Tip Pulse Flow Options is a Dam menu option that is not discussed in this chapter. It is an advanced option needed for effective downstream control operation and is described in Chapter 11, Section 11.3.5.

9.4.3 Renaming Elements

When you add a release element to the Reservoir Tree, ResSim will give it a default name based on the element type. For example, if you add a *Controlled Outlet,* the outlet will be given the name *Controlled Outlet*. If you add a second outlet of the same type without renaming the first one, the new outlet will get the same default name but with a number appended to it so that each outlet has a unique name, e.g. *Controlled Outlet 1*. To give your outlets and outlet groups meaningful and unique names, rename them as soon as you create them.

To **rename** an outlet or outlet group:

- In the Reservoir Tree, *right-click* on the release element and select **Rename** from the context menu (Figure 9.10).
- The **Rename Reservoir Component** dialog will open (Figure 9.11).
- Type the new name for the element in the **Name** field. You may also enter a new description in the **Description** field.

- Click **OK** to close the dialog. The new name will appear in the reservoir tree.

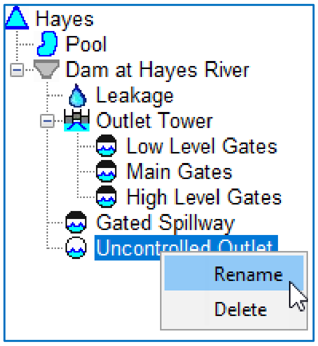


Figure 9.10 Outlet Context Menu—Rename

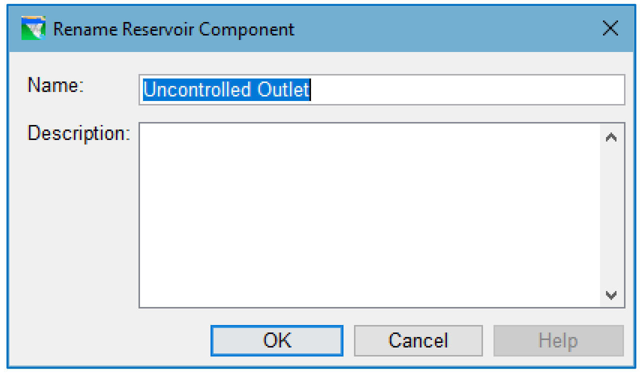


Figure 9.11 Renaming a Reservoir Component

9.4.4 Deleting or Removing Elements

As you build or revise your reservoir model, you may find it necessary to remove an element from the dam or reservoir. For example, you may find that you want to combine two or more existing outlets into an outlet group. Since cut, copy, and paste of reservoir *elements* are not features that have been implemented in the ResSim **Reservoir Editor**, be sure to create the new outlets in the outlet group before deleting the originals.

Take care when *deleting* or *removing* elements since once they're gone, they're gone. In other words, when you delete elements from the reservoir tree, ResSim immediately removes them from the reservoir and the reservoir network; no need to click the apply button. And, if you use cancel to close the Reservoir Editor, the deleted outlets will still be gone when you reopen the editor.

You can *delete* **Outlet Groups**, **Diverted Outlets**, **Controlled Outlets**, **Power Plants**, **Pumps**, and **Uncontrolled Outlets** from your reservoir. You can *remove* **Evaporation**, **Seepage**, and **Leakage** as well as **Tailwater Elevation** and **Forebay Head Loss** specifications from the reservoir and/or dam and its outlets and groups.

When you *delete* or *remove* a reservoir component, it will no longer appear in the reservoir tree. When you delete a **Diverted Outlet**, it will be removed from both the Reservoir Tree and from the network schematic shown in the Map Display Area of the **Reservoir Network** module. To replace a **Diverted Outlet**, you will need to re-draw it in the map display area.

To **delete** or **remove** an element from the dam or reservoir:

- Right-click on the release element in the Reservoir Tree.

- From the context menu, select **Delete** or **Remove** (Figure 9.12).
- A confirmation window will appear. If the chosen outlet or group has *rules* that limit its operation, the confirmation window will show a list of those rules (Figure 9.13).
    - Select **Yes** to verify that you want to remove/delete the selected element and any rules associated with it. The element will disappear from the Reservoir Tree (and any rules associated with it will be deleted as well).
    - Select **No** to cancel the Delete component process.

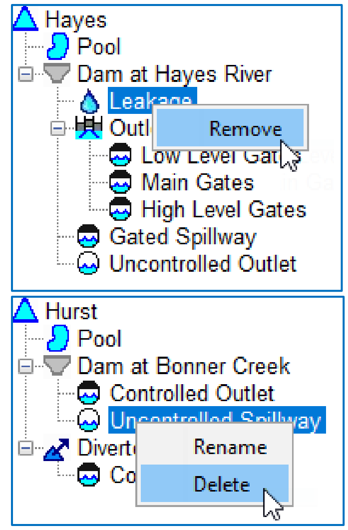


Figure 9.12 Reservoir Tree Element Context Menus—Delete & Remove

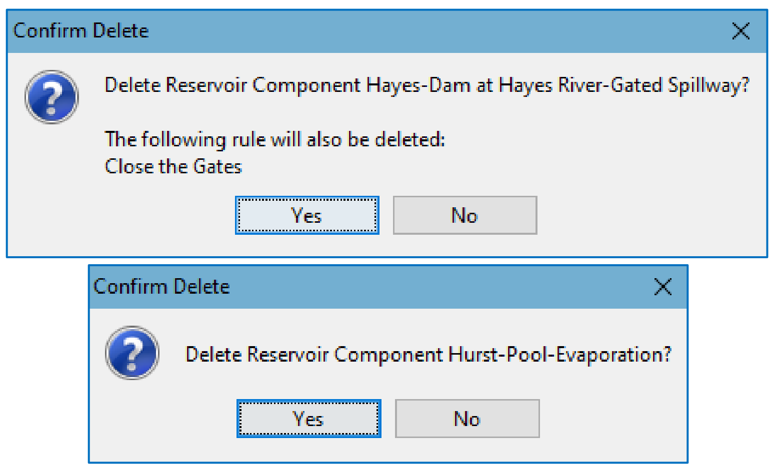


Figure 9.13 Confirm Deletion of Reservoir Component

9.4.5 Leakage

Like **Evaporation** and **Seepage**, **Leakage** is a loss of water from the reservoir pool. Leakage, although referred to as a "loss," is not a true loss of flow to the system. In fact, leakage effectively passes through the dam into the downstream system at the downstream (or tailwater) junction.

In ResSim, Leakage is considered a property of the dam and can only be added to the dam—not to an outlet or outlet group. Also like **Evaporation** and **Seepage**, you can only add one **Leakage** node to the dam, so there is no need, nor option, to rename it.

To add **Leakage** to a dam:

- *Right-click* on the **Dam** in the Reservoir Tree.
- Select **Add Leakage** from the dam's context menu.

- **Leakage** will appear as a node of the reservoir tree, immediately below the dam. It will be the active node and the **Edit Panel** will display the Leakage pane (Figure 9.14).

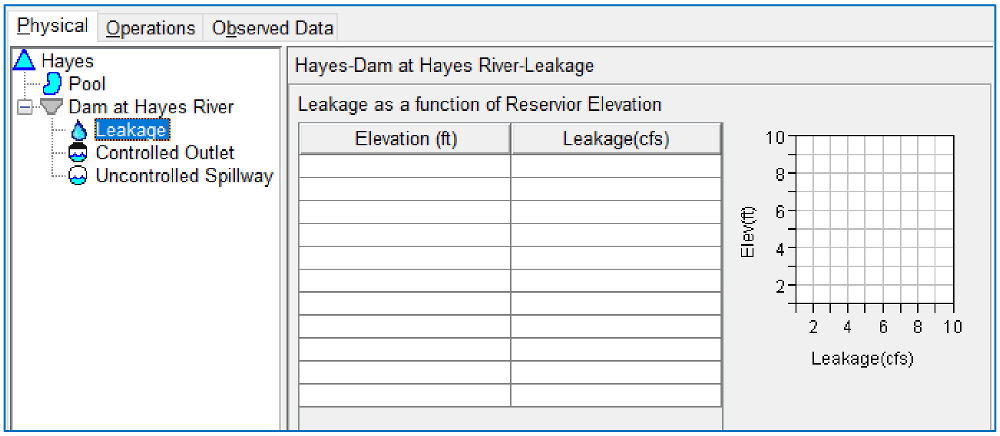


Figure 9.14 Reservoir Editor—Leakage

The Leakage pane allows you to define **Leakage** as a function of pool **Elevation**. To enter data into the Elevation vs Leakage table, either copy & paste it from a spreadsheet application or type in the data manually. Values of elevation must be entered in increasing order. The thumbnail plot to the right of the table will reflect the values entered in the table.

9.4.6 Outlet Groups

Use outlet groups so that you can treat a subset of the outlet in your reservoir as a single unit — in other words, so that you can apply rules to the group rather than to individual outlets or to the dam or reservoir. You can add an **Outlet Group** to a dam, diverted outlet, or other outlet group. An example outlet group, called Outlet Tower is shown in Figure 9.15.

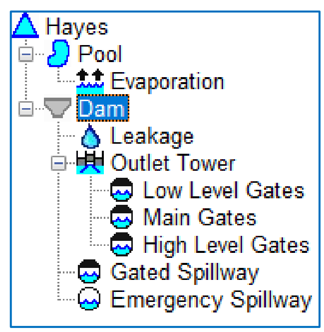


Figure 9.15 Reservoir Tree with Outlet Group

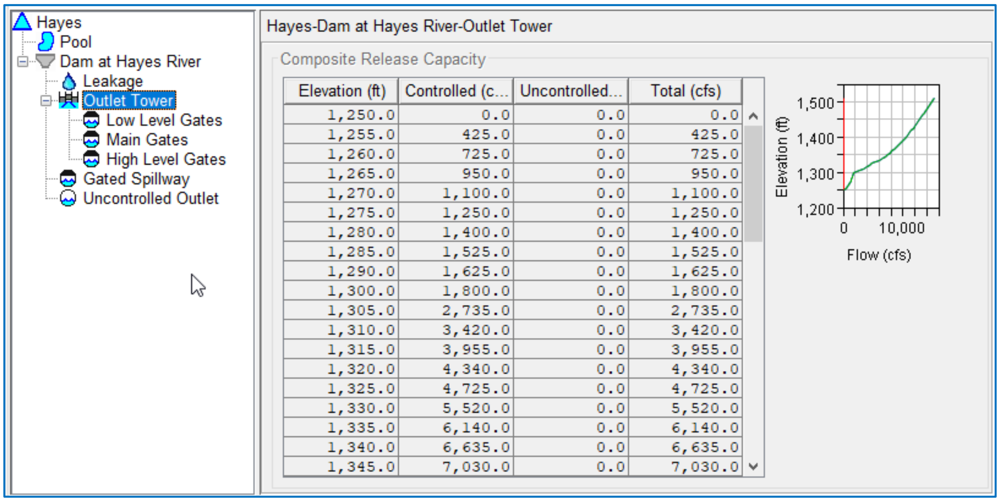
An Outlet Group can include any outlet type as well as Tailwater Elevation and Forebay Head Loss elements.

To add an **Outlet Group**:

- In the Reservoir Tree, *right-click* on the dam (or group) to which you want to add an outlet group.
- Select **Add Outlet Group** from the context menu.
- The new Outlet Group node will appear in the Reservoir Tree beneath the element you selected, and the **Edit Panel** will display the Group pane for your new outlet group.

When you select the **Group** node in the Reservoir Tree, the **Edit Panel** displays the Group. Like the Reservoir and Dam panes, the Group pane contains the name of the

Outlet Group and a non-editable **Composite Release Capacity** table (Figure 9.16) which reflects the total capacity of the outlets contained in the group.



Elevation (ft)	Controlled (c...	Uncontrolled...	Total (cfs)
1,250.0	0.0	0.0	0.0
1,255.0	425.0	0.0	425.0
1,260.0	725.0	0.0	725.0
1,265.0	950.0	0.0	950.0
1,270.0	1,100.0	0.0	1,100.0
1,275.0	1,250.0	0.0	1,250.0
1,280.0	1,400.0	0.0	1,400.0
1,285.0	1,525.0	0.0	1,525.0
1,290.0	1,625.0	0.0	1,625.0
1,300.0	1,800.0	0.0	1,800.0
1,305.0	2,735.0	0.0	2,735.0
1,310.0	3,420.0	0.0	3,420.0
1,315.0	3,955.0	0.0	3,955.0
1,320.0	4,340.0	0.0	4,340.0
1,325.0	4,725.0	0.0	4,725.0
1,330.0	5,520.0	0.0	5,520.0
1,335.0	6,140.0	0.0	6,140.0
1,340.0	6,635.0	0.0	6,635.0
1,345.0	7,030.0	0.0	7,030.0

Figure 9.16 Outlet Group Node and Group Edit Pane

9.4.7 Tailwater Elevation

Tailwater Elevation is used in the computation of head — the difference between reservoir pool elevation (above the dam) and tailwater elevation (below the dam). Specification of Tailwater Elevation is required when you model power plants and/or pumps. You can add **Tailwater Elevation** to Dams, Diverted Outlets, Outlet Groups, or individual outlets.

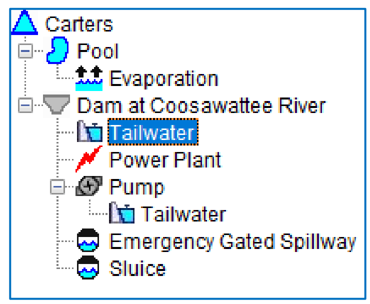


Figure 9.17 Reservoir Tree with Tailwater Nodes

When Tailwater Elevation is specified at the Dam, it applies to all outlets in the dam unless *overridden* by another Tailwater specification at a level closer to a specific outlet. For example, consider the two Tailwater nodes illustrated in Figure 9.17 — one Tailwater node was added to the Dam and another was added to the Pump. The Pump will use the Tailwater definition that was attached it, but the Power Plant will use the Tailwater Elevation specified at the Dam.

To add **Tailwater Elevation**:

- In the Reservoir Tree, *right-click* on the dam, group, or outlet to which you want to add a Tailwater Elevation specification.

- Select **Add Tailwater Elevation** from the component's context menu (Figure 9.18)
- A **Tailwater** node will appear in the reservoir tree beneath the component you selected and the **Tailwater** pane will be displayed in the edit panel.

The **Tailwater** pane (Figure 9.19) provides three options for specifying the tailwater elevation:

1. **Constant Elevation**,
2. **Downstream Control** (elevation of a downstream element), or
3. **Rating Curve**.

You can choose one, two or all three of the options and ResSim will evaluate all those selected and use the highest value among them.

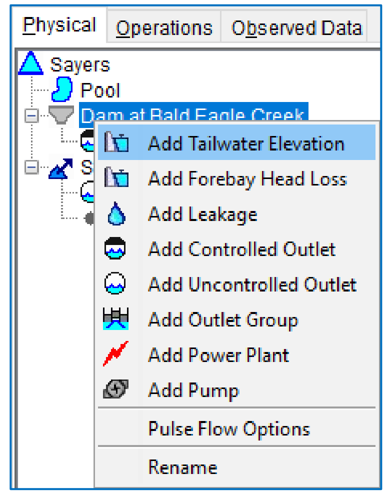


Figure 9.18 Dam Context Menu—Add Tailwater Elevation

If you select **Rating Curve**, you can specify the rating as a **Simple Rating** for which you must enter the relationship between Flow and Stage in the table, or as a **Rating Function** for which you must define tailwater Stage (or elevation) as a function of a Model Variable, External Variable, State Variable, or Two Variables.

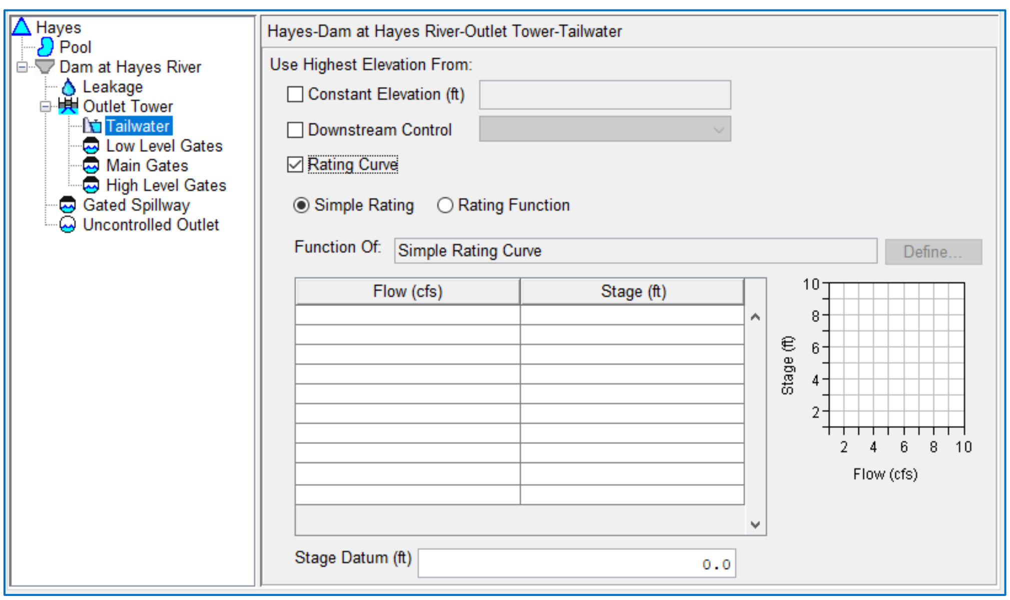


Figure 9.19 Tailwater Node and Tailwater Edit Pane

A rating curve usually relates STAGE at a gage to FLOW. And the measured quantity is usually STAGE, which would make STAGE the independent variable. But to ResSim FLOW is the known variable and it wants to lookup STAGE, so STAGE is seen as the dependent variable. So, be careful when entering the rating table to be sure FLOW is the first column of data and STAGE is the second.

In addition, each rating curve has an option for a Stage Datum to be provided. The **Stage Datum** (or gage zero) is the elevation of a STAGE of 0.0 at the gage. Each time ResSim looks up stage from a rating curve, it also computes elevation using the **Stage Datum**. If you do not provide a value for the **Stage Datum**, ResSim will use a default value of 0.0.

9.4.8 Forebay Head Loss

You can add **Forebay Head Loss** to Dams, Outlets, Diverted Outlets, and Outlet Groups.

To add **Forebay Head Loss**:

- *Right-click* on the dam, outlet, or group to which you want to add **Forebay Head Loss**.
- Choose **Add Forebay Head Loss** from the element's context menu (Figure 9.20)
- The **Forebay Head Loss** you have added will appear in the reservoir tree as a node beneath the element you selected and the Edit Panel will display the Forebay Head Loss pane (Figure 9.21).

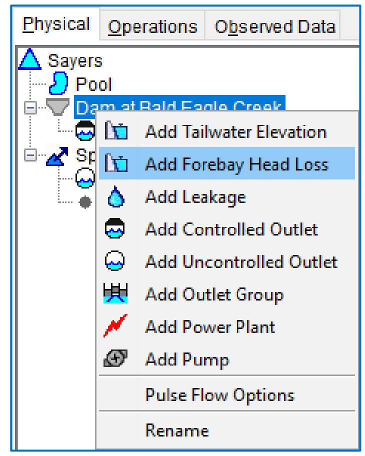


Figure 9.20 Dam Context Menu—Add Forebay Head Loss

Use the **Define** button to the right of the **Function Of:** field to specify the independent variable of the **Forebay Head Loss** function. Options include: Model Variable, External Variable, State Variable, or Two Variables.

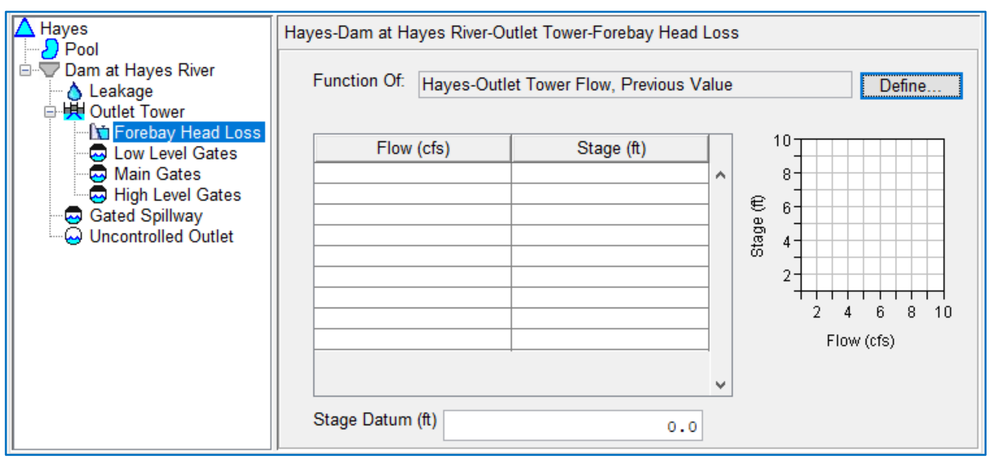


Figure 9.21 Forebay Headloss Node and Forebay Head Loss Edit Pane

9.5 Diverted Outlets

In ResSim, a **Diverted Outlet** is used to represent a withdrawal (or release) of water from the reservoir pool. In the **Reservoir Tree**, a diverted outlet is a special type of *outlet group*. Three things make a diverted outlet *special* with respect to a normal outlet group:

- A **Diverted Outlet** is attached directly to the **Reservoir** in the Reservoir Tree, parallel to the Dam and Pool.

- A **Diverted Outlet** is always created with a **Controlled Outlet** in it. If you want to replace the default outlet with some other type of outlet, add the new outlet first before deleting the default outlet.
- If the diversion (arrow) of the **Diverted Outlet** is connected at its downstream end to a Junction in the network schematic, a **Routing** node will be included in the **Diverted Outlet** group. See Section 7.2.1 for information about the available routing methods.

9.5.1 Adding a Diverted Outlet to a Reservoir

A Diverted Outlet cannot be added to a Reservoir from the **Reservoir Editor** in ResSim. Instead, a **Diverted Outlet** can be only added to the reservoir by using the **Diverted Outlet** tool to draw a reservoir diversion from the reservoir in the network schematic in the Map Display area.

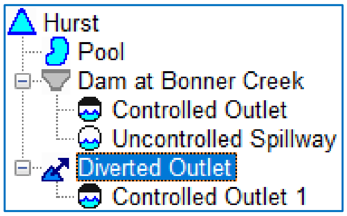


Figure 9.22 Reservoir Tree with Diverted Outlet

Once a diverted outlet has been added to the reservoir in the schematic, a new **Diverted Outlet** branch from the **Reservoir** will be added to the **Reservoir Tree** in the **Reservoir Editor** (Figure 9.22). By default, a Controlled Outlet is included in the Diverted Outlet group when the diverted outlet is added to the reservoir. If you want to replace the default outlet with a different outlet, create the new outlet before deleting the default one. To add an outlet to the diverted outlet group, select the **Diverted Outlet** in the **Reservoir Tree** then use the diverted outlet's context menu (or the **Outlet** menu) to add an outlet to the diverted outlet.

When you select the **Diverted Outlet** node in the Reservoir Tree, the **Edit Panel** displays the Diverted Outlet pane (Figure 9.23). Like the Group pane, the Diverted Outlet pane contains the name of the Diverted Outlet and a non-editable **Composite Release Capacity** table which reflects the total capacity of the outlets contained in the Diverted Outlet group.

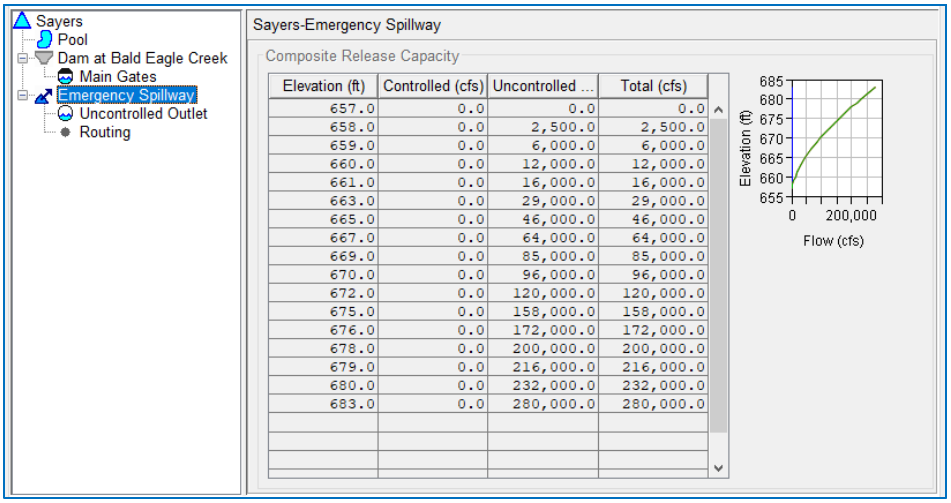


Figure 9.23 Diverted Outlet Pane—Composite Release Capacity Table

9.6 Controlled Outlets

A **Controlled Outlet** is a generic release element that may be used to represent almost any control structure used to regulate releases from a reservoir. The release capacity of a controlled outlet is specified as a relationship between pool elevation and maximum release capacity.

A variation of the maximum capacity relationship is available that allows you to define the release capacity per pool elevation for a series of gate settings or openings. Gate settings are sometimes known as "cranks" because they reflected how many times the operator must turn the crank or lever that opens or closes the gates. At present, ResSim opens or closes all the gates of a given outlet together as a single unit. So, if a given outlet represents 6 gates, ResSim will open or close all 6 gates to the same gate opening in order to produce the desired release.

In addition to Release Capacity, two physical Rate of Change constraints may be entered **Max Rate of Increase** and **Max Rate of Decrease**. However, it is recommended that you leave these fields blank and use rules in your operation sets to model all rate of change constraints — both physical and operational.

To edit data for the **Controlled Outlet**:

- Select the outlet you want to edit in the Reservoir Tree. The Edit Panel will display the Controlled Outlet edit pane (Figure 9.24).

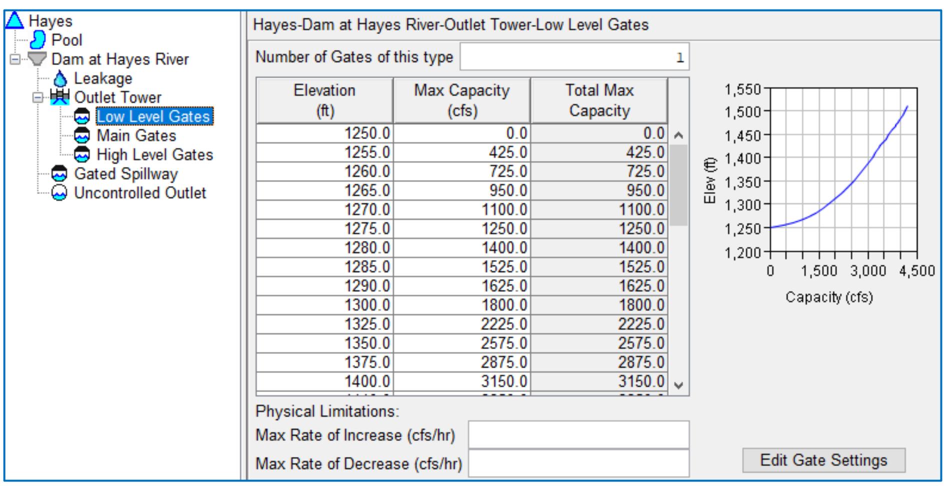


Figure 9.24 Controlled Outlet Pane—Maximum Capacity Table

- Specify the **Number of Gates** of this type.
- Specify the release Capacity of one gate by entering **Elevation** and corresponding **Max Capacity** flow values into the table. The Max Capacity table reflects the *maximum physical* flow constraint *per gate* of the outlet; **Total Max Capacity** of the outlet is the product of the Max Capacity for one gate times the number of gates.
- Optionally, specify Physical Limitations for Max Rate of Increase and Max Rate of Decrease.

- Click **Edit Gate Settings** to access the **Gate Settings** dialog (Figure 9.25). Gate settings are sometimes known as "cranks." Choose either **Specify Maximum Capacity Only** (the default) or **Specify Capacity at Specific Gate Openings**.

  To **Specify Capacity at Specific Gate Openings**:

  - Select the **Number of Gate Settings** (or "cranks") and the **Gate Setting Units** (Length or Percent).
  - Enter the **Setting** for each gate.
  - Click **OK** to close the Gate Settings dialog.

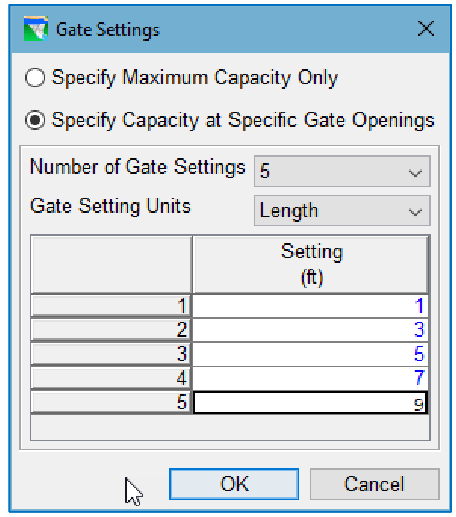


Figure 9.25 Controlled Outlet Specifying Gate Settings

Note: Even though you can enter the appropriate Gate Settings, ResSim does not currently make use of this information in most of its operations. It will, however, produce a time series of gate settings as output based on the final release from the outlet. The exceptions are the induced Surcharge and Prescribed Release rules.

Figure 9.26 shows the Reservoir Editor's Controlled Outlet data editor with Gate Settings specified.

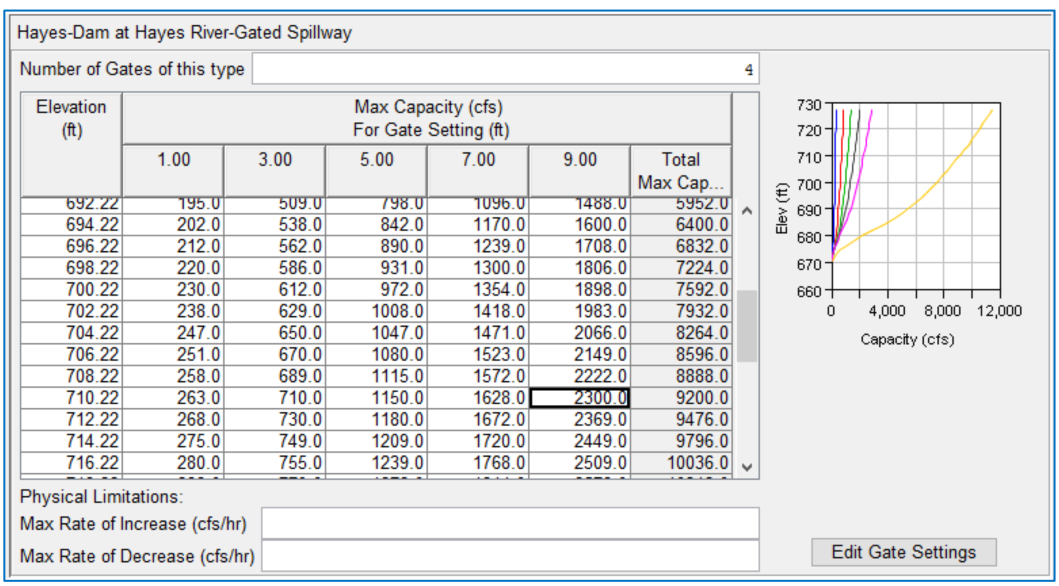


Figure 9.26 Controlled Outlet—Capacity per Gate Setting

When you are done entering Controlled Outlet data, be sure to click **Apply** before moving on to edit the next component.

9.7 Power Plants

A **Power Plant** is a type of controlled outlet that has the additional capability of computing hydropower generation. A **Power Plant** can be added to a Dam, Diverted Outlet, or Outlet Group. To edit data for a **Power Plant**, select the **Power Plant** in the Reservoir Tree. The Power Plant pane (Figure 9.27) will display in the edit panel.

The Reservoir Editor's Power Plant pane has five tabs that allow you to edit data for **Outlet** Capacity, Generating **Capacity**, **Efficiency**, **Station Use**, and **Hydraulic Losses**, as described in the following sections.

Note: Tailwater Elevation must be specified at or above a power plant in the Reservoir Tree in order for the plant to properly calculate energy production

9.7.1 Outlet Capacity

On the **Outlet** tab (Figure 9.27), you will enter the rating table for the physical capacity of this outlet. You can also specify the increasing and decreasing physical limitations (i.e., physical rate-of-change constraints).

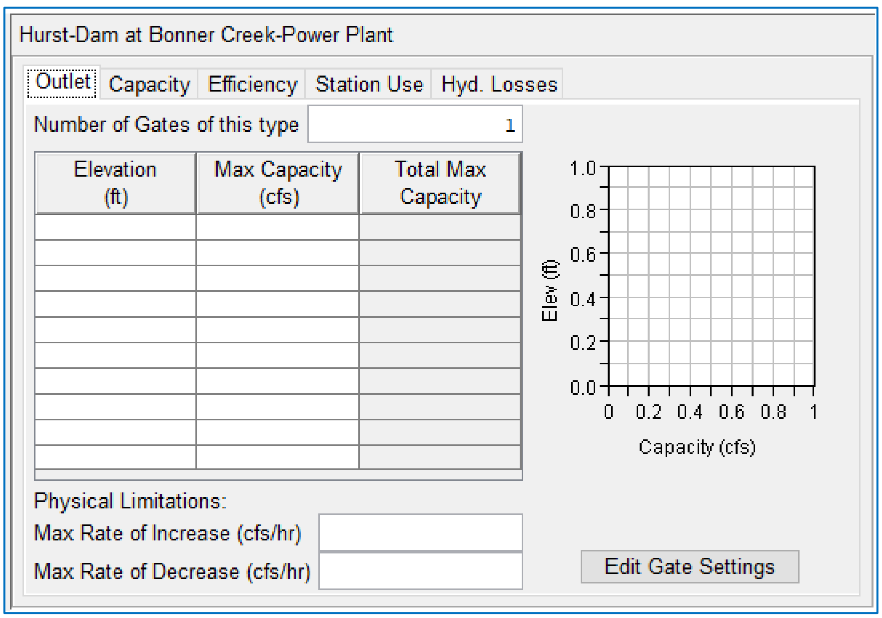


Figure 9.27 Power Plant—Outlet Tab

To enter **Outlet** data:

- Specify the **Number of Gates** of this type.
- Enter **Elevation** and corresponding **Max Capacity** flow values (per gate) into the table. You can either copy and paste data from a spreadsheet application or type in the values manually.
- Optionally, specify Physical Limitations for Max Rate of Increase and Max Rate of Decrease.

9.7.2 Generating Capacity

On the **Capacity** tab, you must specify the generating capacity of the power plant. All power plants come with a general "nameplate" or installed capacity. In addition, the capacity of the plant may vary as a function of elevation, release, or net head. Both forms of generating capacity are required input—the **Installed Capacity** and the **Variable Capacity**.

The **Installed Capacity** will be used by the Hydropower Requirement operating rules that determine required generation using Plant Factor. The **Installed Capacity** is entered as a constant value in units of megawatts (MW).

The **Variable Capacity** will be used by all the Hydropower Requirement rules to **limit** the plant's generating *capability* computed using the power equation and the other properties of the power plant. The **Variable Capacity** can be specified with one of the following five options (as illustrated in Figure 9.28):

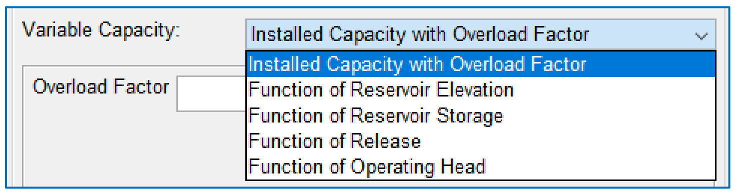


Figure 9.28 Power Plant Capacity—Variable Capacity Options

Installed Capacity with Overload Factor (Figure 9.29)—This option is the equivalent of defining the variable capacity as ***constant***. The **Overload Factor** is used as a multiplier for the Installed Capacity to determine the maximum generating rate the plant can operate at. For example, to overload a plant's installed capacity by 10%, enter a factor of 1.10. To use the full Installed Capacity with no overload, enter an Overload Factor of 1.0.

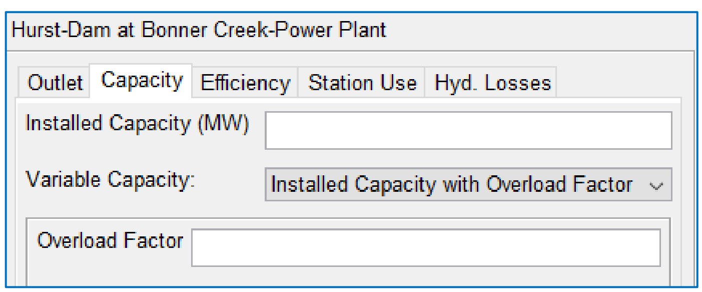


Figure 9.29 Variable Power Plant Capacity—Installed Capacity with Overload

Function of Reservoir Elevation—If the generating capacity of your power plant varies as a function of the pool elevation of the reservoir, select **Function of Reservoir Elevation** for your Variable Capacity. Then, right click on the blank row in the function table and insert the

Tip If you are copying data from a spreadsheet into a table in ResSim, you do not need to size the table to fit the data. It will resize automatically when you paste the table data into the first cell.

number of rows you need to specify the functional relationship. Enter the data specifying the relationship between elevation and generating capacity into the table in increasing order of elevation.

Function of Reservoir Storage—If the generating capacity of your power plant varies as a function of the current storage in the reservoir, select **Function of Reservoir Storage** for your Variable Capacity. Then, right click on the blank row in the function table and insert the number of rows you need to specify the functional relationship. Enter the data specifying the relationship between storage and generating capacity into the table in increasing order of storage.

Function of Release—If the generating capacity of your power plant varies as a function of the previous release from the reservoir, select **Function of Release** for your Variable Capacity. Then, right click on the blank row in the function table and insert the number of rows you need to specify the functional relationship. Enter the data specifying the relationship between release and generating capacity into the table in increasing order of release.

Function of Operating Head option (Figure 9.30)—If the generating capacity of your power plant varies as a function of the net head for the power plant, select **Function of Operating** for your Variable Capacity. Then, right click on the blank row in the function table and insert the number of rows you need to specify the functional relationship. Enter the data specifying the relationship between net head and generating capacity into the tab in increasing order of head.

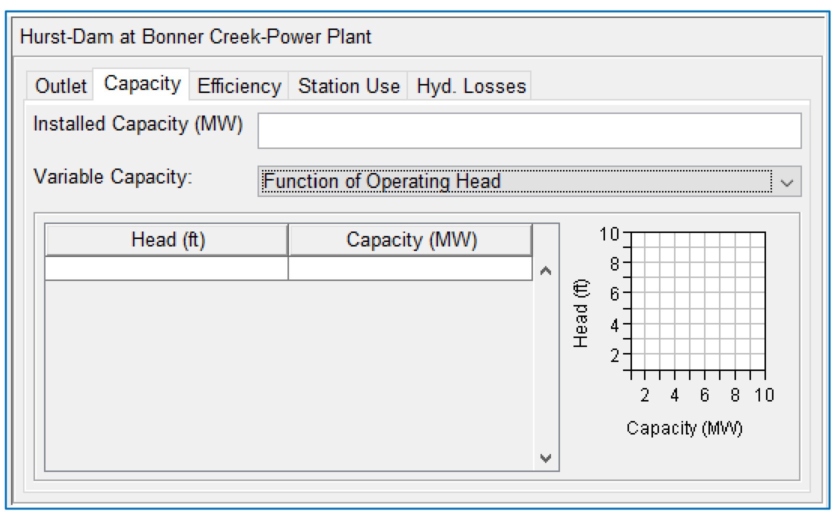


Figure 9.30 Variable Power Plant Capacity—Function of Operating Head

9.7.3 Efficiency

Efficiency represents a percentage of the total potential energy the power plant (generator efficiency & turbine efficiency) could theoretically generate. Typical plant efficiency ranges between 80% and 90%. In actual operation, however, the turbine-generator efficiency varies throughout its range of operation.

Like generating capacity, efficiency may vary as a function of a pool elevation, release or head. These are options are labeled Efficiency Methods (Figure 9.31).

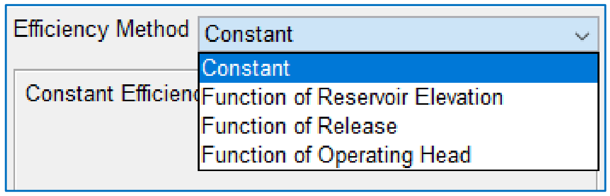


Figure 9.31 Power Plant Efficiency Methods

On the **Efficiency** tab of the Power Plant edit pane, select an **Efficiency Method.** The options include:

Constant (Figure 9.32)—If the efficiency of your power plant does not vary significantly, select **Constant** as the method for specifying the efficiency of your power plant. Enter a constant value for the overall, generator plus turbine, plant efficiency. Efficiency is specified as a percentage.

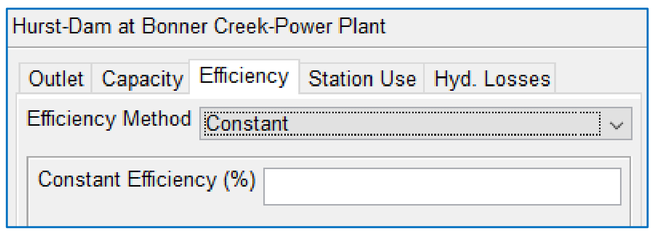


Figure 9.32 Variable Power Plant Efficiency—Constant

Function of a Reservoir Elevation—If the efficiency of your power plant varies as a function of the elevation of the reservoir pool, select **Function of Reservoir Elevation** as the method for specifying the efficiency of your power plant. Next, right click on the blank row in the function table and insert the number of rows you need then enter the data specifying the relationship between elevation and generating capacity into the table in increasing order of elevation. The thumbnail plot will reflect the data you enter in the table.

Function of Release—If the efficiency of your power plant varies as a function of the previous release from the reservoir, select **Function of Release** as the method for specifying the efficiency of your power plant. Enter the data specifying the relationship between release and generating capacity into the table in increasing order of release.

Function of Operating Head (Figure 9.33)—If the efficiency of your power plant varies as a function of the net heat at the reservoir, select **Function of Operating** as the method for specifying the efficiency of your power plant. Enter the data specifying the relationship between head and generating capacity into the table in increasing order of head.

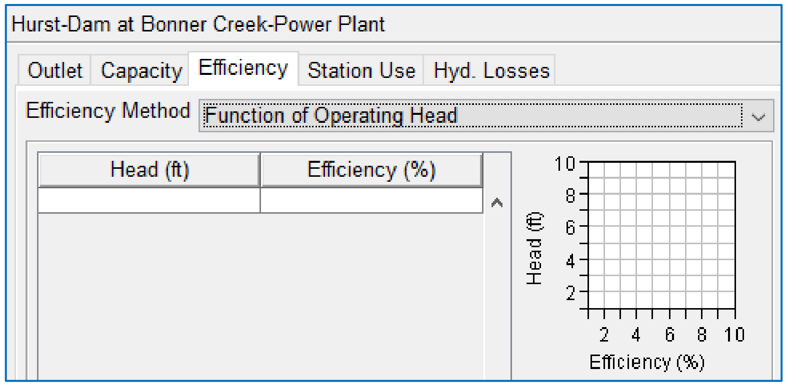


Figure 9.33 Variable Power Plant Efficiency—Function of Operating Head

9.7.4 Station Use

Station Use is the quantity of flow used to run a "station" unit whose power is used to operate the powerhouse. It is a portion of the *flow* that passes through the powerhouse, but it is not used to generate power for the "grid" and is therefore considered a loss.

On the **Station Use** tab of the Power Plant edit pane, you can define the **Station Use** as constant flow rate or as a function of flow entering the powerhouse (release).

Constant (Figure 9.34)—enter the station use as a constant flow rate.

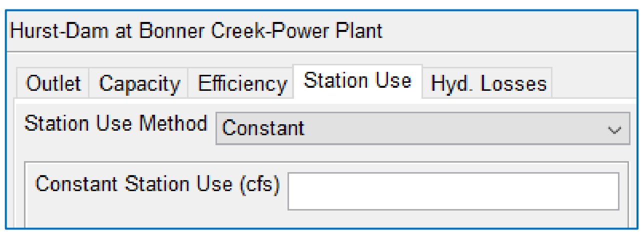


Figure 9.34 Power Plant—Station Use—Constant

Function of Release (Figure 9.35)—enter a table of values to specify the relationship between **Release** and **Station Use**. The thumbnail plot will reflect the data you enter in the table.

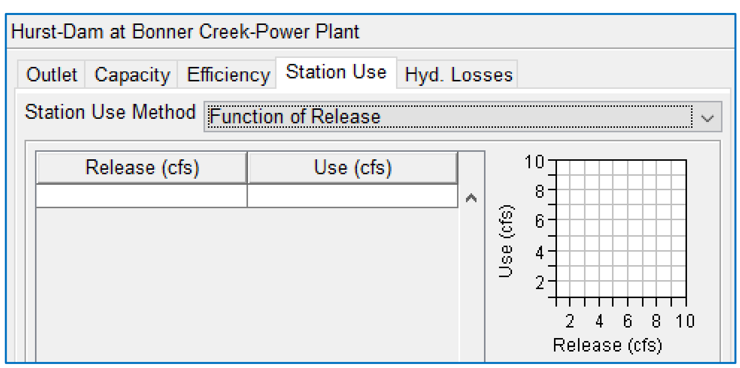


Figure 9.35 Power Plant—Station Use—Function of Operating Head

9.7.5 Hydraulic Losses

Hydraulic loss can also be referred to as head loss and is used to represent the friction losses in the penstock. ResSim uses net head (the difference between the pool elevation and the tailwater elevation) in the power equation to calculate generating capacity. When you specify a hydraulic loss, you are describing a loss to the head, which results in a reduction in the plant's generating capacity.

You can specify the head loss as a constant or as a function of flow through the penstock (entering the powerhouse):

Constant Hydraulic Losses (Figure 9.36)—enter the hydraulic loss as a constant value of distance (ft or m).

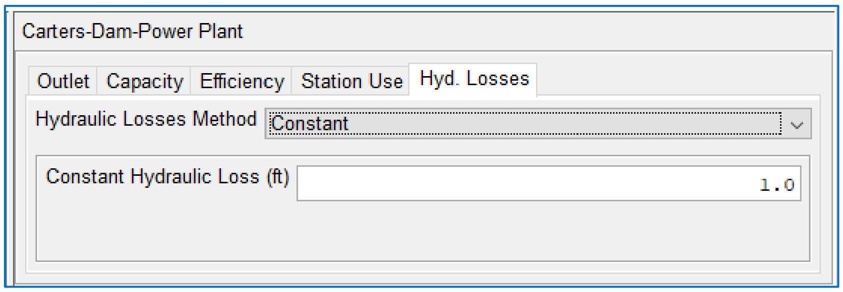


Figure 9.36 Power Plant—Hydraulic Losses—Constant

Function of Release (Figure 9.37)—enter a table of values to specify the relationship between **Release** and **Loss** to describe the head loss as a function of flow through the outlet. The thumbnail plot will reflect the data you enter in the table.

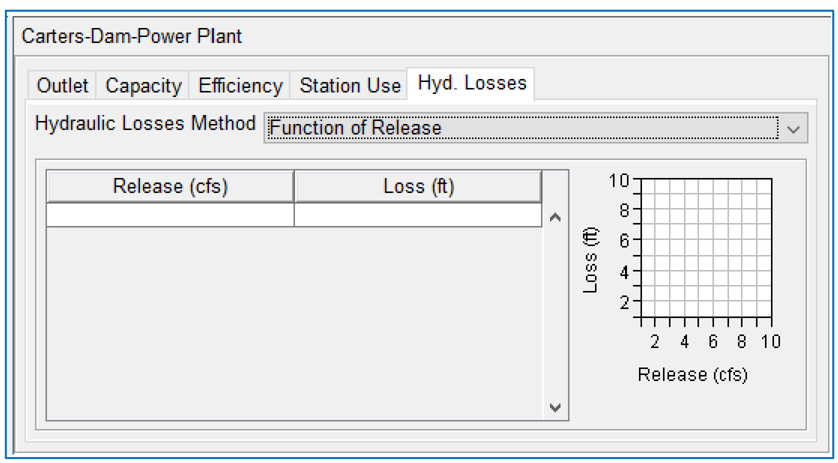


Figure 9.37 Power Plant—Hydraulic Losses—Function of Release

9.8 Pumps

A **Pump** (or pump station) is effectively a *negative* outlet—instead of taking water from the pool and discharging it to the downstream system, it takes water from the downstream system—specifically from another reservoir—and discharges it into the pool. Although the downstream reservoir may not be immediately downstream, a pump outlet creates a virtual pipe between the reservoir that contains the pump and the downstream reservoir. You cannot use a pump to move water from a river into the reservoir. Figure 9.38 shows a pump outlet in the Reservoir Editor Physical tab.

Note: Tailwater Elevation must be specified at or above a pump in the Reservoir Tree in order for the pump to determine if it is allowed to operate.

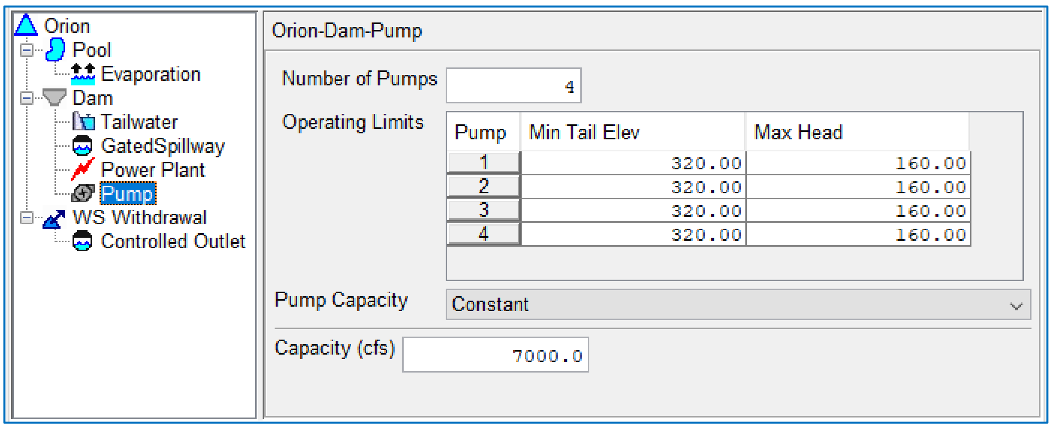


Figure 9.38 Pump Outlet

A **Pump** can be added to the Dam, a Diverted Outlet, or an Outlet Group.

The **Pump** edit pane (Figure 9.39) has three basic input parameters for defining a pump:

Number of Pumps—Think of a pump outlet as a "Pump Station" in which one or more pumps operate together to meet the pumping objective. Enter the number of identical pumps in the pump station. This value will add rows the Operating Limits table, one for each pump, and will be used as a multiplier on **Pump Capacity** to determine the total rate at which the pumps can move water.

Operating Limits—this table requires that you specify the operating range for each pump in the pump station. The operating range is defined by a minimum tailwater elevation and a maximum head. The tailwater and head information both rely on a valid definition of the tailwater, so don't forget to add a tailwater element to the pump or to its parent element.

Pump Capacity—Two options are available for specifying the **Pump Capacity** of each pump in your pump station:

Constant—enter a flow value that represents the rate at which the pump can water (Figure 9.39).

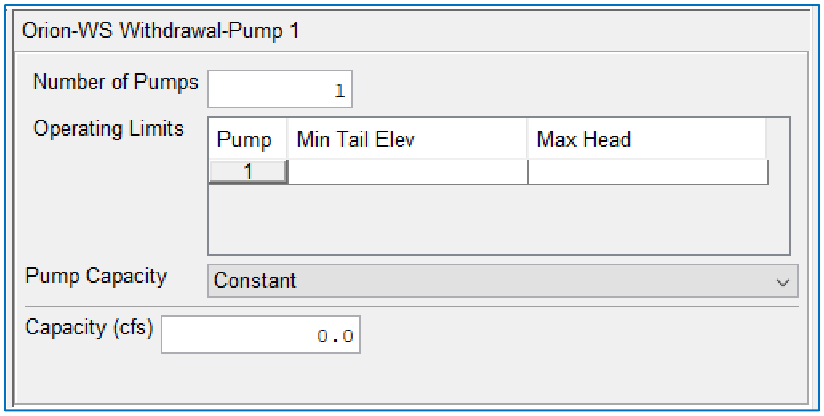


Figure 9.39 Pump Edit Pane—Pump Capacity—Constant

Function of Operating Head—enter the relationship between operating head and pump capacity in the table (Figure 9.40). The values of operating head must in entered in increasing order

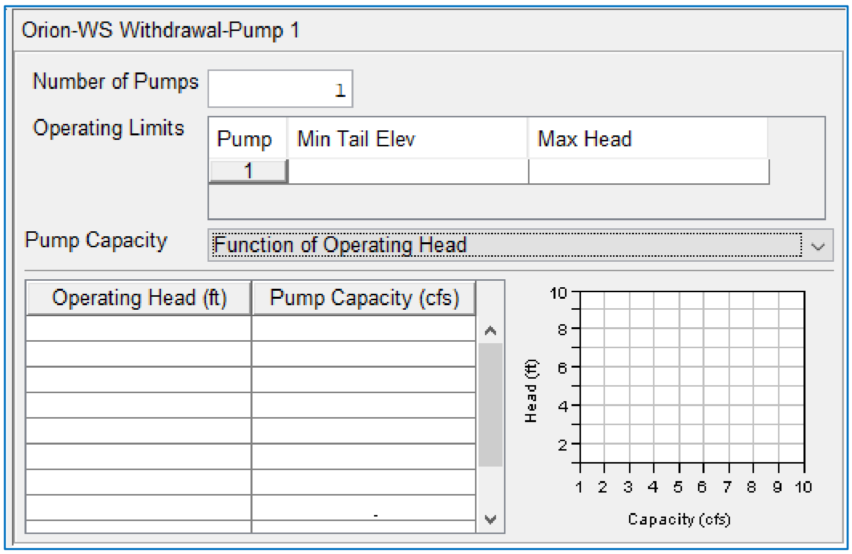


Figure 9.40 Pump Edit Pane—Pump Capacity—Function of Operating Head

9.9 Uncontrolled Outlets

An **Uncontrolled Outlet** can be a component of a Dam, Diverted Outlet, or Outlet Group. An uncontrolled outlet can represent an ungated overflow spillway or any opening in the dam through which water can pass without impediment by a movable gate or panel.

Two options for specifying the capacity of an uncontrolled outlet are provided — a weir equation or an **Elevation** vs **Outflow** capacity table. If the outlet is a standard overflow weir, you can use the weir equation to compute the flow over the spillway using a **Weir Coef**ficient and a spillway **Length** that you must provide. You can use the capacity table for all types of uncontrolled outlets whose capacity can be defined by a simple rating curve (elevation vs outflow relationship.

Weir Coefficient—typically in the range 2.6-4.0 depending on the shape of the spillway crest. This value determines flow over the spillway in the weir equation.

Length—spillway length.

Elevation vs. Outflow Table—enter data into the table in increasing order of elevation. You can either copy-and-paste the data from a spreadsheet application or type it in manually.

Figure 9.41 shows the Reservoir Editor's Uncontrolled Outlet edit pane with the capacity table selected and data entered. Note that the corresponding thumbnail plot reflects the entered data.

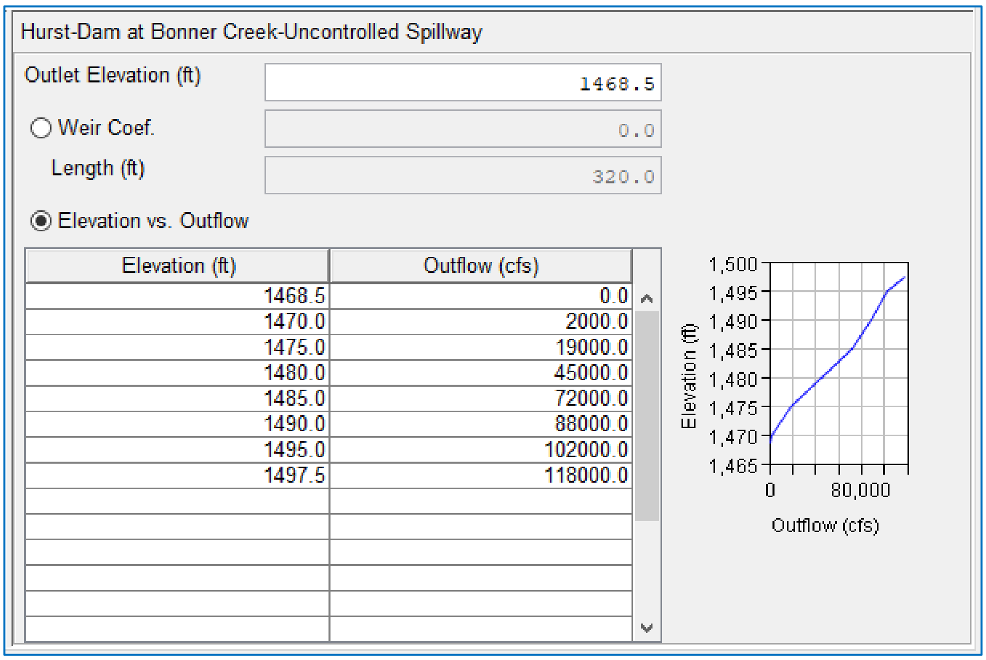


Figure 9.41 Reservoir Editor—Physical Data—Uncontrolled Outlet

Whether you use the weir equation or the Elevation vs Outflow table, you must specify an **Outlet Elevation.** The Outlet Elevation serves as a trigger to determine when this outlet starts "flowing."

Outlet Elevation—The minimum elevation at which the outlet will begin to release water. This elevation is usually the "sill" or bottom of the outlet opening.

.

Chapter 10
Reservoir Operations — The Basics

Reservoir operation is the act or process of storing water in and releasing water from a reservoir. The operations for most reservoirs regulated by the U.S. Army Corps of Engineers are based on a concept called *Guide Curve Operation*.

Guide Curve Operation is the process of determining and making releases from a reservoir in order to get to and maintain the reservoir pool at its target elevation. Thus, the *guide curve* is the target or desired pool elevation for the reservoir.

Reservoir operating zones are another concept related to the guide curve. An operating zone is a horizontal slice of the reservoir pool for which the goals and constraints differ from those in another zone of the reservoir.

The operating plan for most reservoirs is described in a reservoir regulation or water control manual. The water control manual for most Corps of Engineers reservoirs describes a *seasonally-varying* target pool elevation commonly called the **Guide Curve**. The (available) storage of the reservoir *above* this target elevation is referred to as the **Flood Control** pool and the storage *below* the guide curve is called the **Conservation** pool. The guidelines for determining the release from the reservoir are then based on where the current pool elevation is in relation to the guide curve. With guide curve operation as the fundamental objective of the reservoir operation, all other goals and objectives described in the regulation plan or water control manual can be interpreted as limits on guide curve operation.

Since ResSim was designed for modeling Corps reservoirs, the foundation of the ResSim decision logic is basic *Guide Curve Operation*. This means that, to ResSim, the *primary* reservoir operating objective is to maintain the pool at, or return the pool to, the guide curve elevation as soon as possible. So, if the pool is below the guide curve, guide curve operation would reduce or suspend releases in order to refill the pool; if the pool is above the guide curve, then releases would be increased up to maximum capacity in order to draw the pool back down to the guide curve elevation.

Operational rules that reflect the goals and constraints on the reservoir operation act as limits on the guide curve operation. A well-designed set of rules will temper the potentially volatile release behavior of the guide curve operation and produce smooth transitions across reservoir operating zones without abrupt or oscillating changes in the releases.

In a manner similar to the methods a regulator may use, each reservoir in your ResSim network must determine how much water to release at each timestep of a simulation run. In ResSim, reservoir operation is the timestep by timestep simulation of the release decision-making process over a time window.

This chapter will describe the **Operations Tab** of the **Reservoir Editor** and guide you through the process of creating and managing your reservoirs' operating plans (**Operation Sets**), defining the operating **zones** in each plan, and specifying the allocation of the release to your outlets. Creating and managing the operating ***rules*** that you will add to the zones in your operation sets will be covered in Chapter 11. Chapter 12 covers creating and managing **IF-Blocks**, **State Variables**, and **Scripted Rules** (Sections 12.1, 12.2, and 12.3, respectively), as well as some advanced operation features that can be applied to the operation set (Sections 12.5-12.8).

10.1 The Reservoir Editor's Operations Tab

The **Operations** tab of the **Reservoir Editor** (Figure 10.1) allows you to define reservoir operation sets for the reservoirs in your network. As part of the definition of an operation set, you will specify the zones and rules that describe the operating plan for the reservoir.

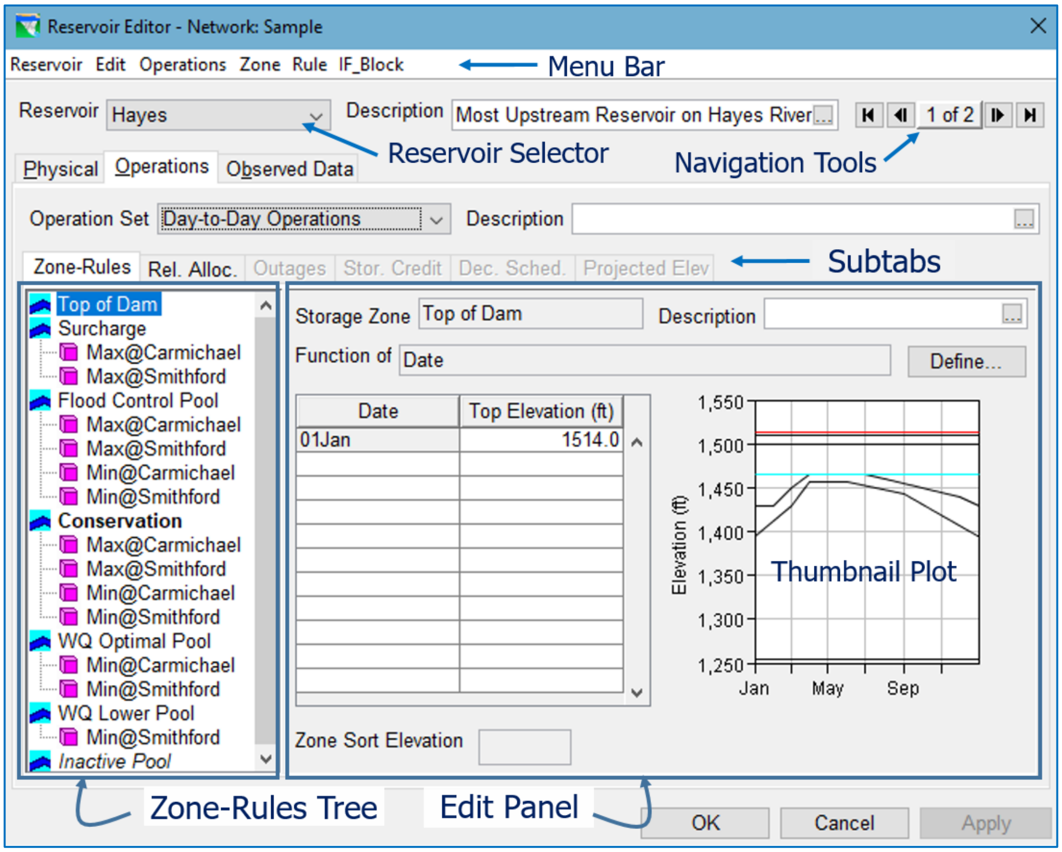


Figure 10.1 Reservoir Editor Operations Tab—Annotated

When the **Operations** tab is selected, four menus unique to the **Operations** tab will be added to the **Menu Bar** at the top of the Reservoir Editor:

- Operations
- Zone
- Rule
- IF_Block.

These menus provide options for creating and deleting operation sets, zones, rules, and IF_Blocks respectively.

At the top of the Reservoir Editor, the **Reservoir** selector contains a list of all of the reservoirs in your network, with the name of the current reservoir displayed. You can access all of the reservoirs in your network either from this list or by using the arrows to navigate through the available reservoirs. Beside the Reservoir name list is the **Description** of the current reservoir; this field is editable. Use the description field to keep notes on decisions you made while developing the data for the reservoir, your plans, intentions, references, etc. You can enter a longer description by using the button to access the full text editor for the description.

At the top of the **Operations** tab, the **Operation Set** selector contains a list of all the operation sets you have defined for the current reservoir. Use the **Operation Set** selector to select the operation set you wish to edit. The rest of the **Operations** tab will fill with the data for the selected operation set.

Beside the **Operation Set** selector is the **Description** text box. Enter information in the **Description** field to describe the significant features of the operation set and how this operation set differs from others you may have created for this reservoir. You can enter and view longer descriptions by using the button to access a larger text editor for the description.

NOTE: In the Reservoir Network module of ResSim, the Operation Set selector defaults to the first operation set in the list when the editor is first opened or when you change to a new reservoir, so be sure the operation set you wish to edit is displayed in this selector before making changes.

Below the **Operations Set** selector and the **Description**, the **Operations** tab contains a set of sub-tabs that provide the editors for specifying various features of your operation set. The first two tabs are active by default because they contain the features that must be defined for every operation set. The remaining tabs are inactive by default because they are for optional features of your operation set. The options to active the inactive tabs are available in the **Operations** menu of the **Reservoir Editor**.

The first sub-tab is the **Zone-Rules** tab. As the name implies, this is where you will create and edit the zones and rules for your operation set. This tab has two parts, the **Zone-Rules Tree** and its associated **Edit Panel.**

The **Zone-Rules Tree** displays the zones you have defined for the current operation set. With each new operation set, ResSim creates a default set of zones that includes **Flood Control**, **Conservation**, and **Inactive**. Beneath each zone in the tree is a prioritized list of the rules that apply to that zone. As you add zones and rules to the reservoir, the tree will expand to show them.

Like the **Physical** tab, the **Edit Panel** of the **Zone-Rules** sub-tab of the **Operations** tab, changes depending upon the element you select in the **Zone-Rules Tree**. If you select a zone, the zone editor will appear in the Edit Panel. If you select a rule, the specialized rule editor for that rule type will be displayed. A **thumbnail plot** on the right side of the Edit Panel reflects the data you enter in the table of the current editor. The thumbnail plot can be viewed in full size when you double-click on it.

The **Rel. Alloc**. sub-tab contains the editor for specifying the *Release Allocation* scheme for the current operation set. The *release allocation* scheme describes the method the reservoir will use to distribute the release from the reservoir across the available outlets.

10.2 Reservoir Operation Sets

To guide the decision-making process in ResSim, a paradigm has been developed for describing the goals, objectives, and constraints that define how a reservoir should operate; this paradigm is called an operation set.

An **Operation Set** is the operation plan or scheme upon which a reservoir bases its decisions regarding how much water to release at each timestep of a simulation run. You can define multiple operation sets for each reservoir, but each alternative can follow only one operation set per reservoir.

An operation set consists of the definition of at least four basic features:

Zones are operational subdivisions of the Reservoir Pool. Each zone is defined by a curve describing the top of the zone. When you create an operation set, ResSim establishes a default set of zones within the operation set. These zones are **Flood Control**, **Conservation**, and **Inactive**. The ***Inactive*** zone is a special zone in the operation set. It represents the "dead" storage of the reservoir. The reservoir cannot release water from the Inactive pool, and rules cannot be added to this zone.

Rules represent the goals and constraints upon the release(s). Rules can be applied to selected zones of the reservoir to describe the different factors influencing the release decision when the reservoir elevation is within each zone.

The **Guide Curve** is identified by selecting the top of one of your operational zones to represent the target elevation of the reservoir. By default, ResSim assigns the Guide Curve to the top of the **Conservation** zone.

> The guide curve concept is used as the basis for the release decision process in ResSim. *Basic Guide Curve Operation* means "get the reservoir pool elevation to the current guide curve elevation as fast as possible, within the physical and operational constraints of the outlets".

> ***Tip*** An operation set that has the zones defined but no rules will cause ResSim to follow the *Basic Guide Curve Operation*. For testing purposes, every reservoir should have an operation set of this type. It is the easiest way to verify that your physical data, your operational zones, and your guide curve have been properly defined.

Release Allocation is the specification of how the release from the reservoir is divvied up across the reservoir's outlets.

A variety of advanced features can be added to the operation set to further describe the regulation plan for the reservoir. These features are defined using the various sub-tabs of the Operations tab. See Chapter 12 for information on modeling Outages (Section

12.5), Storage Credit (Stor. Credit sub-tab, Section 12.6), Decision Schedule (Dec. Sched. sub-tab, Section 12.7), and Projected Elevation (Projected Elev sub-tab, Section 12.8).

This section describes how to create and edit an operation set. The next section (Section 10.3) describes how to configure the zones within your operation set.

10.2.1 Creating a New Operation Set

To create a new Operation Set:

- Select **New** from the **Operations** menu of the Reservoir Editor. The **New Operation Set** dialog will open (Figure 10.2).
- Give the new operation set a **Name** and a **Description**.
- Click **OK** to complete the process and to close the dialog.

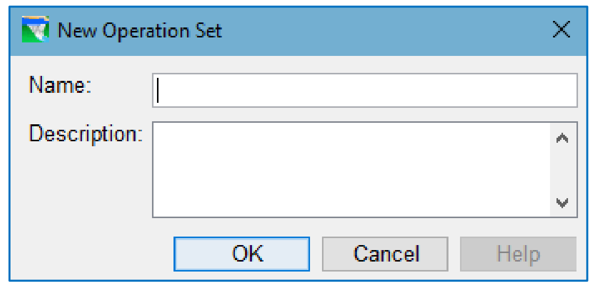


Figure 10.2 New Operation Set

The name you entered will now appear in the **Operation Set** list, and the description will appear in the **Description** field. The Description field is editable. Use the description to describe the purpose of the operation set, the expected behavior the operation set should provide, and any changes you had to make to accomplish your goal.

ResSim establishes a default set of zones within the new operation set. These zones are **Flood Control**, **Conservation**, and **Inactive**. These zones can be renamed or deleted. However, the **Inactive** zone is a special zone in the operation set. It represents the "dead" storage of the reservoir. The reservoir cannot release water from the Inactive pool and rules cannot be added to this zone.

The remainder of this chapter will detail all the options available for defining and editing your operation set.

10.2.2 Renaming an Operation Set

To rename an operation set:

- Select **Rename** from the **Operations** menu of the Reservoir Editor. The **Rename Operation Set** dialog will open (Figure 10.3).
- Give the operation set a new **Name**.

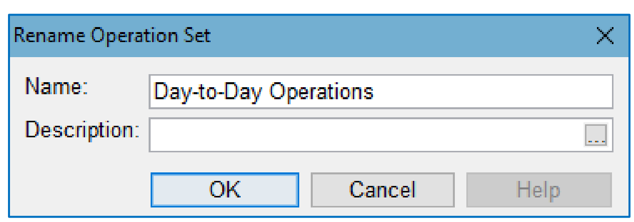


Figure 10.3 Rename Operation Set

- Click **OK** to complete the rename process and to close the dialog. A confirmation dialog will appear asking if you really want to rename the selected operation set.

10.2.3 Copying an Operation Set

To copy (or duplicate) an operation set:

- Select the Operation set you want to copy from the **Operation Set** selector on the **Operations** tab.
- Select **Duplicate** from the **Operations** menu of the Reservoir Editor. The **Duplicate Operation Set** dialog will open (Figure 10.4).

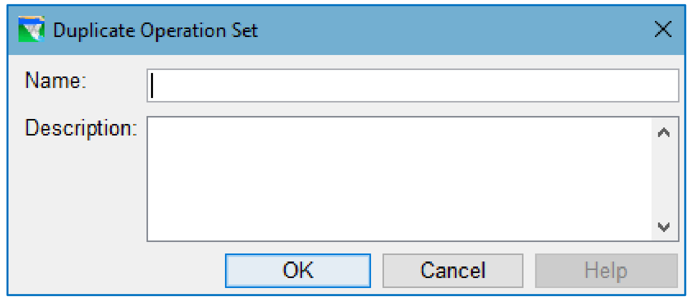


Figure 10.4 Duplicate Operation Set

- Give the operation set a new **Name** and a **Description**.
- Click **OK** to complete the copy process and to close the dialog.

The new operation set will have a copy of all the zones that were in the original operation set. Each zone in the new operation set will also list the same rules that were used by the original operation set.

10.2.4 Deleting an Operation Set

To delete an operation set:

- Select Delete from the Operations menu of the Reservoir Editor. The **Select Operation Set to Delete** dialog will open (Figure 10.5).
- Highlight the operation set you wish to delete. Its name should appear in the grey box at the bottom.
- Click **OK** to complete the delete process and to close the dialog. A confirmation dialog will appear asking if you really want to delete the selected operation set.

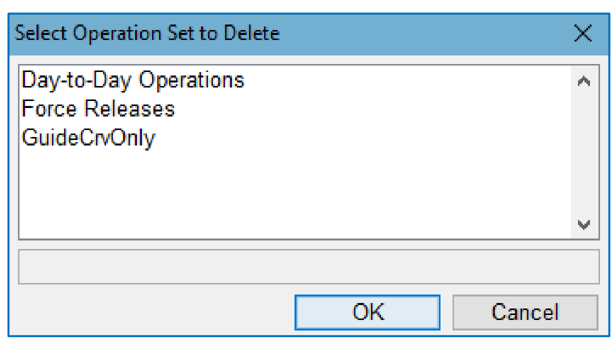


Figure 10.5 Select Operation Set to Delete

10.3 Reservoir Operation Zones

As illustrated in Figure 10.6, when you create a new **Operation Set**, ResSim automatically creates a **Zone-Rules** tree that contains three default reservoir operation zones: **Flood Control**, **Conservation**, and ***Inactive***. You may wish to rename these default zones and add descriptions. You may need to define additional zones as well. This section will detail how to edit the zone definitions of your operation set.

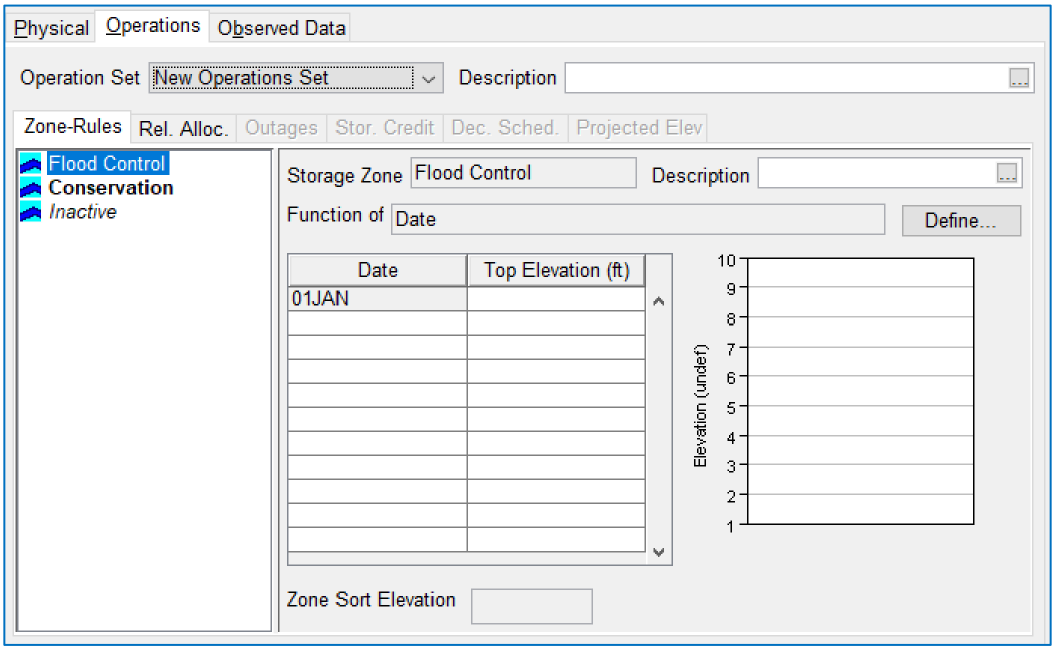


Figure 10.6 Reservoir Editor Showing New Operation Set

10.3.1 Adding a New Reservoir Storage Zone

To add a new Reservoir Storage Zone:

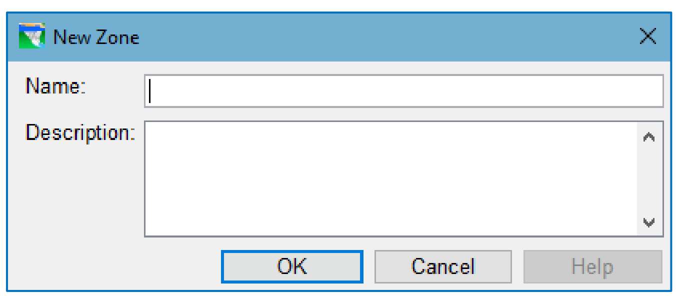


Figure 10.7 New Zone

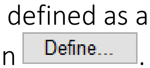
- Select New from the Zone menu of the Reservoir Editor. The New Zone dialog will open (Figure 10.7).
- Enter a **Name** and **Description** for the new zone.
- Click **OK** to complete the zone creation and close the **New Zone** dialog.

The new zone you created now appears in the Zone-Rules Tree. Complete the process of creating the zone by defining the curve that represents the *top of the zone*, as explained in the next section.

10.3.2 Defining Operation Zones

An operation zone of a reservoir is the storage between two elevations or levels of the reservoir. To ResSim, a zone is defined by a curve representing the top of the zone and is usually named for that curve or upper level; the bottom of the zone (or lower level) is the top of the next lower zone in the operation set. The bottom of the bottom-most zone is the bottom of the reservoir pool.

You will need to define the curve representing the *top* of each zone you create. The default definition of a zone is a linear function of date and is specified with a table of dates and elevations. The first date in the table is always 01Jan and the table is expected to represent the shape of the level over a single year. When plotted, the *top-of-zone curve* is drawn with a series of straight lines connecting the points defined in the table.

Although **function-of-Date** covers the majority of zone definitions, a *top-of-zone curve* can be defined as a function of something other than **Date** by selecting the **Define** button . Your options include:

- **Date *and* Time**,
- **Model Variable**,
- **External Variable**,
- **State Variable**, or
- **Two Variables**.

For example, if the elevation of the top of the zone varies from year to year, then it can be defined as a function of an external time series (**External Variable**) by reading the top of zone from HEC-DSS. Or, top of zone could be computed by a **State Variable** script within your model allowing it to vary as a function of multiple variables and conditions over time.

When you select a zone in the **Zone-Rules Tree**, the **Zone Editor** is displayed in the Edit Panel and the thumbnail plot at the right of the editor will display the curves representing the top of each zone (Figure 10.8). The color of the current zone's curve will be red. All other zones will be black except the zone that has been designated as the guide curve which will be cyan. The figure below shows the top-of-zone curve definition for a zone named *WS Curve B.*

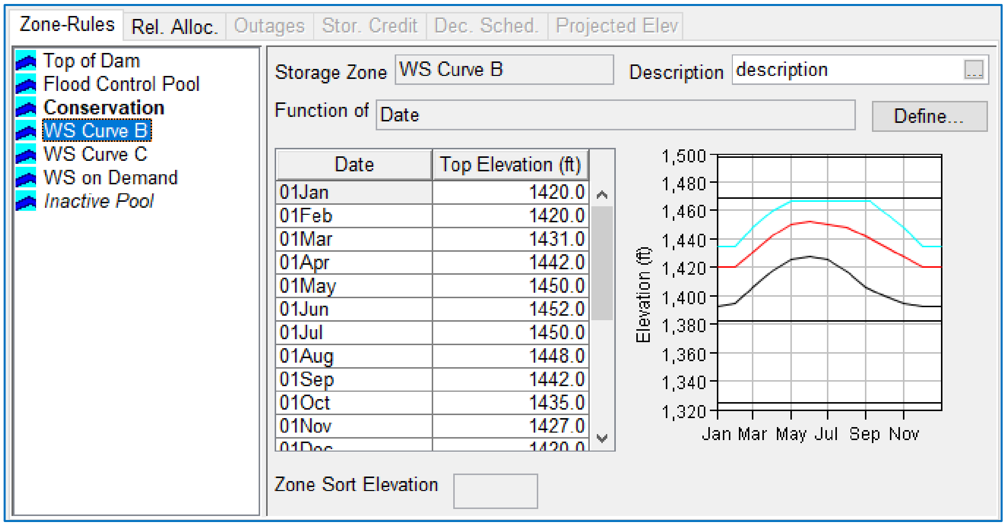
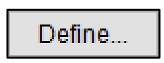


Figure 10.8 Reservoir Editor—Operations Tab—Zone Editor

To define a top-of-zone curve:

- Select the zone in the **Zone-Rules Tree** to access the Zone Editor and the data for that zone.
- Use the **Function of:** 

 button to open the **Independent Variable Definition** editor (Figure 10.9):

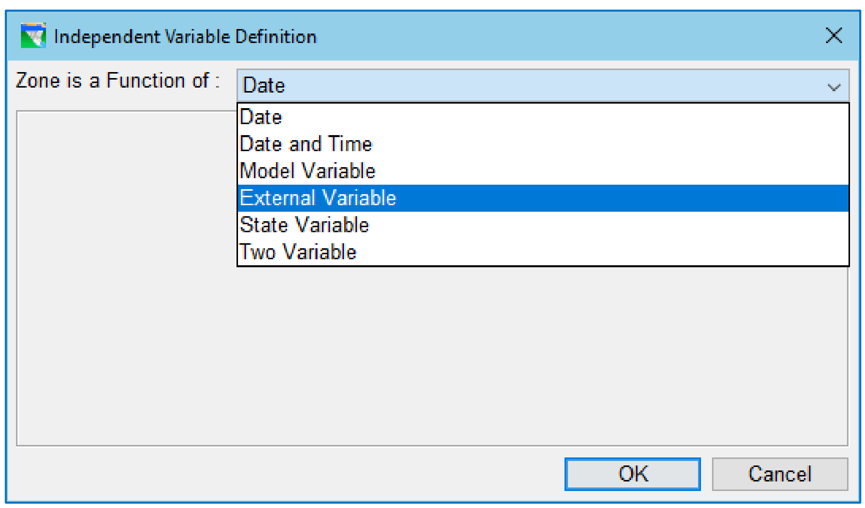


Figure 10.9 Independent Variable Definition
"Zone is a Function of:" Selector

- Select the type of function (and its variables) you want to use to define the top-of-zone curve. Your options include:

Date: By default, the function type for a zone defintion is set to **Date**. When a zone is defined as a **Function of**: **Date**, the data in the function table is expected to represent the top-of-zone elevation for a single year, which will be repeated as needed for a simulation.

If *function-of-date* is appropriate for your zone, specify the top-of-zone curve by entering the **Date** and **Elevation** values for each inflection point of your curve into the function table. You can either copy and paste data from a spreadsheet application or type in the values manually. The data you enter in the table will be displayed as the red curve in the thumbnail plot to the right of the table.

Be careful when copying "Date" data from a spreadsheet. Most spreadsheet applications use their own specialized numeric format for dates. Only 'text' and dates in DDMMM format can be pasted into a date cell in ResSim.

Dates are entered in a DDMMM format — signifying a two-digit day (yes, use a preceding zero for days 1-9) and a 3-character month. Dates can also be entered using the calendar tool, which can be accessed by double-clicking in a date cell and then clicking the small ellipsis button that appears on the right side of the cell.

NOTE: Function-of-date tables in ResSim ALWAYS start on 01Jan, which will appear in the first cell of the table. This first cell is grey to indicate that it cannot be changed.

The *function-of-date* curve for a zone is always a linear function, which means that the value of a point between two specified points in the table is computed using linear interpolation and the curve is plotted with a series of straight lines connecting the points defined in the function table.

Since the Independent Variable Definition editor is used by a variety of features throughout ResSim, to minimize repetitiveness, the description for using it has been moved to Appendix C. However, there are two unique options specific to zones that should be mentioned, and they are included below.

External Variable (Figure 10.10): Although **External Variable** is a standard option in the Independent Variable Definition editor, two aspects of the **External Variable** panel are unique when used to define a zone curve:

Variable Name—Rather than requiring you to enter a **Variable Name** for the **External Variable**, ResSim will generate a name for the using the form: *ReservoirName-ZoneName,* where *ReservoirName* is the name of the current reservoir and *ZoneName* is the name of the zone for which you are specifying the independent variable.

Define Zone with Time-Series checkbox — this option is included at the bottom of the External Variable panel. By checking this option, you are telling ResSim that the data in the **External Variable** time series you will provide the alternative explicitly defines the zone elevation and no **Function of**: relationship is needed. As such, after you complete your setup of the External Variable and click **OK**, the function table in the **Zone** editor will be greyed out, indicated that the function is not required.

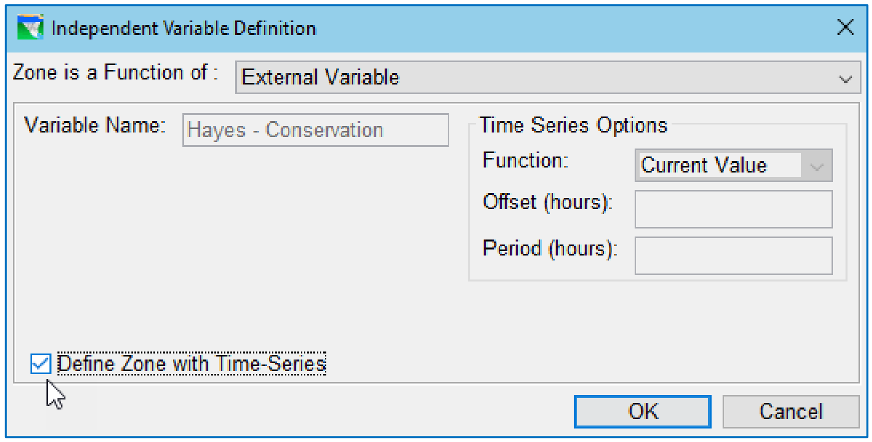


Figure 10.10 Zone as a Function of External Variable

New

Two Variable (Figure 10.11): This is a new option for a zone definition. It allows you to define the top of zone Elevation as a function of two independent variables. The expected use of this option is to define the guide curve as a function of both Date and an External Variable that represents the forecasted inflow volume expected over the remainder of the season.

The first independent variable, the **Row Variable**, can be one of:

- **Date**,
- **Date and Time**,
- **Model Variable**,
- **External Variable**, or
- **State Variable**.

The second independent variable, the **Column Variable**, can be one of:

- **Model Variable**,
- **External Variable**, or
- **State Variable**.

Don't forget to set the **Interpolation** type and **Time Series Options** for each variable (as needed). The values for the second independent variable must be defined for your function; use the Edit Column Values button open the editor to enter the values for the second variable.

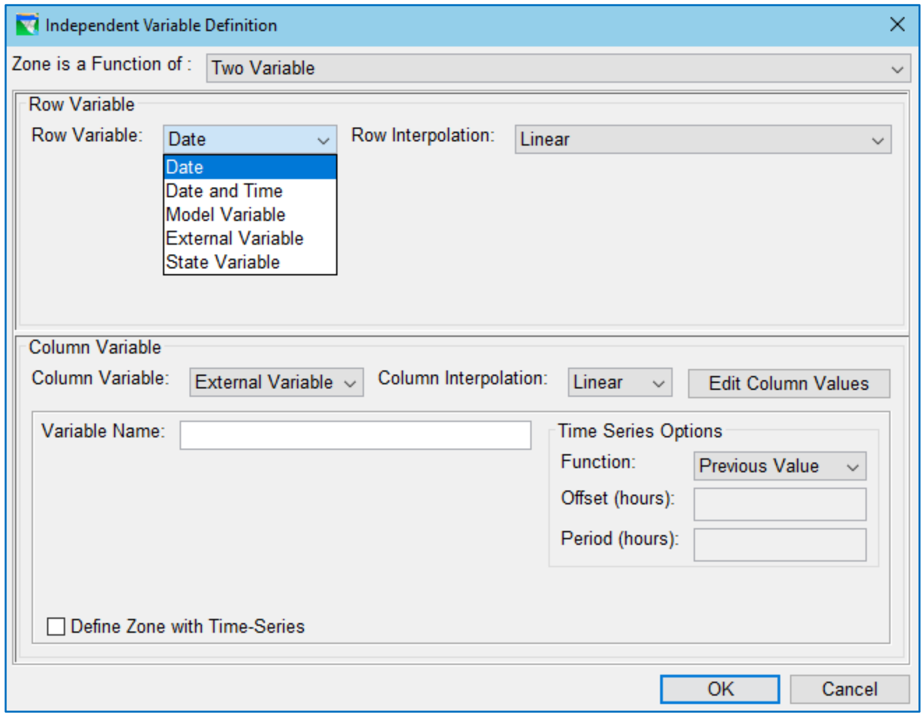


Figure 10.11 Zone as a Function of Two Variables

- When you have finished defining the top of zone curve for the current zone, be sure to:
  - Specify the zone's **Zone Sort Elevation** value; this value tells ResSim the sort order of the zones in the **Zone-Rules** tree and their relative elevation so they can be illustrated in the thumbnail plot.. If your zone is a function of **Date** (or **Date and Time**), the Zone Sort Elevation field is greyed-out and ResSim uses the elevation for 01Jan as the Zone Sort Elevation.
  - Click **Apply** to save your settings before moving on to the next zone.

Be careful when defining each top-of-zone curve — zones may be coincident, but they should not cross one another.

10.3.3 Renaming and Describing Operation Zones

To change the name of a Zone as it appears in the Zone-Rules Tree and/or edit its description:

- Select the zone (to be renamed) in the **Zone-Rules Tree**.
- Select **Rename** from the **Zone** menu or select **Rename** from the context menu by right-clicking on the zone you wish to rename in the Zone-Rules Tree. Enter the new name and/or description. The **Description** field for the Storage Zone is also editable in the Edit panel of the reservoir editor.

Any changes you make to the **Zone** name will be reflected in the **Zone-Rules Tree**.

Renaming the Inactive zone does not change its nature. The inactive zone is still a special zone from which the reservoir cannot release water and in which no rules can be added.

10.3.4 Deleting Operation Zones

To delete a zone from an operation set:

- Select the Zone you wish to delete from the Zone-Rules Tree.
- Select **Delete** from the **Zone** menu or select **Delete** from the context menu by right-clicking on the zone you wish to delete in the Zone-Rules Tree.
- Click **OK** in the **Delete Storage Zone** dialog to complete the delete process.

10.4 Selecting the Reservoir Guide Curve

The top of zone curve of any zone in your reservoir can be selected to represent the Guide Curve (i.e., target elevation) of your reservoir. The guide curve (or rule curve) represents the basic objective of the reservoir – get the pool elevation to, and hold it at, the guide curve. Without any other operational constraints, the decision logic will attempt to get to and keep the reservoir at the guide curve, within maximum outlet capacity and physical rate of change constraints. By *default*, the zone initially labeled ***Conservation*** is selected as the guide curve.

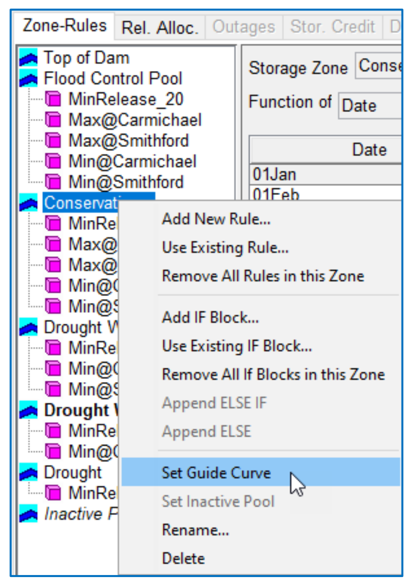


Figure 10.12 Set Guide Curve

To select a different zone to represent the guide curve for your reservoir operation set:

- R*ight-click* on the zone to be used as the guide curve in the **Zone-Rules Tree** and select **Set Guide Curve** (Figure 10.12) from the context menu.
- The name of the zone that is defined as the **Guide Curve** will be displayed with **bold** text in the **Zone-Rules Tree**.

10.5 Identifying the Inactive Pool

The *Inactive Pool* is a special zone in the reservoir; its top of zone curve represents the level below which the reservoir may not release water. Even if it is physically possible to release water below the top of the *Inactive Pool*, the reservoir is not allowed to do so. As such, rules cannot be placed in the *Inactive Pool* since, operationally, the allowable release range is already zero. Any zone in your reservoir can be selected to represent the *Inactive Pool* of your reservoir. By *default*, the zone initially labeled ***Inactive*** is selected as the *Inactive Pool*. The name of the zone that is identified as the *Inactive Pool* will be *italicized* in the **Zone-Rules Tree.**

To select a different zone to represent the *Inactive Pool* for your reservoir operation set:

- *Right-click* on the zone in the **Zone-Rules Tree** to be used as the *Inactive Pool* and select **Set Inactive Pool** from the Zone's context menu.

10.6 Specifying Release Allocation

As we'll see in the next chapter when we create rules, rules can be specified to apply to a specific outlet, thus imposing an "allocation" of the release determined for that rule to the identified outlet. Although rule-based allocation takes precedence, there are usually releases that are not specifically allocated by the rules; these are typically the releases made due to rules or operations that are applied to the dam or reservoir as a whole and are the ones the Release Allocation feature is intended to manage.

Release Allocation is the method ResSim uses to allocate the release from the reservoir to its outlet groups and their available outlets. The Release Allocation feature is available on the **Rel. Alloc** sub-tab of the Reservoir Editor's Operations tab. This feature enables you to specify the release allocation method by weighting or prioritizing the outlets in your reservoir's outlet hierarchy.

Three Release Allocation methods are currently available in ResSim:

- **Balanced**—releases distributed based on default or user-defined weights,
- **Sequential**—releases distributed sequentially to outlets prioritized by ranking, and
- **Stepped**—combines the balanced and sequential approaches.

By default, HEC-ResSim uses an *evenly balanced* (or weighted) allocation scheme to share the unallocated release across the *available* (see Section 12.5) outlets of a reservoir. The default *Balanced* allocation scheme is illustrated in Figure 10.13 below.

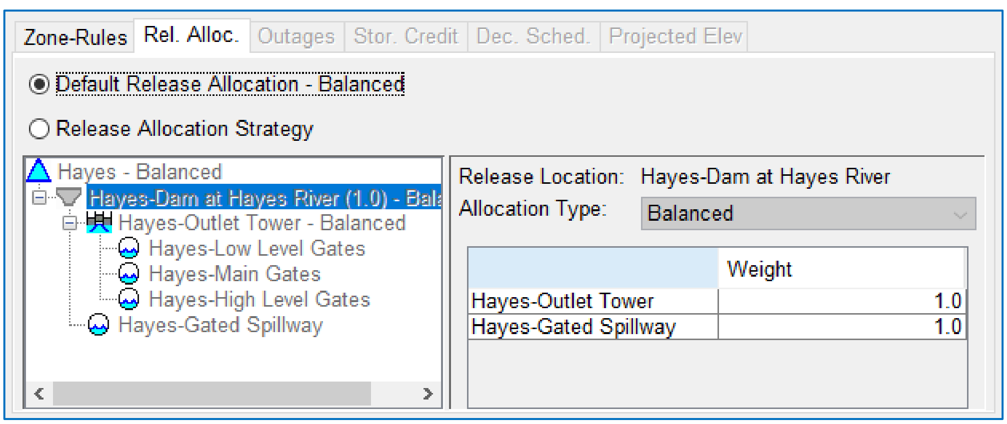


Figure 10.13 Release Allocation Editor—Default Allocation—Balanced

However, this default allocation scheme may not be desirable or appropriate for one or more of your reservoirs. For example, in many hydropower reservoirs, *all* releases are directed to the power plant first (to achieve incidental power generation); other outlets would be used only when the power plant has reached capacity (or when the desired release is too small to send through the power plant efficiently). For this example, a sequential or *prioritized* scheme for allocating the release should be used.

To edit the reservoir's **Release Allocation** scheme for the active Operation Set:

- Select the Release Allocation sub-tab (Figure 10.14) of the Reservoir Editor's Operations tab to access the Release Allocation editor.

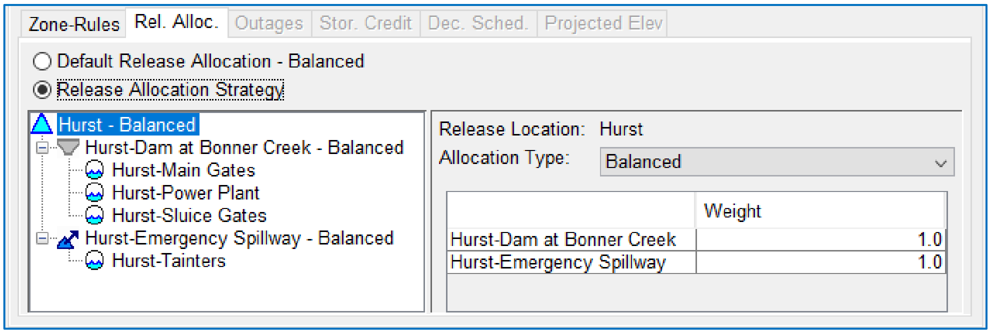


Figure 10.14 Release Allocation Editor—Balanced Allocation—Even Balance Example

- In the outlet tree at the left, select the group for which you want to define the allocation scheme. The group may be the reservoir itself, the dam, or an outlet group that you defined in the reservoir's outlet hierarchy on the Physical tab of the Reservoir Editor. The release allocation is defined at all but the lowest level of the outlet hierarchy.
- Next, select the **Allocation Type** and specify the associated weights or order for the outlets or outlet groups within the selected group. Your options are:

Balanced—Each outlet in the outlet group is assigned a **Weight** for the **Balanced** allocation method. The weights, entered in the table to the right of the outlet hierarchy tree, will be used to distribute the outlet group's release between its outlets. ResSim will normalize the values entered in the table to produce a set factors that are then applied to the group's total release to produce the release for each outlet within the group.

In the example illustrated in Figures 10.13 and 10.15, the **Allocation Type** for both Hurst (the reservoir) and Hurst-Dam is **Balanced**. The weights specified in the table for Hurst (the reservoir) define an *even* balance (i.e., a 1:1 ratio) between the Dam (1.0) and the Emergency Spillway diverted outlet (1.0). The weights specified in the table for Hurst-Dam define an uneven balance (or distribution) among the available outlets in the Dam; the Power Plant is allocated 50% of the release from the Dam while the Main Gates and Sluice Gates are allocated 30% and 20% of the Dam release, respectively.

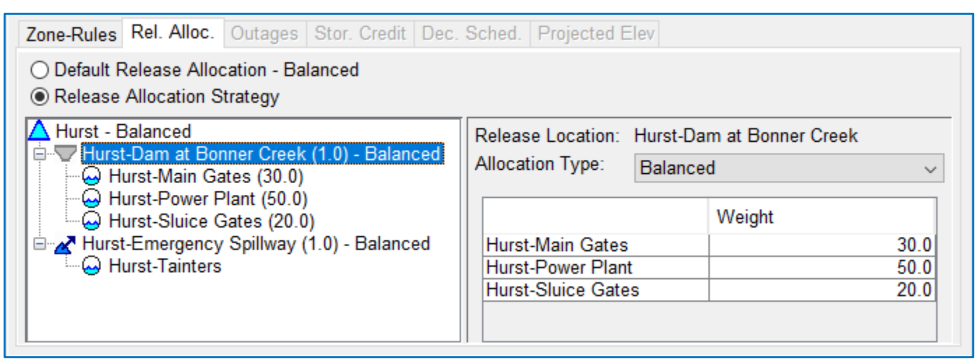


Figure 10.15 Release Allocation Editor—Balanced Allocation—Uneven Balance Example

Sequential—To define the Sequential allocation type, the available outlets are assigned an order of priority, indicating which outlet gets the release first. When the first outlet it reaches capacity, the next outlet gets the remainder of the release until it reaches capacity, and so on.

In the example illustrated in Figure 10.16, the group called Sayers-Dam has been assigned the **Sequential Allocation Type**, and the outlets within that group have been ordered so that the Power Plant is first, then the Sluice Gates, and the Main Gates are last.

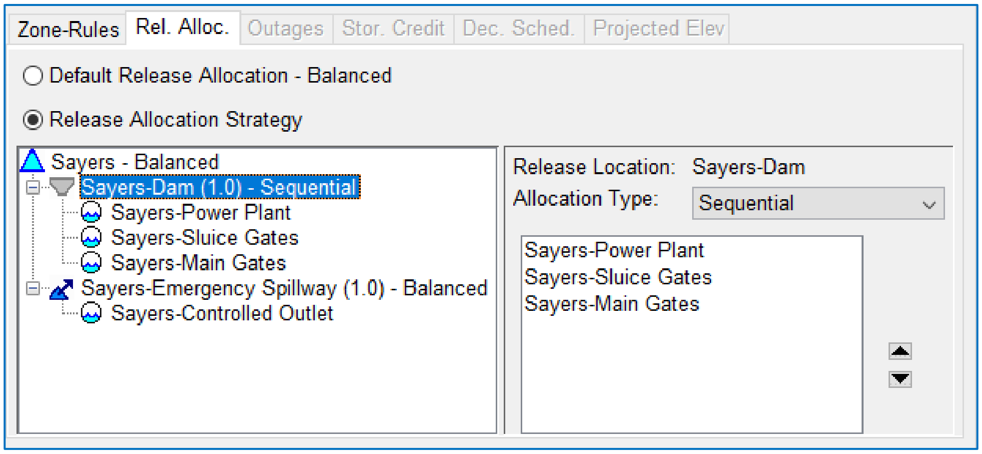


Figure 10.16 Release Allocation Editor—Sequential Allocation Example

To set up the **Sequential** allocation type and define the order of the outlets:

- Highlight the appropriate component (e.g., the "Dam") in the left panel of the editor.
- In the right panel of the editor, select **Sequential** from the **Allocation Type** selector. Below the **Allocation Type** selector, a text box will display the list of outlets in the selected outlet group.
- Select an outlet in the list and use the arrows located on the right side of the textbox to move the outlet up or down in the list. The outlet at the top of the list will be the first to release (up to its capacity), then the next outlet in the list will release (if needed, up to its capacity), and so forth.

Stepped—The **Stepped** allocation type is a combination of the **Balanced** and **Sequential** allocation types. Stepped allocation allows for distribution of a reservoir release among the outlets of the outlet group with the portion of the release allocated to each outlet varying over a specified range of outlet capacities.

The distribution of the group's total release across its outlets, and varying by step, is defined using a column for each outlet and rows describing the steps (or ranges of outlet capacity). A dropdown menu that defines *units* for values entered can be selected from the **Table Units** selector. The Stepped allocation method is designed to allow two or more options for the **Table Units**, although only one option has been implemented for ResSim version 3.3 – **% Release Capacity**.

To define the Stepped allocation method, enter values into the table, in decreasing order of *% Release Capacity*, to describe how the total release to the group should be distributed to each outlet. Each value represents that outlet's *share* of the total release that its row in the table can accommodate.

The table always starts with a row in which a value of 100% is entered for each outlet. This means that if the total release assigned to the group is equal to the sum of the total capacity of each outlet, then each outlet will be allocated 100% of its capacity to release.

Enter values in each succeeding row such that the *% Release Capacity* value for at least one of the outlets is less than the value entered for that outlet in the previous row. The sum of the flows represented by the % capacity value for each outlet in a given row equals the total release that can be made by the group using that row's distribution.

If needed, additional rows can be added to the table by right-clicking in a cell of the table and selecting **Insert Row(s)** or **Append Row** from the table's context menu. **Insert Row(s)** will add the number of (blank) rows you specified into the table above the row of the cell from which you accessed the context menu. **Append Row** will add a blank row below the last row of the table.

It is not necessary to enter a final row where the allocation to each outlet is 0% since that will by assumed by ResSim. However, that (assumed) last row (of all zeros) represents a total group release of 0 cfs.

At each timestep, ResSim will calculate the release that each value in the table represents based on the current release capacity of the assoicated outlet. It then sums these release values by row to determine the total release that can be made by the group using that row's distribution. Next, ResSim determines the two successive rows that bound the total release allocated to the group and uses linear interpolation to determine the release each outlet will make.

Although defining the data for the Stepped allocation type is not as intuitive as defining the other two allocation methods, the following example should clarify how the data will be used and thus help you to determine the values to enter to produce the desired allocation.

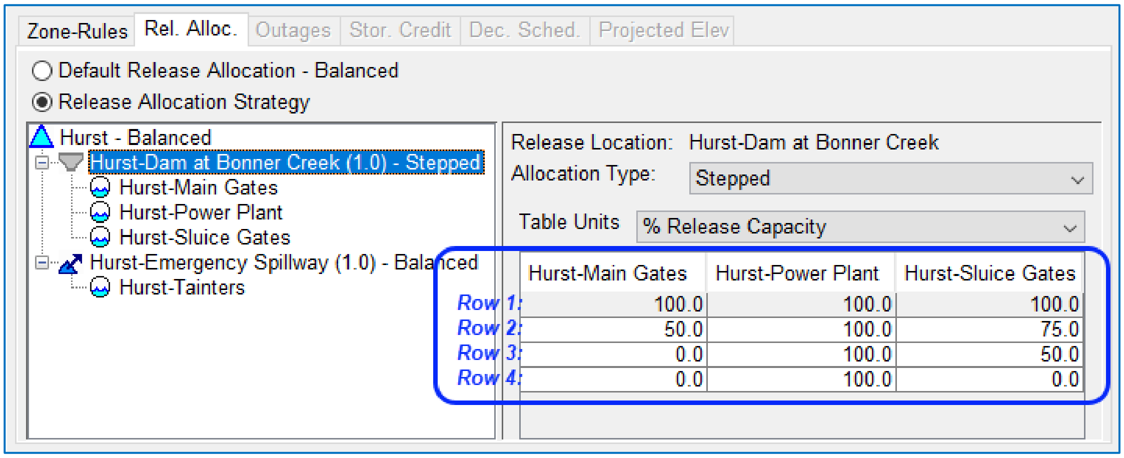


Figure 10.17 Release Allocation Editor—Stepped Allocation Example

In the Stepped release allocation example shown in Figure 10.17 , the Sayers-Dam outlet group has been assigned the **Stepped** allocation type and the table is filled with values (in units of % Release Capacity) to define how the total release to the group should be divied up between the outlets. To understand how these values will be used, read the entries in the table from bottom up.

The bottom row (row 4) indicates that the total dam release, if less than or equal to 100% of the Power Plant's capacity, should go to the Power Plant.

The next row up (row 3) indicates that if the total dam release is less than or equal to 100% of the Power Plant's capacity ***plus*** 50% of the Sluice Gates' capacity, the releases to Sayers-Dam are intended to go to the Power Plant first and then to the Sluice Gates.

The next row up (row 2), as well as the top row (row 1), indicate that if the total dam release is greater than the sum of the Power Plant's capacity plus 50% of the Sluice Gate's capacity, the excess is shared between the Sluice Gates and the Main Gates. The distribution of that excess between the Sluice Gates and the Main Gates appears to be 1:2, but it's not quite that simple because the values in the table are not % Total Release but rather % Release Capacity. As such, the weighting of the release between the two outlets is a function of their capacity at any given time.

The significance of the use of % of Release Capacity can be seen by looking at at the arithmetic for the state of the system at *timestep* i :

- The **Power Plant's** release *capacity* = 5,000 cfs
- The **Sluice Gates's** *release* capacity = 5,000 cfs
- The **Main Gates's** *release* capacity = 20,000 cfs

Using these capacities, the table of releases computed for the timestep is:

Row #	Main Gates (%: cfs)	Power Plant (%: cfs)	Sluice Gates (%: cfs)	Total Release (cfs)
1	100: 20,000	100: 5,000	100: 5,000	30,000
2	50: 10,000	100: 5,000	75: 3,750	18,750
3	0: 0	100: 5,000	50: 2,500	7,500
4	0: 0	100: 5,000	0: 0	5,000
	0: 0	0: 0	0: 0	0

If the ***total release*** to the Dam is 9000 cfs, then the release falls between rows 2 and 3. Using linear interpolation with the data in rows 2 and 3, the calculations for divvying up the 9000 cfs release are:

- Total Flow Ratio = 9000 – 7500/ 18750 – 7500 = 0.13333
- Power Plant = 5000 + 0.13333 x (5000 – 5000) = 5000
- Sluice Gates = 2500 + 0.13333 x (3750 – 2500) = 2666.66
- Main Gates = 0 + 0.13333 x (10000 – 0) = 1333.33

Chapter 11
Reservoir Operations — The Rules

Operation Rules represent the operational goals and constraints for each zone of the operation set. Each zone can contain a different set of rules depending on how the regulation plan describes the flow limits and requirements for that operating zone.

As previously described in Section 10.3.2 and illustrated in Figure 10.8, as you create and arrange your operating zones and rules in the current operation set, they are displayed in the **Zone-Rules Tree** on the left side of the **Operations** tab of the **Reservoir Editor**. The order of the rules in each zone indicates their relative priority.

In Chapter 10, you learned about guide curve operation and operation zones. In this chapter, you'll learn about the variety of Operation Rules you can create to describe the goals and constraints on the operation of your reservoirs. But, to understand how rules work, it is important to understand the release decision process and how it uses the rules and their relative priority.

The Release Decision Process

The ResSim decision logic for determining the reservoir releases in each timestep is called the Release Decision Process. It involves determining an *allowable release range*, calculating the *desired guide curve release*, and comparing the two to arrive at a release decision.

The *allowable release range* is defined by a minimum allowable release and a maximum allowable release. The reservoir's physical and operational constraints are used to determine the allowable release range.

The *desired guide curve release* is, by definition, the release the reservoir should make to get to or stay at guide curve in this timestep. This calculation derives from the basic Conservation of Mass Equation:

Inflow minus *Outflow* equals *Change in Storage*.

There are few limitations on how the desired guide curve release is computed since guide curve operation is the primary objective of the reservoir operation. However, the computations do include logic to limit or prevent oscillating releases as the reservoir pool elevation approaches the guide curve.

The steps that make up the release decision process are:

Estimate Physical Limits—The decision logic begins by estimating the physical capacity of the reservoir to release water, given the reservoir's state at the start of the timestep and an estimate of the pool elevation at the end of the timestep. Unless the dam leaks, the physical minimum release limit is usually zero. The physical maximum release limit is a function of the estimated average elevation for the timestep. The physical release limits establish the initial values for the minimum and maximum limits of the ***allowable release range***.

Identify the Current Zone—To determine the current zone, the decision logic computes the elevation value at the end of the current timestep from each top-of-zone curve or relationship. Then, working from the bottom up (based on the zone-sort order), it compares the reservoir pool elevation computed at the end of the *previous* timestep to each "current" top-of-zone elevation until it finds the zone elevation value that is greater than or equal to the current pool elevation.

Identify the Active Rules—If necessary, the decision logic evaluates all relevant State Variables, IF-Blocks, and Rule Modifiers to identify the active rules in the current zone and assemble them in an ordered list based on priority.

Apply the Rules—The decision logic then works its way through the list of active rules *from the highest priority rule to the lowest*. As each rule is evaluated, its current release limit is applied to the allowable range. The maximum allowable release is reduced if a rule calls for a lower maximum, and the minimum allowable release is increased if a rule calls for a higher minimum. However, if a rule has a desired maximum limit that is greater than the current maximum allowable release or has a desired minimum limit lower than the current minimum allowable release, the allowable range will remain unchanged. Since rules are evaluated from highest to lowest priority, the allowable range can never be widened, because the rule attempting to do so would violate the higher priority rule or physical constraint that set the limit.

Calculate the Guide Curve Release—After evaluating all the rules, the decision logic then determines the desired guide curve release. This is the release needed to bring the reservoir pool elevation to the guide curve in the current timestep (computation interval) based on the starting pool elevation, the prior release, and the current inflow.

Determine the Release—In this final step, the decision logic compares the guide curve release to the limits of the allowable range. If the guide curve release is within the allowable range, the *release decision* will be the guide curve release. However, if the guide curve release is outside the allowable range, the release decision will be the limit of the allowable range nearest the guide curve release.

For example: if the desired guide curve release for this timestep (guide curve release) is 35,987 cfs but the final maximum limit of the allowable release range is 10,000 cfs, then the release decision will be 10,000 cfs.

Why are rules "prioritized"?

Sometimes, the rules that describe the desired operation of the reservoir conflict with one another and a means for resolving the conflict is needed. A conflict is when two rules call for a desired release or a release limit that cannot both be satisfied.

For example, a reservoir might have a minimum release rule of 500 cfs and a maximum downstream control rule of 12,000 cfs. At a given timestep during high flow conditions, the downstream control rule evaluates to a maximum release of 100 cfs from the reservoir. If the reservoir releases 500 cfs, the downstream rule is violated. If the reservoir only releases 100 cfs, the minimum flow rule is violated.

ResSim uses rule prioritization as the method for resolving rule conflicts — *the highest priority rule wins*. And, it is up to you, the modeler, to order your rules appropriately for all circumstances.

The fundamental rule for applying rules to the allowable release range is:

As each rule is applied, from highest priority to lowest, the allowable range may narrow, by reducing the maximum limit or increasing the minimum limit, but it may not widen.

> ***Therefore, a minimum limit rule may raise but not lower the minimum limit of the allowable range.***
>
> For example, if the current minimum allowable release is 100 cfs, a minimum limit rule of 500 cfs may raise the minimum allowable release to 500. This is allowed because a release of 100 cfs is still met by a 500 cfs release so the higher priority rule (min 100) is not violated by the lower priority rule (min 500).
>
> But, if the next rule calls for a minimum release of 10 cfs, the allowable range would not change. The minimum would stay at 500 cfs. This is not considered a violation of the 10 cfs minimum rule, since a minimum release of 500 cfs still meets the minimum 10cfs objective
>
> ***And, a maximum limit rule may lower the maximum limit of the allowable range but not raise it.***
>
> For example, if the current maximum allowable release is 9000 cfs, a maximum limit rule of 5000 cfs may lower maximum allowable release to 5000. Remember a maximum limit of 9000 allows for the release to be any value less than 9000. By reducing the maximum limit to 5000, a value less than the previous limit of 9000, the 9000 cfs limit is not violated.

These examples can be considered illustrations of the following two ***Rules of Thumb*** that you should remember when prioritizing rules:

> **When two maximum-limit rules are beside one another in the rule stack, their relative priority doesn't matter – the smaller max wins** (sets the maximum limit of the allowable range).
>
> **When two minimum rules are beside one another in the rule stack, their relative priority doesn't matter – the larger min wins** (sets the minimum limit of the allowable range).

11.1 Managing Rules

Before we discuss each of the available rule types and all their parameters, we are going to learn about how to manage rules in general.

The **Zone-Rules Tree** of the **Zone-Rules** sub-tab of the **Operations** tab of the **Reservoir Editor** (Figure 11.1) is the primary tool for interacting with the rules in your operation set.

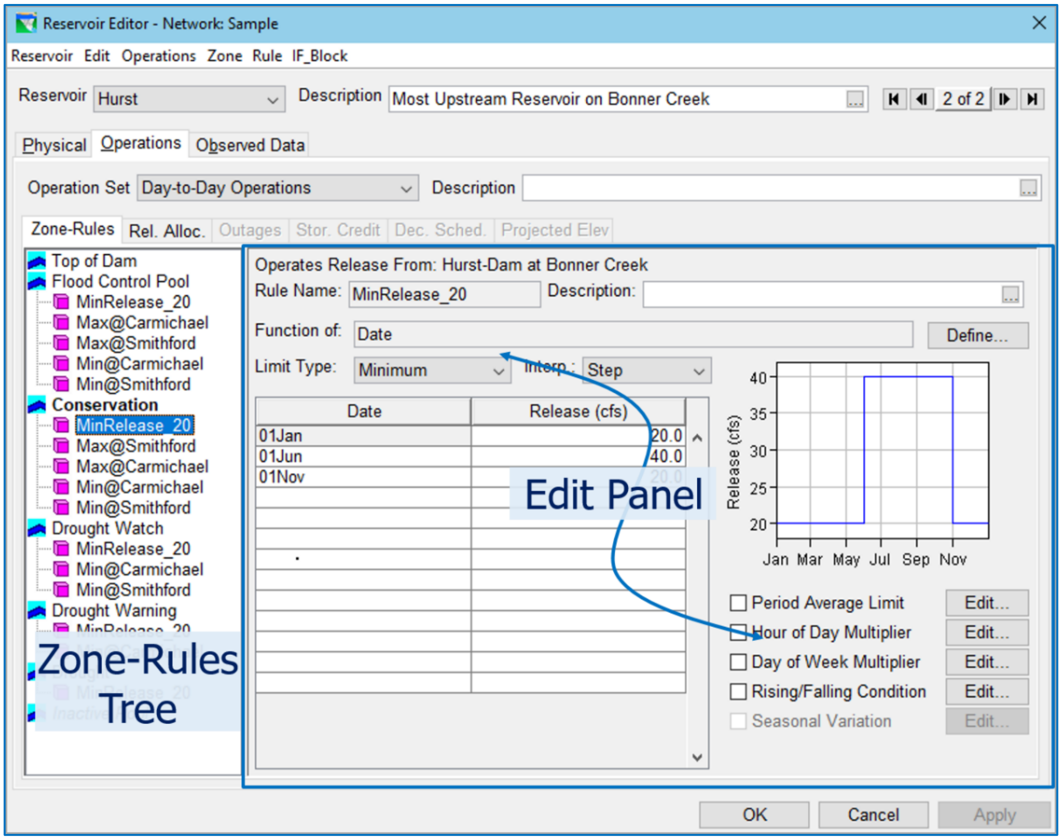


Figure 11.1 ResSim Rule Editor

When you *click* on a zone or rule in the tree, you have *selected* that element of the operation set, and the **Edit Panel** to the right of the tree displays the corresponding zone or rule editor.

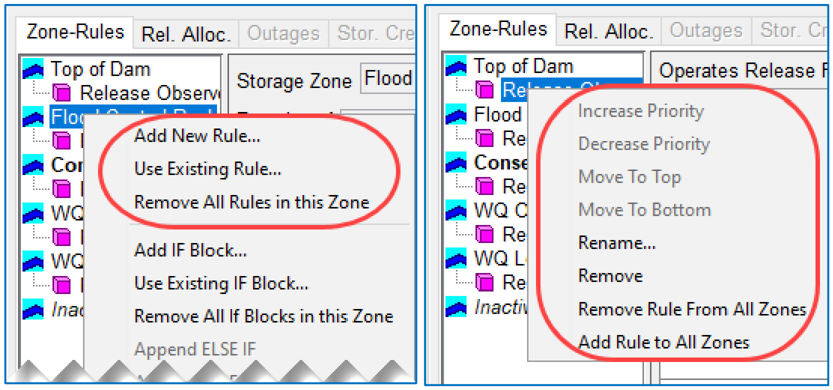
The **Rule Editors** that display in the **Edit Panel** have a few common attributes which are located at the top of each editor:

Operates Release From: this field identifies the *release element* that was chosen when the rule was created. The release element that the rule applies to cannot be changed once a rule has been created.

Rule Name: this field displays the name you entered when you created or renamed the rule. Rule names should be short but reasonably descriptive of the type, purpose, and/or objective of the rule.

Description: this field is editable and does not have a length limit. Be sure to enter a description for each rule you create. Your description should include the purpose of the rule, what operating constraint it represents, and any other important characteristics. To open a larger description editor/viewer, click the ellipsis button ....

When you *right-click* on a zone or rule in the tree, its context menu will display (Figure 11.2).



Zone Context Menu

Rule Context Menu

Figure 11.2 Rule Management Functions in the Zone and Rule Context Menus

A zone's context menu provides functions for adding or removing rules from the selected zone. A rule's context menu includes functions for renaming that rule, managing that rule within the current zone, and adding or removing the rule from all zones in the current operation set. Most of these rule management functions can also be found in the **Zone** and **Rule** menus that are included in the menu bar of the **Reservoir Editor** when the **Operations** tab and its **Zone-Rules** sub-tab are active. Because of this duplication between the context menus and the **Zone** and **Rule** menus (illustrated in Figure 11.3) and to minimize repetitive descriptions, the following sections will provide instructions primarily using the context menu. Use the menu you are most comfortable with (the menu in the menu bar or the context menu from the tree nodes).

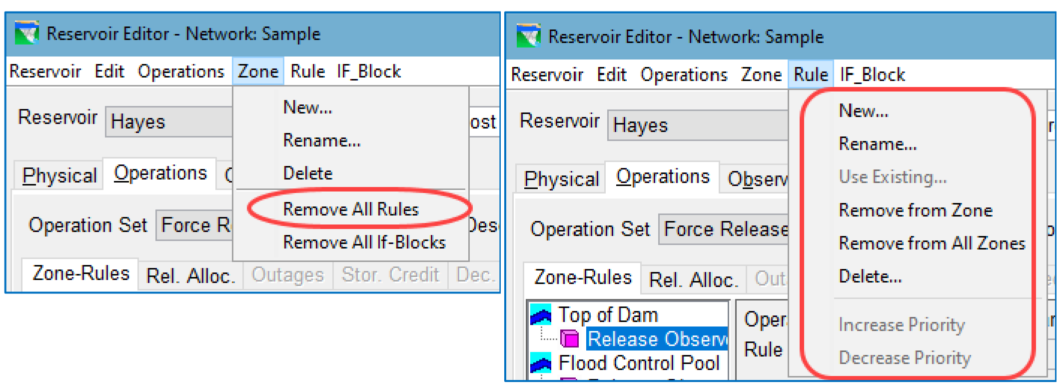


Figure 11.3 Rule Management Functions in Zone and Rule Menus in Menu Bar

11.1.1 Creating New Rules

When you create a rule, you must:

- identify the *release element* that the rule constrains,
- select the *rule type*, and
- give the rule a *name* (and an optional, but recommended, description).

You learned about release elements in Chapter 9. They include the reservoir itself, the dam, the outlet groups, diverted outlets, and the outlets. The only release element to which you cannot assign an operation rule is an uncontrolled outlet because its releases cannot be controlled, and rules are all about controlling releases.

ResSim has several rule types to choose from. The availability of the rule types varies depending on the release element. Some rules can only be applied to a specific type of outlet while other rules can be applied to more than one type of release element. Table 11.1 lists the various release elements and the rule types that can be applied to them.

Table 11.1 Matrix of Rule Relese Element and their Available Rule Types

Release Element/ Rule Type	Release Function	Downstream Control	Flow Rate of Change	Elevation Rate of Change	Tandem Operation	Hydropower Demand	Pump Schedule	Scripted	Prescribed Release New
Reservoir	✓	✓	✓	✓	✓			✓	✓
Dam or Group	✓		✓					✓	✓
Controlled Outlet	✓		✓					✓	✓
Power Plant	✓		✓			✓		✓	✓
Pump							✓		
Uncontrolled Outlet	Rules cannot be applied to an uncontrolled outlet								

To add a new operation rule to a zone:

- *Right-click* on the zone in the **Zone-Rules Tree** to which you want to add the new rule.
- Select **Add New Rule…** from the zone's context menu. The **New Operating Rule** dialog will open (Figure 11.4).

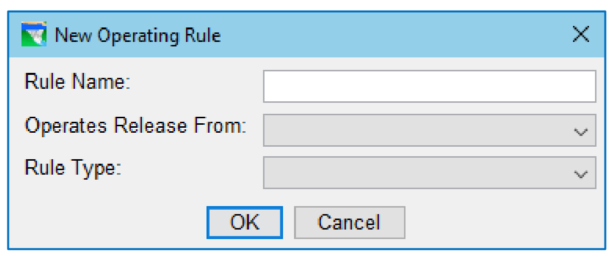


Figure 11.4 New Operating Rule

- The three fields of the **New Operating Rule** dialog include:

  Rule Name:—Enter a name for the new rule. You may want to enter this *after* specifying the other two parameters

  Operates Release From:—Select the *release element* of the reservoir to which you will assign this rule (reservoir pool, dam, outlet group, or outlet)

  Rule Type:—Select the rule type from the selection list. The available rule types in the list will depend upon your selection for **Operates Release From.**

The new rule will appear, highlighted, at the bottom of the set of rules in the selected zone in the **Zone-Rules Tree** and the edit panel will display the appropriate rule editor for the new rule's type. The first two fields for each rule editor contain the name of the rule and its description. The description of the rule can be added or changed whenever the rule editor is displayed in the edit panel.

11.1.2 Using Existing Rules

A rule can be used in more than one zone and in more than one operation set.

Note: Rules belong to the reservoir *in the current network*, not to a zone or to an operation set. So, when you see a rule in a zone, it is a *reference* to the rule, NOT a *copy* of the rule. Thus, a change to a rule in a zone carries through all operation sets and zones that use that rule *in the current Network.*

To add a rule that you have already created to a zone:

- *Right-click* on the zone you want to add the rule to and select **Use Existing Rule…** from the context menu. The **Select Existing Rule** dialog will open (Figure 11.5).

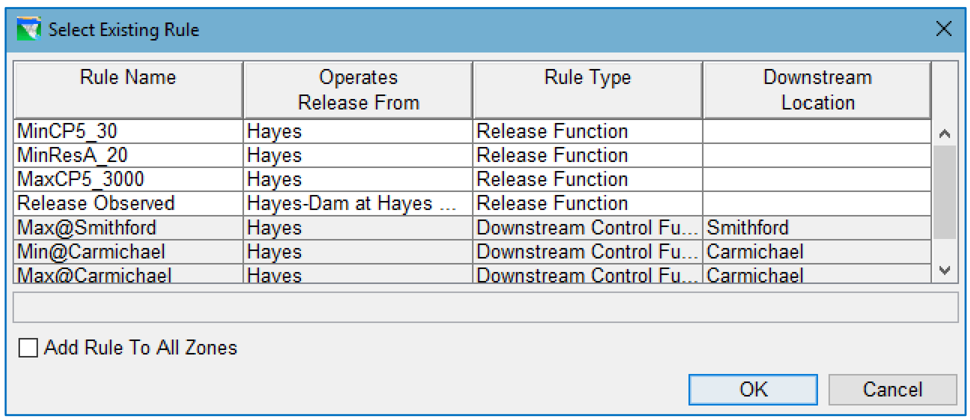


Figure 11.5 Select Existing Rule

- Select the rule you want to add from the *white* rows in the table. The *grey* rows in the table are rules that are already being used in the current zone.
- Click **OK** to add the rule and close the **Select Existing Rule** dialog. (**Cancel** will close the dialog without adding a rule to the zone.)
- The selected rule will appear at the bottom of the list of rules in the selected zone. See Section 11.1.3 for instructions on prioritizing rules.

To add a rule to ***all*** zones in the current operation set:

- Follow the instructions above but place a checkmark in the **Add Rule to All Zones** checkbox before clicking **OK** in the **Select Existing Rule** dialog.

If you have a rule in one zone that you want to add to all zones in the current operation set:

- *Right-click* on the rule and select **Add Rule to All Zones** from the rule's context menu.

11.1.3 Prioritizing Rules

Each time you add a rule to a zone, it is placed at the bottom of the rule stack. The rule stack is the list of rules in the zone. The order of the rules in the rule stack indicates their relative priority—the highest priority rule is first in the list (at the top) and the lowest priority rule is at the bottom of the list.

To raise or lower the priority of a rule within the rule stack of a zone:

- *Right-click* on the rule in the zone and select one of the move functions from the context menu:

  Increase Priority—moves the rule up one position in the list
  Decrease Priority—moves the rule up one position in the list
  Move to Top—moves the rule to the top of the list
  Move to Bottom—moves the rule to the bottom of the list

NOTE: The prioritization (move) functions only apply to the position of the selected rule in the *current zone* of the current operation set. The rule stack of each zone is prioritized independently of all other zones and all other operations sets.

Some rules are designed to work best when applied at the top or bottom of the rule stack, so pay special attention when prioritization of a rule is discussed in that rule's description later in this chapter. For example, **Induced Surcharge** rules should be placed at the top of the rule stack and **Tandem** rules should be at the bottom.

11.1.4 Removing Rules

Removing a rule from a zone does not *delete* the rule from the reservoir. The rule still exists in the reservoir for inclusion in other zones and operation sets. Several options are available for removing rules from a zone or zones of the current operation set. They include:

- Remove a rule from a selected zone
- Remove a rule from all zones
- Remove all rules from a selected zone

To remove *a rule* from a selected zone…

- *Right-click* on the rule you want to remove in zone you want to remove it from in the **Zone-Rules Tree** and select **Remove** from the context menu.
- A confirmation dialog will appear asking you to verify that you want to remove the rule (Figure 11.6). Click **OK** to complete the removal (or **Cancel** to abort the process). The rule will disappear from the current zone.

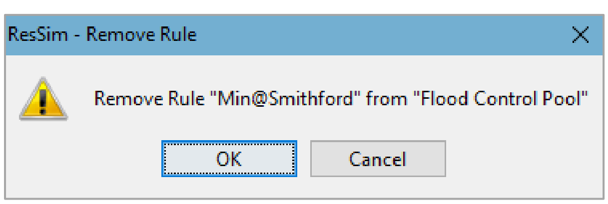


Figure 11.6 Confirmation Dialog—Remove Rule

To remove a rule from *all zones*:

- Select **Remove Rule From All Zones** from the context menu.

To remove *all rules* from a selected zone:

- *Right-click* on the zone and then select **Remove All Rules in this Zone** from the context menu.

11.1.5 Deleting Rules

Delete is the only rule management function that is not available from the rule's *context menu*. To delete a rule from the reservoir:

- Select the **Delete** option from the **Rule** menu. This will open the Delete Rules dialog (Figure 11.7).

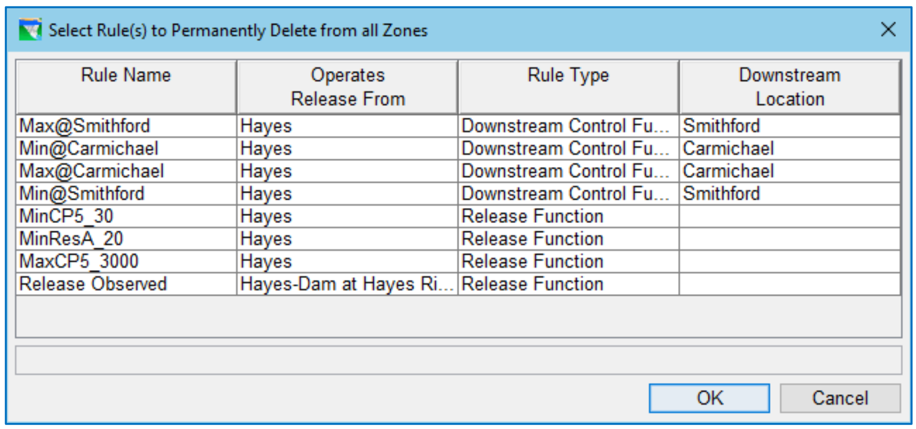


Figure 11.7 Delete Rules Dialog

- As the title bar implies, the Delete Rules dialog will allow you to select one or more rules to delete. *Clicking* on a row in the table will select a rule. To select multiple rules, use the **Shift** or **Ctrl** keys on your keyboard when you *click* a row.
- Click **OK** to accept your selection(s) and close the Delete Rules dialog.

Use caution when deleting rules!

The selected rule(s) will be deleted from all zones *and* operation sets of the *current reservoir* in the *current network*. *In addition*, downstream control and/or system hydropower rules will be deleted from all zones and operation sets of *all reservoirs* in the current network that used those rules.

11.1.6 Renaming Rules

To rename a rule:

- *Right-click* on the rule in the **Zone-Rules Tree** and select **Rename** from the context menu. The **Rename Rule** dialog will open.
- Enter a new name in the **Name** field.
- If desired, add or revise the description for the rule in the Description field.
- *Click* **OK** to complete the rename process. **Cancel** will dispose of your changes and close the dialog.

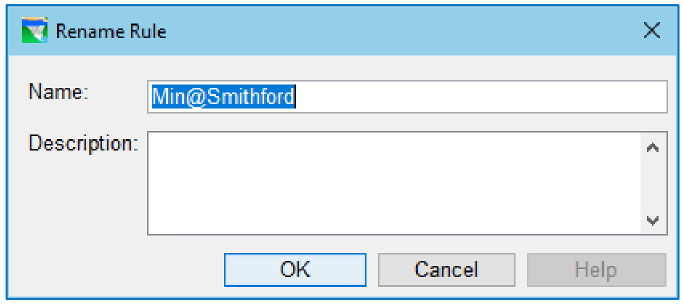


Figure 11.8 Rename Rule Dialog

Use caution when renaming rules. Because rules belong to the reservoir, when you rename a rule, all instances of the rule in all the zones and operation sets of the current network will be renamed.

11.2 Release Function Rules

The **Release Function** rule type is one of the two most flexible rule types available in ResSim (the other being the **Downstream Control Function** rule). Most of the rules you create for your reservoirs will be Release Function rules.

With this rule type, you can define a wide array of "function of" rules—meaning rules whose desired release (limit) is a function of the *current* date or pool elevation or inflow or pretty much whatever you can think of. A rule of this type can be assigned to any of the release elements — the reservoir (pool), the dam, an outlet, or an outlet group. And, this rule allows you to specify the maximum, minimum, or *specified* flow to be released through the release element.

The simplest **Release Function** rules are defined as a function of *Date*. A function-of-date release rule describes a seasonally-varying release limit whose seasonal pattern, which always starts on 1 January, repeats annually.

A *constant* release rule is usually specified as a function-of-date rule where only one date and its associated value is entered in the function table.

The more *complex* **Release Function** rules are defined as a function of a *time-series variable*. The time-series variable types are internal (model) variables, external variables, and state (scripted) variables.

A function-of-external-variable release rule is a little more abstract than a function-of-model-variable rule simply because the variable itself is provided as input and is not computed by ResSim. The variable and its relationship to the release is limited only by your imagination and the time-series data available.

Since state variables are user-defined variables whose value is computed in each timestep of the simulation by a Jython script written by the modeler, a Function of State Variable Release Function rule could represent almost any operating constraint.

Here are some example release function rules that could be found in a water control manual or reservoir regulation plan:

> "*The maximum release from the reservoir during the growing season is 6,500 cfs and 8,500 cfs during the non-growing season*" is an example of a seasonally-varying operating constraint. This can be represented with a function-of-date release rule.
>
> "*The reservoir must release at least 3 cfs or 10 percent of inflow up to a maximum of 100 cfs, whichever is greater*" is an example of a more complex operating constraint. Since inflow to the reservoir is one of several model variables computed by ResSim, this constraint could be defined as a function-of-model-variable release rule.
>
> "*During spawning season, releases from the reservoir should be guided by Table 7.3, which relates the forecasted maximum daily air temperature at the spawning grounds to releases from the low level outlets*" is a more abstract complex operating constraint that could be modeled with a function-of-external-variable release rule. Since ResSim cannot compute or forecast air or water temperature, a time-series record that describes the predicted or observed values for that variable would be required in order to determine each timestep's the release limit based on the function you define.
>
> "*The reservoir must release at least 4800 cfs unless system drought operations have been declared whereby the minimum release is 4000 for drought level 1, 3800 for drought level 2, and 3600 for drought level 3*" is an example of an operating constraint that would need a state variable and an associated release function rule to represent it. The state variable would calculate the current drought level and the rule would use that value to determine the rule's desired release limit for each timestep.

To define or edit a **Release Function** rule:

- Create a new rule as described in Section 11.1.1. Be sure to select **Release Function** for the **Rule Type** in the **New Operating Rule** dialog.
- Or, if necessary, select the Release Function rule you want to define/edit from the **Zone-Rules Tree**.

The edit panel of the **Operations** tab of the **Reservoir Editor** will display the **Release Function Rule** editor (Figure 11.9).

The following sections describe each field, option, or attribute of the **Release Function Rule** editor and how to define the associated data.

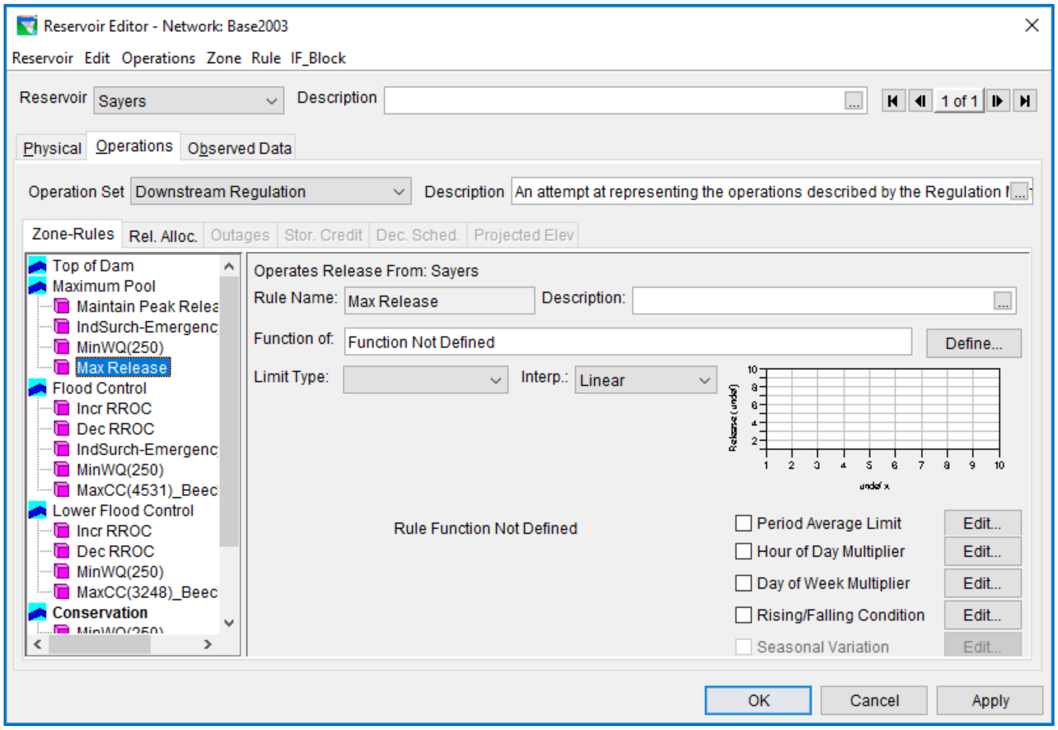


Figure 11.9 Reservoir Editor—Operations Tab—New Release Function Rule

11.2.1 Select the Limit Type

Remember, each rule you create describes a desired limit on the allowable range of release from the reservoir. Although some rules are inherently a specific limit type, the release function rule, with all its flexibility requires that you select the limit type as part of the rule definition.

Select that the **Limit Type** from the selection list (Figure 11.10). Your options are:

- **Minimum**
- **Maximum**
- **Specified**

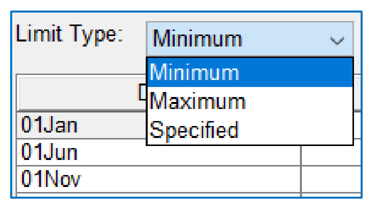


Figure 11.10 Rule Limit Type

Specify the limit type early in the process of defining your rule because it keeps your mind focused on your objective while you complete the rest of the rule definition.

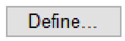
NOTE: Use the specified limit type with caution.
Most operational constraints describe either a minimum or maximum release limit. Specified rules describe the precise amount of flow to be released, neither more nor less, making them effectively both a minimum and a maximum limit at the same time. As such, a specified release rule is very restrictive because it sets the allowable range of the release to a single value. If specified is the appropriate limit type, consider carefully where to place the rule in the rule stack. Unless the rule applies to a specific outlet, the bottom of the rule stack is usually the best place for specified-limit rules; in this position, other rules have a chance to influence the allowable range.

11.2.2 Define the Function

The function (or Release Function) is the relationship between the desired release (limit) and the variable it is a "function of". The release function is defined by:

- The specification of the independent variable or variables (what the release is a "function of").
- The table of values describing the function
- The interpolation type to be used when looking up a value in the table.

11.2.2.1 Select the Independent Variable of the Function

Use the 

button to the right of the **Function of:** field to open the **Independent Variable Definition** editor (Figure 11.11). Although limited instructions for using the **Independent Variable Definition** editor are provided below, the full description of how to use this editor was placed in Appendix C to minimize duplication since the **Independent Variable Definition** editor is used in several of places throughout ResSim.

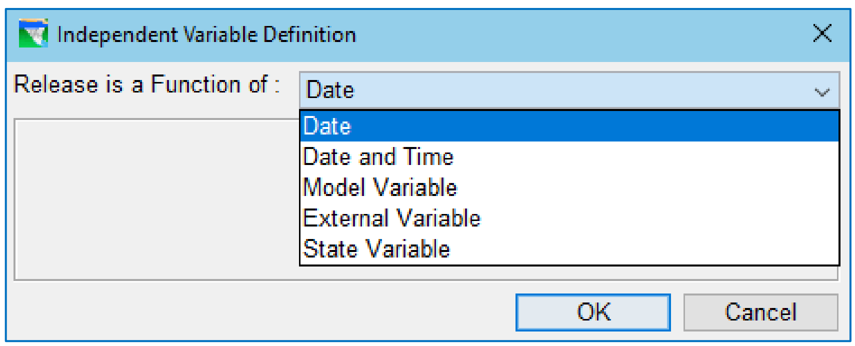


Figure 11.11 Release Function Rule—Independent Variable Definition Editor

Release is a Function of:—this field, at the top of the editor, is the i*ndependent variable* (**Date**, **Date and Time**) or *variable type* (**Model Variable**, **External Variable**, or **State Variable**) for your function.

Variable Selection Panel—below the **Release is a Function of:** field is a panel for identifying the specific variable.

If you select **Date** or **Date and Time**, no additional information is needed to define the independent variable so the Variable Selection Panel will be blank.

If you select **Model Variable** or **State Variable**, a table of available variables of that type will be displayed and you must select the specific variable for the independent variable of your function.

If you select **External Variable**, a single **Variable Name** field will be provided. Enter a unique name for the **External Variable**. Make the name descriptive enough that you will recognize it in the **Alternative Editor's Time Series** tab when it comes time to associate an HEC-DSS pathname to this variable.

Time Series Options—If you selected **Model Variable**, **External Variable**, or **State Variable**, the **Time Series Option** section is included on the right side of the Variable Selection Panel. These options allow you to define how the value of the independent variable is determined in each timestep before being used to look up the desired release from the release function.

Function—Choose a time series function from the selection list. The available functions include:

Previous Value—this function returns the value from the selected variable's time series that was computed at the end of the *previous* timestep. The *previous value* of several model variables, such as elevation, can be considered the starting condition of the current timestep.

Current Value—this function returns the value from the selected variable's time-series for the *current* timestep. NOTE: some variables, like *inflow*, are known for the current timestep while others, like *elevation*, ***have been computed yet*** because they are computed *at the end* of the timestep, *after* release decisions have been made.

Offset Value—this function returns a value from the selected time-series that is some number of hours *offset from the end of the current timestep*. This function requires a value in the **Offset** field.

Period Average—this function returns the *average* value from a set of values retrieved from two or more timesteps of the selected variable's time series. This function looks for values in both the **Offset** and **Period** fields to determine the range of values to use to compute the average.

Period Maximum—this function returns the *maximum* value from a set of values retrieved from two or more timesteps of the selected variable's time series. This function looks for values in both the **Offset** and **Period** fields to determine the range of values to use to determine the maximum value.

Period Minimum—this function returns the *minimum* value from a set of values retrieved from two or more timesteps of the selected variable's time series. This function looks for values in both the **Offset** and **Period** fields to determine the range of values to use to determine the minimum value.

Offset—the offset identifies a timestep in the selected time-series *relative to the current timestep*. Enter a value in **hours**.

A positive value indicates a future timestep while a negative value indicates a past (or completed) timestep. ResSim will convert the offset from hours to timesteps by dividing by the length of the timestep (compute interval) and truncating any remainder.

For example, to get the value computed 3 hours earlier, enter a -3 in the offset field. NOTE: If the compute interval is 2 hours, a -3 offset will cause ResSim to return the value from the previous timestep (an offset of -1 *timesteps*).

Period—the period is the number of hours over which one of the **Period** functions computes a value. Enter a value in **hours**.

The period is always *back* in time relative to the time indicated by the offset. If the offset is zero or blank, the period is back in time relative to the *current* timestep. ResSim will convert the period from hours to timesteps by dividing by the length of the timestep (compute interval) and truncating any remainder.

When you have identified the independent variable and its Time Series Options, click **OK** to complete the definition of the Independent Variable of your release function and close the **Independent Variable Definition** Editor.

11.2.2.2 Select the Interpolation Type

The interpolation type indicates the type of curve that the data in the release function table describes. It also specifies the interpolation method to be used when performing a lookup from the release function table. The three **Interp** options are:

Linear—this option will cause ResSim to interpret the data in the table as a *linear* curve — where each pair of points (2 consecutive rows in the table) are connected by a straight line. And, linear interpolation is used when performing a lookup on the table to obtain a release. Figure 11.12 shows an example of a release function with the interpolation type set to **Linear**. The thumbnail plot shows how ResSim interprets the data as a linear curve (a series of straight line segments).

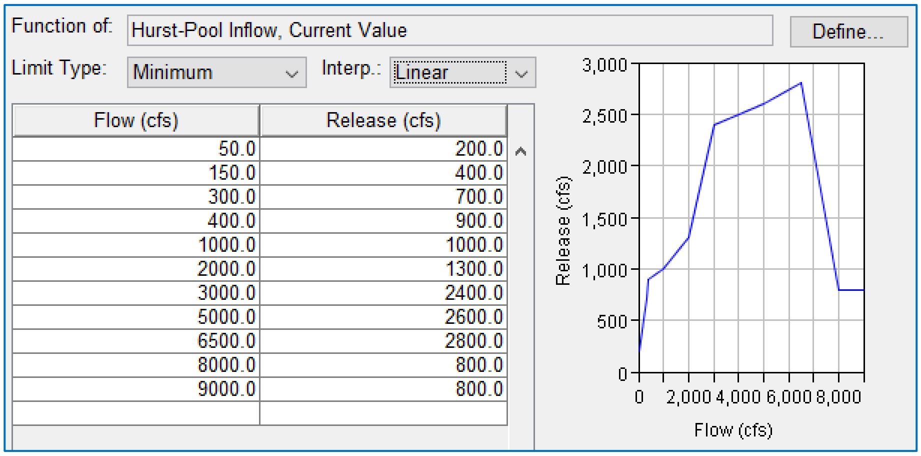


Figure 11.12 Example Release Function Rule using Linear Interpolation

Step—this option will cause ResSim to interpret the data in the release function table as a *step* function — where the release value in each row is held constant (drawn horizontal) until changed by the next row. *Step* interpolation is used to determine a release value when performing a lookup on the table—if the independent value falls between rows in the table, the value of the release (the dependent variable) is the release value in the row above. Figure 11.13 shows the thumbnail plot for the same function shown in Figure 11.12 but with the interpolation type set to **Step**.

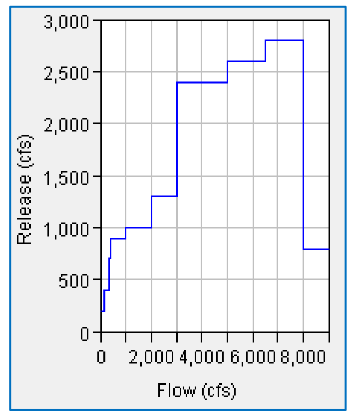


Figure 11.13 Step Interpolation

Cubic—this option will cause ResSim to interpret the data in the release function table as a *3-point Cubic Spline* curve. When a value for the independent variable is used to perform a lookup on the table to obtain an associated release (the dependent variable), the release is determined by the cubic spline function defined with each ***three*** consecutive rows in the table. Figure 11.14 shows the thumbnail plot for the same function shown in Figure 11.12 but with the interpolation type set to **Cubic**.

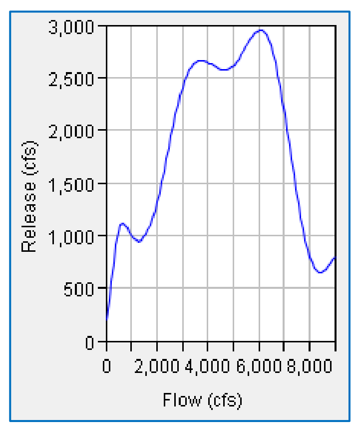


Figure 11.14 Cubic Interpolation

11.2.2.3 Seasonal Variation

If you selected a time series variable — **Model**, **External**, or **State Variable** — as your independent variable, a second independent variable, **Date**, can be selected to describe your release function. For example, if your release limit varies as a function of pool elevation but the lookup table has a column for growing season and non-growing season, then your first independent variable is the model variable Elevation and the second independent variable is date (or season).

To add Date as a second independent variable of your release function:

- Click the **Seasonal Variation** Edit... button located at the bottom of the list of *modifiers* to the right of the release function table.
- The **Seasonal Variation** editor (Figure 11.15) will open.

  Dates—Use the **Seasonal Variation** editor to define the seasons or date values to be entered into your release function table. The seasons or dates that you enter here define the columns of your function table (Figure 11.16).

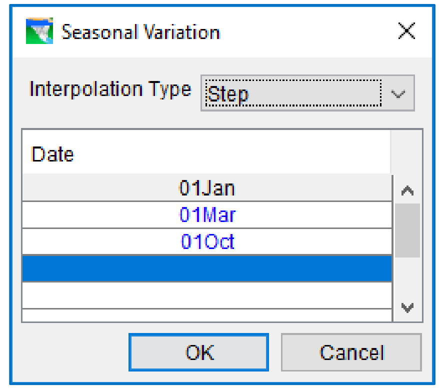


Figure 11.15 Seasonal Variation Editor

  Interpolation Type—this interpolation type selection applies to the columns of the release function table. The interpolation type on the main edit panel applies to the rows of the table (between the values list for the first independent variable.) Your options are: **Linear**, **Cubic**, or **Step**.

Flow (cfs)	Release (cfs)		
	01Jan	01Mar	01Oct

Figure 11.16 Release Function table for a Seasonally-Varying Function of Inflow Relationship

- When you have finished entering the dates that represent the start of each season, click **OK**. The **Seasonal Variation** editor will close, the **Seasonal Variation** checkbox will be checked ☑ Seasonal Variation Edit..., and the function table will have a column for each season start date you specified (Figure 11.16).

11.2.2.4 Filling in the Function Table

The release function is defined by the data you enter in the table (Figure 11.16) in the rule editor. The first column of the table represents your independent variable and the second column represents the desired release.

The data you provide should span the full range of values of the independent variable that could be used to lookup a desired release when the rule is evaluated in each timestep of a simulation. If, during the table lookup process, the value provided for the independent variable is outside the range of the data in the table, the returned release value will be from the end of the table whose independent variable value is closest to the provided value.

In addition, the data values entered into the table for the independent variable(s) must be monotonically increasing; this means that the value in the first (top-most) row in the table must be less than the value in the next row, and the value in the next row must be less than the value in the row below that, and so on, with no duplicate values.

Manual Data Entry

You can type values directly into cells in the table. Although you can use your mouse to move between cells in the table, the **Tab** key and the **Enter** key on your keyboard may make the process of data entry faster.

Tab—each click of the **Tab** key will move your cursor from one cell to the next cell in a row (left to right). And, when you reach the last cell of a row, the next **Tab** will move the cursor to the first cell of the next row.

Enter—each click of the **Enter** key will move the cursor to the next cell down in the current column. When you reach the last cell in a column, the next **Enter** will move the cursor to the first cell of the next column (to the right).

Neither the **Tab** nor the **Enter** keys will add rows to the table. The following options for adding rows to the table are available from the table's context menu:

Append Row—If you need to add a row to the bottom of the table, *right-click* on any cell in the table and select **Append Row** from the context menu of the table.

Insert Rows—to add multiple rows to the table, *right-click* on a cell in the table *above* which you want to add the new rows, then select **Insert Rows**. A dialog will open asking how many rows to insert. Enter the number of rows you want to insert *above* the selected cell and click **OK**.

Copy and Paste

If the data for your function is available in a spreadsheet application or any program that displays the data in a table, you can use the **copy and paste** functions to enter the data into the function table in ResSim.

To COPY:

- In the source program, select the range of data
- If available, select **Copy** from a menu of the editor or from the context menu of the table – OR – press Ctrl-C on your keyboard.

To PASTE:

- Select the first cell where you want the data placed in the function table in ResSim
- *Right-click* on the cell and select **Paste** – OR – press **CRTL-V** on your keyboard.

Note: When pasting a range of data into the function table, you do not need to select an equal range of cells in the table to paste the data into; you only need to select the first cell where you want the data placed. The paste function will overwrite preexisting data in the table. And, if the number of rows of data you have copied is greater than the number of rows available in the table *below the selected cell*, the paste function will expand the table (append rows) to accommodate the data.

Be careful when copying dates from a spreadsheet. Most spreadsheet applications use their own specialized numeric format for dates. Only 'text' dates in the form DDMMM or DDMmm can be *reliably* pasted into a date cell in ResSim.

The thumbnail plot to the right of the function table will reflect the values in the table. This is very useful to verify the data you are entering in the table; you can quickly see when a value entered is out of range or order with respect to the other values. To view the plot "full-size", *double-click* on the thumbnail plot. If it seems that the plot does not reflect the data you just pasted in, click the **Apply** button or *double-click* in a cell, re-enter in the data for that cell, then click in another cell. This action will cause the table to notice that it contains new data and the thumbnail plot should update accordingly.

11.2.3 Release Rule Modifiers

Several options are available to modify the value returned by your rule or to influence its applicability. These *rule modifiers* are located below the thumbnail plot, at the bottom right of the rule editor. The rule modifiers are:

Period Average Limit—to meet the flow limit over a period of time rather than timestep by timestep

Hour of Day Multiplier—modify the flow limit hour by hour

Day of Week Multiplier—modify the flow limit for each day of the week

Rising/Falling Condition—apply the rule only if the current conditions match the rising or falling condition specification.

11.2.3.1 Period Average Limit

ResSim treats most rules as an "instantaneous" limit on the release, which means that it tries to meet that limit every timestep. The **Period Average Limit** rule modifier allows you to specify a range of time, or *period,* over which the release limit determined from the release function applies; options for the averaging period include: **None**, **Daily**, and **Weekly**. Occasional violations of the rule's limit would be allowed as long as the average release over the period meets the rule's limit. The averaging period is *not* a sliding period or time window relative to the current timestep; instead, it is a standard calendar day (0000-2400) or week (7 sequential calendar days starting on a day of the week of your choosing).

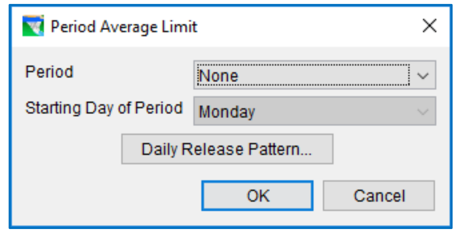


Figure 11.17 Period Average Limit Editor

The **Period Average Limit** rule modifier also allows you to specify a **Daily Release Pattern** for each day in the averaging period; options for the pattern type include **All Week**, **Weekdays and Weekend**, and **Each Day**. The **Daily Release Pattern** is defined by an hourly table, but the values specified in the pattern table are *weights*, not multipliers (like the **Hour of Day Multipliers**). ResSim will normalize the set of weights you specify in the daily pattern so that the *average* value of the set of weights for each day (**Daily** period) or for each week (**Weekly** period) equals 1.0. The *normalized* weights can be thought of as multipliers of the flow limit determined from the function table to produce the rule's desired limit for the current timestep. If the objective for a given timestep is not met, the weights for the remaining timesteps of the period will be scaled so that the overall objective can still be met over the period.

To make your release function rule a **Period Average Limit** rather than an instantaneous limit:

- *Click* the **Edit...** button to the right of the **Period Average Limit** label.
- The **Period Average Limit** editor will open (Figure 11.17). Specify each of the attributes needed to define the **Period Average Limit**:

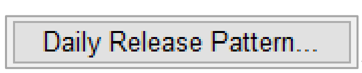
Period—Select the averaging **Period** from the selector. The options for the averaging **Period** are:

None—indicates that the **Period Average** option is *not* active.

Daily—for timesteps less than 1DAY, ResSim will attempt to make the average flow over each 24-hour period (0000 – 2400) equal to the given rule value. Not useful with a 1DAY timestep.

Weekly—the averaging period is 1 calendar week, starting on the **Starting Day of Period**. The **Weekly** period is useful with longer timesteps, especially 1DAY.

Starting Day of Period—use this selector to specify the first day of the Week for a **Weekly** averaging period.



—use the **Daily Release Pattern** button to open the **Period Average Requirement Pattern** editor (Figure 11.18). The purpose of this editor is to allow you to specify the daily pattern and hourly weights to be used for the averaging period; you can even specify a *set* of daily patterns for seasonally varying demand schedules.

Seasonal Variation—If the daily release pattern changes with the time of year, click the **Seasonal Variation Edit** to open the **Seasonal Variation** editor and define the seasons. Section 11.2.2.3 describes how to use the **Seasonal Variation** editor.

Specify Pattern for—select the pattern type you want to use. Your options are **All Week**, **Weekday and Weekend**, and **Each Day** as described above. Three types of **Daily Release Patterns** are available:

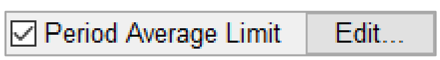
All Week—applies the same pattern for every day of the week. The pattern table provides a row for each hour of the day.

Weekdays and Weekend—allows you to specify a different release pattern for weekdays (Monday through Friday) than for weekend days (Saturday and Sunday).

Each Day—allows you to specify different release requirement patterns for each day of the week.

Figure 11.18 illustrates the pattern tables for each of the three pattern types.

- Click **OK** to save your settings and close the **Period Average Limit** editor.



Back in the rule editor, if you selected a **Period** of **Daily** or **Weekly**, the check box in front of the **Period Average Limit** will be checked indicating that this option is "ON". The next two modifiers—**Hour of Day Multipliers** and **Day of Week Multipliers**—are not available when the Period Average Limit option is active.

All Week Weekdays and Weekends Each Day

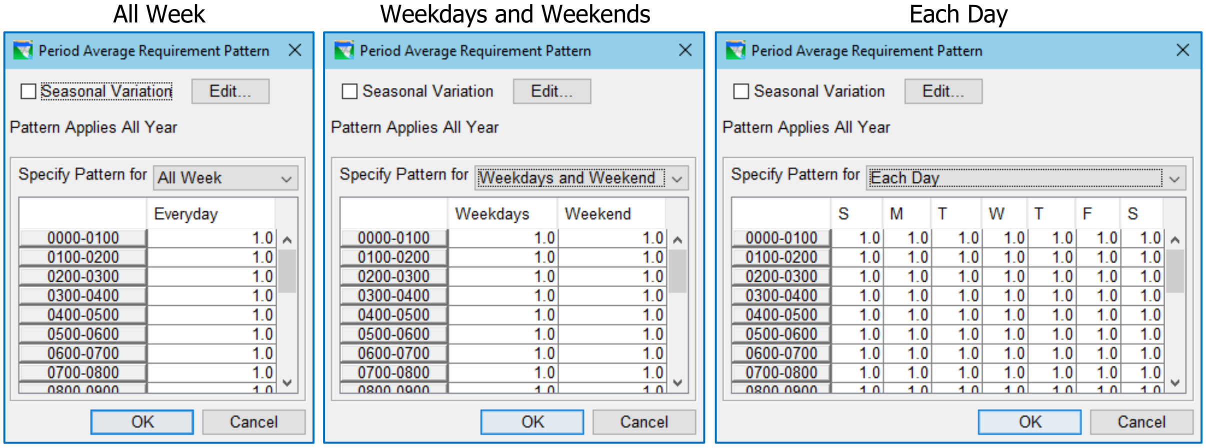


Figure 11.18 Period Average Limit—Daily Release Patterns

11.2.3.2 Hour of Day Multiplier

The **Hour of Day Multiplier** allows you to specify a table of factors (based on the *time of day*) that will be applied to the desired release returned from the release function.

Use the **Hour of Day Multipliers** to represent an operating constraint that requires a minimum (or maximum) flow during only a portion of the day.

For example, to specify a rule that would only apply from 8:00 a.m. to 5:00 p.m. In addition, the midday hours need twice as much flow than the early morning and late afternoon hours. To reflect these requirements, set the multipliers to:

1—for 0800, 0900, 1500, and 1600

2—for 1000, 1100, 1200, 1300, 1400

0—for all other hours

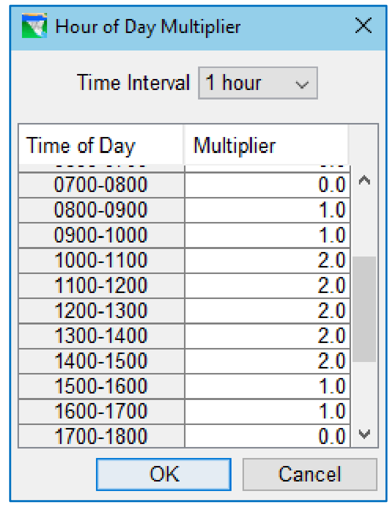


Figure 11.19 Example Hour of Day Multipliers

Figure 11.19 illustrates these settings in the **Hour of Day Multiplier** editor.

To define a set of **Hour of Day Multipliers**:

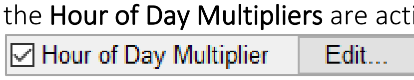
- *Click* the **Edit...** button to the right of the **Hour of Day Multiplier** label.
- The **Hour of Day Multiplier** editor will open. Specify the attributes needed to define the Hour of Day Multipliers:

  Time Interval—At the top of the editor is a selector for the **Time Interval**. Figure 11.20 shows the **Hour of Day Multiplier** editor with the **Time Interval** selector open to display its list of options.

  Multipliers—The rest of the editor is a table for you to specify the multipliers for each interval of the day. A multiplier of 1.0 has no impact on the value of the desired release so the table will initially be filled with 1.0.

- Click **OK** to save your settings and close the **Hour of Day Multiplier** editor.

  Back in the rule editor, if the table of multipliers you just saved has any cell whose value is not 1.0, the check box in front of the **Hour of Day Multiplier** label will display a check mark indicating that the **Hour of Day Multipliers** are active:



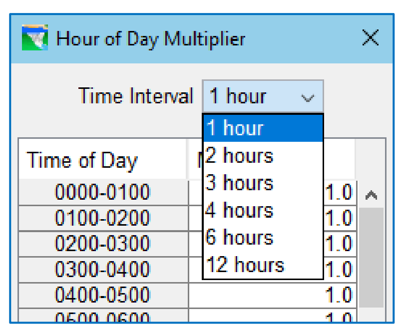


Figure 11.20 Hour of Day Multiplier Editor—Time Interval Options

11.2.3.3 Day of Week Multiplier

The Day of Week Multiplier allows you to specify a factor (based on the *day of the week*) that will be applied to the desired release determined from the release function. To define a set of Day of Week multipliers:

- Click on **Day of Week Multiplier Edit** button
- The **Day of Week Multiplier** editor (Figure 11.21) will open. This editor contains a table with a row for each day of the week. A multiplier of 1.0 has no impact on the value of the desired release so the table will initially be filled with 1.0.
- Use the **Day of Week Multipliers** when representing an operating constraint that only applies to certain days of the week or that varies by day of week.

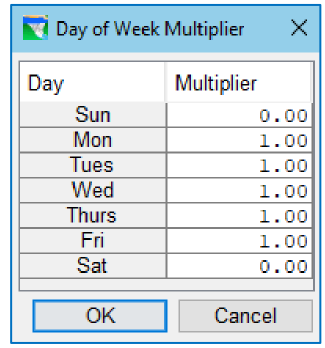
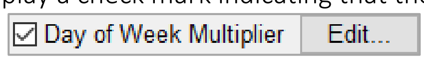


Figure 11.21 Example Set of Day of Week Mulitpliers

Figure 11.21 illustrates the following example:
for a minimum flow that is only required on weekdays but not weekends, set the multiplier for Monday–Friday to 1.0 and set Saturday and Sunday to 0.0.

- Click **OK** to close the **Day of Week Multiplier** editor.

Back in the rule editor, if the table of multipliers you just saved has any cell whose value is not 1.0, the check box in front of the **Day of Week Multiplier** will display a check mark indicating that the **Day of Week Multipliers** are active: 

11.2.3.4 Rising / Falling Condition

The **Rising/Falling Condition** rule modifier can be used to restrict the applicability of a rule (i.e., to turn the rule on or off as a function of the specified condition). The condition is evaluated every timestep. If the condition evaluates to TRUE, then ResSim includes the rule in the rule stack, evaluates it, and applies it within the release decision logic. However, if the condition evaluates to FALSE, ResSim ignores the rule (does not include the rule in the rule stack, does not evaluate the rule, and does not apply the rule in the release decision logic).

The **Rising/Falling Condition** rule modifier allows you to select one of eight conditions with which you can restrict the applicability of the rule. These conditions include:

1. rising pool elevation
2. rising or constant pool elevation
3. falling pool elevation
4. falling or constant pool elevation
5. rising inflow
6. rising or constant inflow
7. falling inflow
8. falling or constant inflow.

To apply a rising/falling condition to a rule:

- Click the **Rising/Falling Condition Edit** button.
- The **Rising/Falling Condition** dialog will open (Figure 11.22). A **Rising/Falling Condition** is defined by the following attributes. You must select or enter a value for each attribute:

  Condition—choose from:

  - Rising
  - Rising or Constant
  - Falling or Constant
  - Falling

  Parameter—choose from:

  - Pool Elevation
  - Inflow

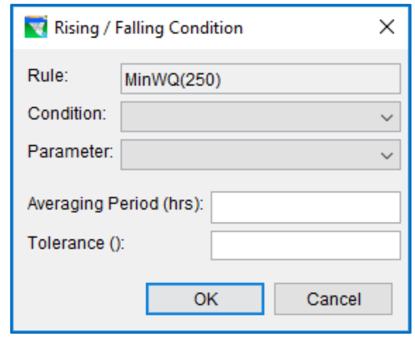


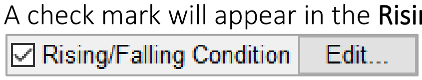
Figure 11.22 Rising/Falling Condition

  Averaging Period—this is a range of time define in hours *back* from the current timestep. The period is used, along with **Tolerance**, to determine if the **Condition** is met. Think of the averaging period as time span over which the *slope* of the **Parameter** is computed.

  Tolerance—The tolerance is used for the determination of the "constant" portion of the condition. If the difference in the **Parameter** over the Averaging Period is less than the **Tolerance**, the condition is considered *constant* — neither rising nor falling.

- Click **OK** to save your settings and close the **Rising/Falling Conditions** dialog.

  A check mark will appear in the **Rising/Falling Conditions** checkbox:



When you have finished entering data for your rule, be sure to click **Apply** before moving on to the next rule.

An example of a completed **Release Function** rule is shown in Figure 11.23.

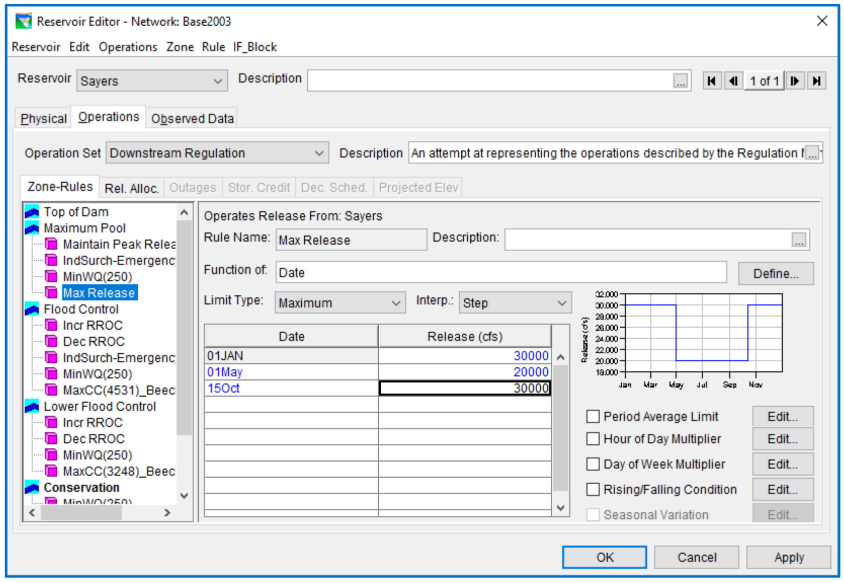


Figure 11.23 Example of Completed Release Function Rule

11.3 The Downstream Control Function Rule

The **Downstream Control Function** rule describes a minimum or maximum ***flow*** or ***stage*** limit (e.g., flow requirement, channel capacity, flood stage) *at a* ***control point*** rather than an explicit limit on the reservoir ***release***. However, like all ResSim rules, when the rule is evaluated for a given timestep, it must produce a desired *release* limit. The desired release limit that the downstream control rule calculates will be determined based on the downstream objective, the influence of routing to the control point, previous and future releases, and the cumulative local flows at the control point.

A Downstream Control rule can be used to implement or impose an implicit *system operation.* System operation occurs when two or more parallel reservoirs are operated together to meet a common objective, in this case—the downstream constraint. Once a Downstream Control rule is created at one reservoir, it will be included in the list of *Existing* Rules at each reservoir in the network *that is upstream of the control point*. All it takes to establish an implicit system operation is to use the same downstream control rule in each applicable reservoir's operation set. Chapter 13 discusses ResSim's methodology for defining and managing system operation.

Defining a **Downstream Control Function** rule is very similar to defining a **Release Function** rule. In fact, the **Downstream Control Function** rule editor is an extension of the **Release Function** rule editor described above, so this section will focus on describing how to define the three attributes that are unique to the **Downstream Control Function** rule — **Parameter**, **Flow Contingency**, and **Advanced Options**.

Like all ResSim rules, a **Downstream Control Function** rule must be assigned to a release element when the rule is created. However, a **Downstream Control Function** rule can only be assigned to the reservoir itself because the rule evaluation logic must account for all releases from the reservoir, *including releases through a diverted outlet*, that could influence the flow or stage at the downstream control point.

To *create* a **Downstream Control Function** rule:

- *Right-click* on the zone in the **Zone-Rules Tree** to which you want to add the new rule.
- Select **Add New Rule...** from the zone's context menu. The **New Operating Rule** dialog will open.
- Select the *reservoir* from the **Operates Release From:** selector. It will be the first entry in the list.
- Select **Downstream Control Function** from the **Rule Type** selector. This selection will cause a new field — **Downstream Location** — to appear in the **New Operating Rule** dialog (Figure 11.24).
- Select a *junction* from the **Downstream Location** selector that represents the control point of interest. The list of available locations will include all the reaches and junctions in the current network that are downstream of the current reservoir.

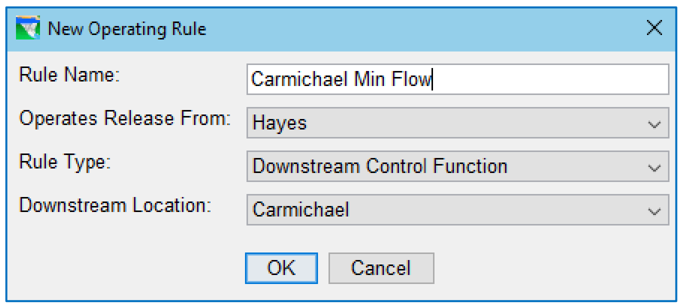


Figure 11.24 Reservoir Editor—New Operating Rule—Downstream Control Function

Reaches are included for completeness, but there are very few situations in which a reach is the appropriate selection for a downstream control point.

> Note: When you see a grey box drawn around a junction in the Map Display area of ResSim, this is a *visual cue* that the junction is a *control point* for a downstream control rule.

- Give the rule a name and click **OK** to complete the creation of the new rule and close the **New Operating Rule** dialog.

As illustrated in (Figure 11.25), your new rule will appear at the bottom of the rule stack of the current zone in the **Zone-Rules** tree and the Downstream Control Function rule editor (in the edit panel) will show the name of your rule and the **Downstream Location** you selected when you created the rule.

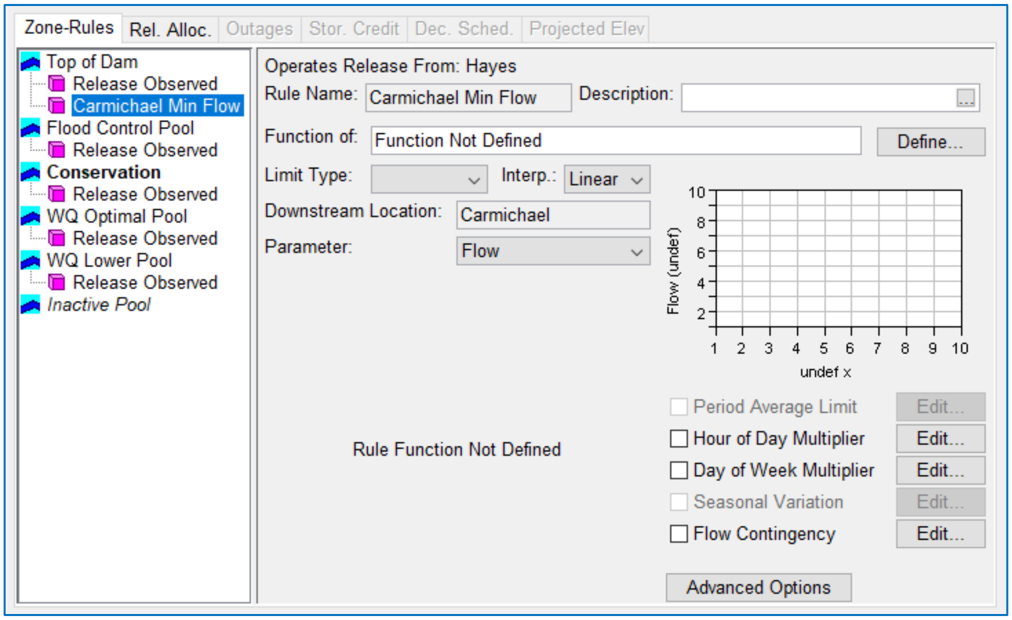


Figure 11.25 Reservoir Editor—Downstream Control Function Rule

Refer to the Release Function Rule section above for descriptions of the features and parameters that are common to both the Release Function rule and the Downstream Control Function rule. The features and parameters that are unique to the **Downstream Control Function** rule editor are described in the following sections.

11.3.1 Parameter

The **Parameter** is the objective variable of the rule; as such it is the *dependent* variable of the downstream control rule's function table. Your **Parameter** options are **Flow** and **Stage**.

Since the downstream control logic in ResSim operates for a Flow target, if you choose Stage as the Parameter for your downstream control function, you must provide a *rating curve* that relates flow to stage at the junction you identified as the control point so that ResSim can, in each timestep, lookup the value of the flow limit represented by your rule's stage limit value.

Although a *Rating Function* (stage as a function of two variables) can now be specified in ResSim, if the rating function at your control point reflects a backwater influence, the downstream operation may not be able to produce a valid estimate of the flow limit at the control point during the rule evaluation in each timestep. For this reason, a *Simple Rating* is *recommended* at the control point when operating for Stage.

11.3.2 Limit Type

For a Downstream Control Function rule, your options for the limit type are **Minimum** and **Maximum**. **Specified** is ***not*** an available limit type for a Downstream Control Function rule.

11.3.3 Flow Contingency Factor

The **Downstream Control Function** rule editor offers a *Rule Modifier* that is not available to a **Release Function** rule— a **Flow Contingency** Factor. The purpose for this factor is to enable you to apply an *uncertainty* factor to the cumulative local flow so that the downstream control objective can still be met even when the quantity of the local flow is most uncertain.

If the limit type of the downstream control rule is ***Maximum***, ResSim will use the contingency factor to *increase* the cumulative local flow at the control point which effectively reduces the available space at the control point. The reduced available space will cause the reservoir to release less than it would have without the contingency factor. This would make the operation appear more "conservative".

If the rule's limit type is **Minimum**, the contingency factor will be used to *decrease* the cumulative local flow, effectively increasing the available space at the control point and causing the reservoir to release more than it would have without the contingency factor, thus reducing the risk of not meeting the objective.

To apply a contingency to the cumulative local flows at the **Downstream Location**:

- Click the **Flow Contingency** Edit... button to open the **Flow Contingency for Downstream Operation** dialog (Figure 11.26).

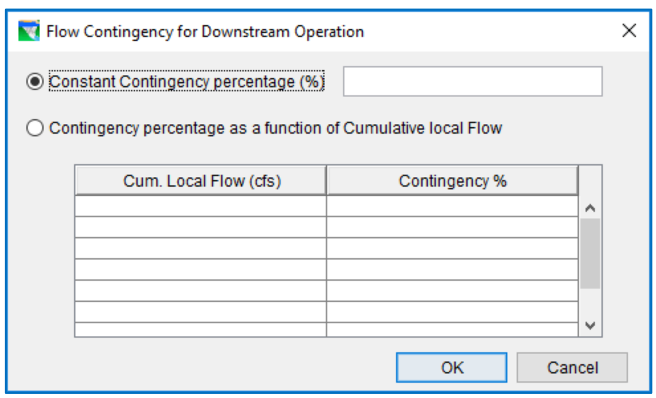


Figure 11.26 Flow Contingency for Downstream Operation

- Two options are available for specifying the **Flow Contingency** factor:

  Constant Contingency percentage—select this radio button if you want the local flow to be increased by a constant percentage, then enter a value in the textbox. For example, if you want to adjust the cumulative local flow by 20%, enter 20 in the textbox.

  Contingency percentage as a function of Cumulative local flow—select this ratio button if you want the contingency factor to vary with the magnitude of flow. With this option, you must enter a relationship between the computed cumulative local flow at the control point and the contingency factor to be applied to it in the downstream control computations. For example, if the estimate of local flow is more uncertain under high flow conditions than under low, base flow conditions, you could enter a relationship where the contingency factor is zero or very small for smaller flow values and increases with larger flow values.

- Click **OK** to save your settings and close the **Flow Contingency**… editor.

  If you specified a flow contingency factor or relationship, the **Flow Contingency** checkbox will be checked: ☑ Flow Contingency Edit... in the **Downstream Control Function** rule editor.

NOTE — the current implementation of the Flow Contingency Factor is not quite what has been described here. Testing has revealed that the Contingency Factor is currently being applied to the *limit* of the rule rather than to the cumulative local flow at the control point.
Look for this to be corrected in a future version of ResSim. If your model uses this feature, the correction will cause the newer ResSim to produce different results than the current version does.

11.3.4 Advanced Options

In addition to local inflows, the downstream control logic is affected by two other external factors — *routing* and *rate of change constraints*. Additional logic, referred to as *Advanced Options*, has been added to ResSim's downstream control algorithm to enable you to make adjustments that may help ResSim respond better to these external factors or minimize their influence on the downstream control operation. Specifically, the A*dvanced Options* relate to how routing, attenuation, and rate of change constraints are considered during the operation for downstream control rules. The four Advanced Options are named:

1. **Correct for Attenuation**
2. **Consider Rate of Change Constraints**
3. **Routing Time Window**
4. **Pulse Flows**

The parameters that control the first three of the Advanced Options are available in the downstream control rule's **Advanced Options** editor and can be specified (as a set) to apply to all downstream rules or to just the current rule. The fourth Advanced Option, **Pulse Flows**, is described the next section.

To edit the Advanced Options:

- Click the **Advanced Options** button Advanced Options located at the bottom right of the **Downstream Control Function** rule editor. The **Advanced Options** editor will open (Figure 11.27).

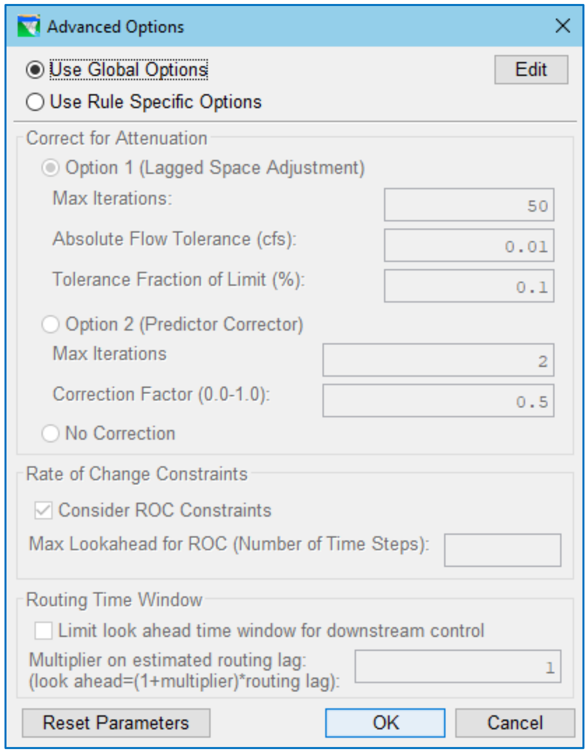


Figure 11.27 Advanced Options for Downstream Control

- Specify whether this rule should use the **Global** settings for the Advanced Options or if it should use its own, **Rule Specific**, set of Advanced Options settings. Make this choice by selecting one of the two radio buttons located at the top of the **Advanced Options** editor. Your choices are:

  Use Global Options—this is the default for all downstream control rules unless you select the other option in the rules that should use a different set of settings.

  To view and/or modify the *global* options, *click* the **Edit** button to the right of the **Use Global Options** radio button. The **Global Downstream Options** editor will open (Figure 11.28). You may view or edit the *global* Advanced Options settings whether or not the Use Global Options radio button is selected for the current rule.

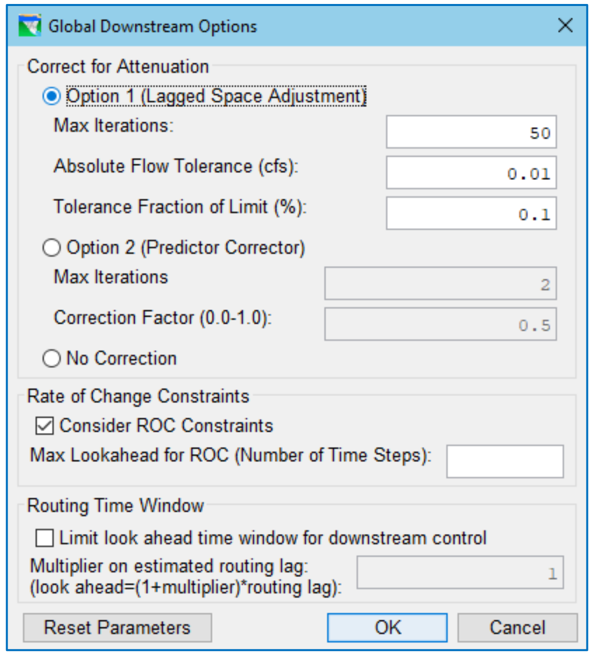


Figure 11.28 Global Downstream Options

Use Rule Specific Options—choose this radio button if you want to specify a set of Advanced Options setting that will apply *only* to the *current* downstream rule. If you choose this option, the panel below the radio buttons will become active, allowing to you modify the available options.

Whether you are editing the **Global** settings or the **Rule Specific** settings, the available Advanced Options and their parameters are:

Correct for Attenuation—Choose one of the three radio buttons in this section of the editor (Figure 11.29) to identify which method the downstream control logic will use to adjust its desired release limit to account for the effects of attenuation. Your choices include:

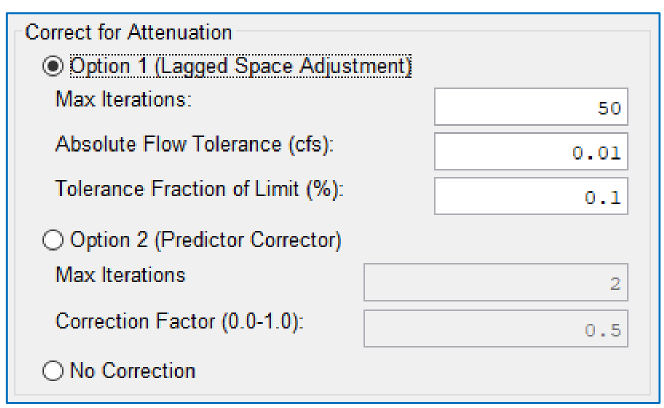


Figure 11.29 Advanced Downstream Options—Methods to Correct for Attenuation

Option 1 (Lagged Space Adjustment)—if you choose this option (the default), you can enter Max Iterations, an Absolute Flow Tolerance (cfs or cms), and a Tolerance Fraction of Limit (%).

However, before making changes to these parameters, see how well the default values work for your model. This method *seems* to work well with non-linear routing.

Option 2 (Predictor Corrector)—enter Max Iterations and a Correction Factor (from zero to one). This method *seems* to work well with linear routing.

No Correction—select this option when you do not want any adjustments made based on the routing attenuation parameters. This option is usually the fastest, but it is applicable only if your local inflows change gradually or do not oscillate from high to low values two or more times within the routing window.

Rate of Change Constraints—By default, ResSim considers rate of change limits when determining the routing time window for each downstream control rule. This means that ResSim estimates the number of timesteps it needs to make adjustments to its releases so that both the rate of change is satisfied and the downstream objective is not violated and adds that estimate to the routing window to be used by the downstream control logic.

Consider ROC Constraints—In this section, you can turn OFF the influence of rate of change constraints on the downstream operation by *unchecking* this checkbox (Figure 11.30).

Max Lookahead for ROC (Number of Time Steps)—If **Consider ROC Constraints** is ON, you can use this field to specify the maximum number of timesteps that can be added to the routing time window to account for rate of change constraints. This is a way to limit how much ResSim can extend the routing window to accommodate rate of change constraints.

Blank—When **Consider ROC Constraints** is ON (checked) and the **Max Lookahead...** field is blank (the default), no limit is placed on how many timesteps can be added to the routing time window to account for rate of change constraints. Unfortunately, ResSim's estimate of the number of timesteps needed may be excessive; especially if the model has one or more *variable* rate of change constraints.

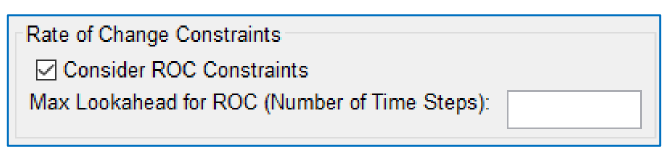


Figure 11.30 Advanced Downstream Options—Rate of Change Constraints

Routing Time Window—for each downstream control rule, ResSim computes the number of timesteps into the future over which a release made in the current timestep will influence (be part of) the flow at the control point.

The length of this time window is a function of the routing between the reservoir and the control point. It is determined at the start of the compute process, before the input data has even been loaded. The next section, titled **Pulse Flow Options**, includes a description of how ResSim computes the routing window and about how you can help ResSim improve those computations.

Limit look ahead time window for downstream control—If you active this checkbox, ResSim will *replace* the computed routing time window with a value equal to the *lag time* multiplied by a factor (Figure 11.31). The factor that ResSim will use is [1 + ***Multiplier***].

Use this option if your model is using non-linear routing and the routing time window that ResSim computed is too long, causing the downstream control operation to appear to "not be working right".

To activate and use this option:

- *Click* in the **Limit look ahead time...** checkbox to add (or remove) the checkmark. When a checkmark appears in the checkbox, the option is ON or active.
- Enter a **Multiplier** value.

Although *Multiplier* is not a great name for this parameter, the value you enter will be added to 1.0 to produce a *factor*. The lag time will be *multiplied* by this factor to produce a new value (or limit) for the Routing Time Window.

Be sure to carefully consider the value you enter for the *multiplier*. If you do not enter a value, the default multiplier value is 1 which results in a factor of 2. Thus, the new routing window = 2 x lag time.

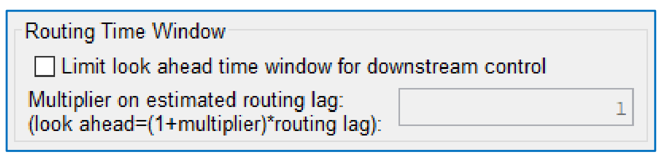
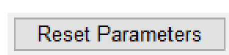


Figure 11.31 Advanced Downstream Options—Limit the Routing Time Window

Reset Parameters—this button 

will clear your settings and restore the settings and their parameters back to their original, default values.

Note: Since these Advanced Options are complex, "trial and error" runs might be needed to test their impact on downstream operations.

11.3.5 Pulse Flow Options

The purpose of the **Pulse Flow Options** is to allow you to specify a new value for the **Default Pulse Flow** and/or to specify a reservoir-specific **Pulse Flow** value for one or more of the reservoirs in your network. ResSim uses the Pulse Flow value in its algorithm for determining the routing window that will be used in the downstream control operations logic.

In order to identify when the value of the pulse flow needs to be changed and what value should be used, you need to understand how ResSim estimates the routing time window.

The Routing Time Window

When downstream control (DSC) rules are active in an alternative, ResSim adds a special compute section, labeled **Evaluating Routing for Downstream Operations**, to the beginning of the compute of the alternative. During this compute section, ResSim routes a "pulse" of water from each reservoir (with downstream control rules) to that reservoir's downstream control points. It then compares the hydrograph of the pulse as it left the reservoir to the routed version of the pulse at each control point in order to determine the peak to peak "lag time" and the overall spread of the pulse at the control point. The routing time window for each control point starts with the timestep when the pulse left the reservoir and ends at the end of the "spread". The routing time window computed by ResSim for each active downstream control rule will be reported in the compute log at the start of the **Computing Regulated Flow** section—look for the messages that start with "**Pulse Width Steps =**". An example of these messages is shown in Figure 11.32.

```
Pulse Width Steps = 5 for analysis of Downstream Rule Min@Carmichael

Pulse Width Steps = 5 for analysis of Downstream Rule Max@Carmichael

Pulse Width Steps = 6 for analysis of Downstream Rule Min@Smithford

Loading Input Timeseries
C:/Current Projects/ExampleWatersheds/Hayes_Basin/rss/Oct_99_Event_-_More_(
```

Figure 11.32 Example Compute Log Messages—Computed Window Size for each DSC Rule

To view the routed hydrographs of the pulse flow produced by your model:

- Open DSSVue from the **Tools** menu of your simulation
- Filter the DSS pathnames for a C-part of **Flow-Routing-*YourReservoir*-1** where *YourReservoir* is the name of the reservoir you are interested in.
- Select the dataset generated at the outflow junction of the reservoir *and* the dataset generated at the downstream-most control point. Then,
- Plot the selected datasets.

An example plot of the two suggested pulse flow hydrographs is shown in Figure 11.33. Did you notice that the pulse illustrated above is actually an *inverted* pulse? Routing an inverted pulse of water is numerically more stable for the hydrologic routing methods used by ResSim. It is also more consistent with how reservoir operations actually work; reservoirs rarely send out pulses of water, instead they often maintain a relatively constant release then cut back when necessary.

If you look carefully at the hydrograph leaving the reservoir, you'll see that ResSim is routing a ***constant flow*** and then cutting that flow in half for one timestep before returning the flow to the original constant value. The inverted *pulse* is the one-timestep cutback. Although it is a bit of a misnomer, the **Pulse Flow Options** editor labels the ***constant*** flow value **Pulse Flow**. The default value for the **Pulse Flow** is 5000 cfs (142 cms).

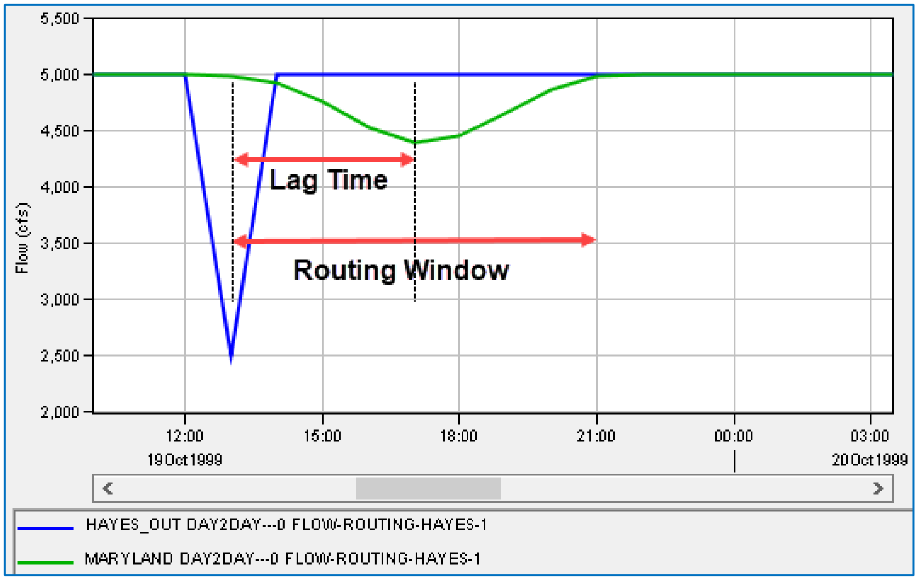


Figure 11.33 Plot of Pulse Flow and Routed Pulse Flow

The **Pulse Flow Options** is the only other option currently available in ResSim for influencing the estimate of the *Routing Time Window* used by the downstream control operation logic. However, this option is not located in the **Downstream Control Function** rule editor. Instead, it is located in the **Dam** menu or the **Dam**'s context menu (Figure 11.34) on the **Physical** tab of the **Reservoir Editor**.

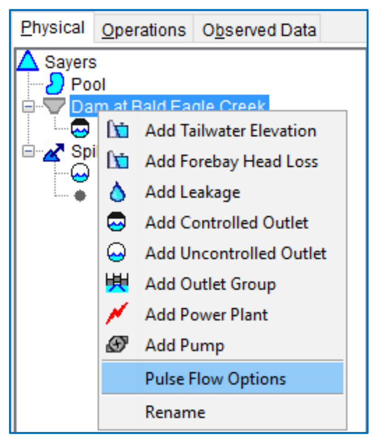


Figure 11.34 Dam Context Menu—Pulse Flow Options

The **Pulse Flow Options** is the only option, other than those included in the downstream control rule's **Advance Options**, that is currently available in ResSim for influencing the estimate of the *Routing Time Window* used by the downstream control logic. However, this option is not located in the **Downstream Control Function** rule editor. Instead, it is located in the **Dam** menu or the **Dam**'s context menu on the **Physical** tab of the **Reservoir Editor** ():

To edit the **Pulse Flow** value(s):

- Open the **Reservoir Editor** and *click* on the **Physical** Tab.

- *Right-click* on the **Dam** in the **Reservoir Tree**, then select **Pulse Flow Options** from the context menu. The **Pulse Routing Options** editor will open: (Figure 11.35).

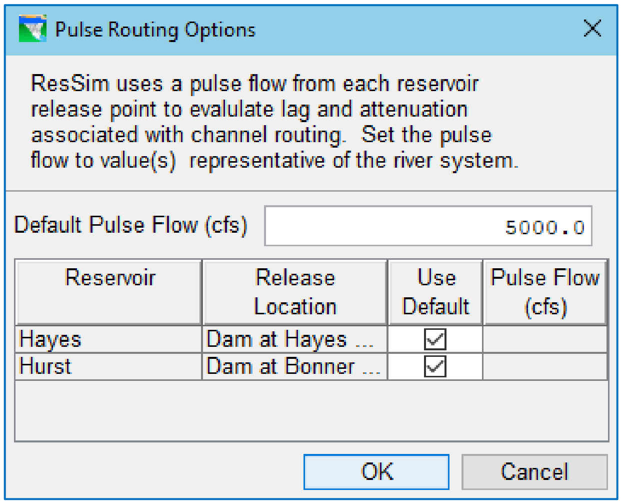


Figure 11.35 Pulse Routing Options Editor

- Enter a new *Pulse Flow* value in the **Default Pulse Flow** textbox.

  Remember, the value you are entering as the *Pulse Flow* is actually the constant flow of the pulse flow hydrograph and the magnitude of the inverted pulse is half of that. As a rule of thumb, use a pulse flow value that is 1.5—2 times larger than the average value of the limit that the reservoir should usually be operating for.

- And/or *uncheck* the **Use Default** box for each reservoir for which you want to specify a new **Pulse Flow** value, then enter the new **Pulse Flow** value in the last column of the table.

If your model is using a linear routing method in your reaches, then there should be no need to change the Pulse Flow value(s) from the default since the routing window will not change as a function of flow. However, if you are using a non-linear routing method, then determining an appropriate value for the pulse flow can be a challenge, especially if you are operating for both a minimum and a maximum limit at the control point. Decide which limit is the most important, then, as a rule of thumb, set the pulse flow to 1.5—2 times the average value of the downstream limit defined at the control point. See how that value affects the routing window and the downstream operation. If necessary, adjust from there. It won't take long to zero-in on an appropriate value.

- When you have finished setting the **Pulse Flow** value(s), *click* **OK** to save your setting and close the **Pulse Flow Options** editor.
- Be sure to click **Apply** in the **Reservoir Editor** to save your changes to the reservoir data in the network before proceeding to another tab or reservoir.

11.4 The Induced Surcharge Rule

In the context of watershed hydrology, the word *surcharge* refers to the *extra* storage that naturally occurs behind an *uncontrolled* spillway when the pool elevation exceeds the spillway's crest (and inflow exceeds outflow from the reservoir). *Thus, surcharge storage* is the volume of water in the reservoir above the spillway crest elevation. This concept is illustrated in Figure 11.36.

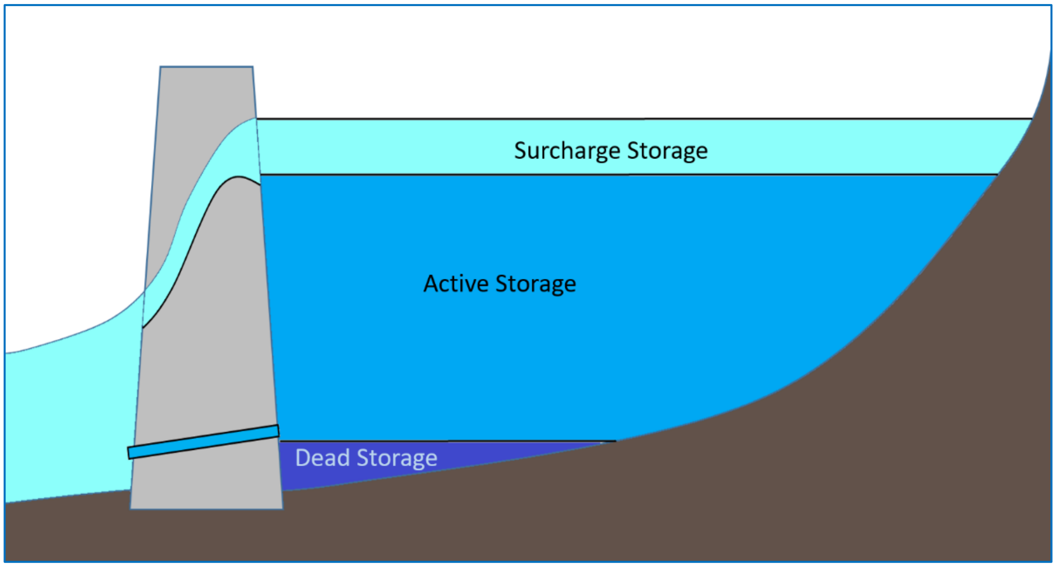


Figure 11.36 Surcharge Storage

Induced surcharge is the extra storage that occurs behind a *controlled* spillway when the spillway gates are at least partially open. The adjective *induced* is used to indicate that the extra storage was *caused* by man's actions and not a natural occurrence. So, since induced surcharge is *caused* by opening the gates, then *induced surcharge storage* is the volume of water in the reservoir above the elevation of the top of the *closed* gates. As the gates are opened (together), the elevation of the top of the gates rises providing more room for water to be held behind the dam (as long as inflow exceeds outflow). Figure 11.37 below shows both the minimum and the maximum positions of the gates on a spillway, closed and fully open, and the maximum size of the induced surcharge pool.

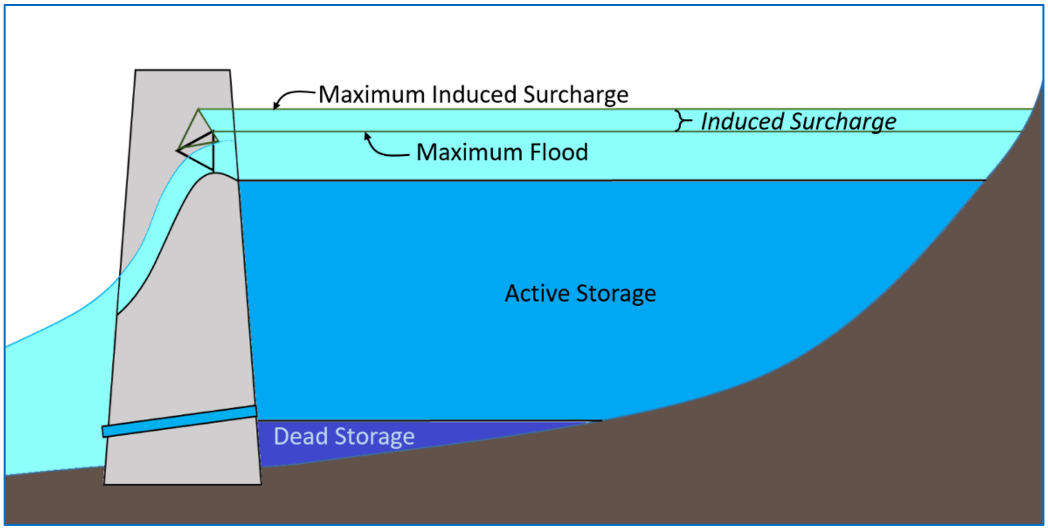


Figure 11.37 Induced Surcharge Storage

Induced surcharge *operation* is the process of adjusting the position of the spillway gates for the specific purpose of inducing surcharge. When the gate opening is set to limit the spillway flow to less than free overflow, water is intentionally surcharged—or stored—behind the gates.

To water managers (reservoir regulators), *Induced Surcharge Operation* is a flood risk reduction operation that calls for larger releases (than would be made under normal flood operations) when the current pool elevation together with a rapidly rising inflow (or pool elevation) threaten dam stability. As such, it could be considered a "save the dam" operation. Induced surcharge operation allows operators and regulators to manage an extreme flood event by utilizing the additional volume above the top of the (closed) gates.

Although induced surcharge operations are predicated on the assumption that the reservoir has a gated spillway, this is not necessarily a requirement. The actual requirements are (a reserved quantity of storage at the top of the usable reservoir pool and enough release capacity to pass the peak inflow of the PMF or reservoir design storm. For example, a reservoir may be constructed without a gated spillway but with one or more lower level outlets have far more release capacity than would ever be utilized under normal conditions. If a portion of storage at the top of the pool of this reservoir is identified for use only under extreme inflow conditions, an operating plan could be developed that, when needed, follows the induced surcharge operation described above to utilize the reserved reservoir storage and the full release capacity of the reservoir.

The induced surcharge operation in ResSim is a special operation whose primary objective is to "*protect the dam without making the downstream flooding worse than if the dam were never there*". To do this, ResSim will utilize the last remaining "safe" storage in the reservoir to store the peak of the incoming flood event while passing the rest of the inflow through the reservoir. The assumption is that by holding back the peak of the inflow, the downstream peak will be reduced. In addition, during the rising limb of the inflow hydrograph, the Induced Surcharge rule will not call for releases that are greater than inflow; thus, this operation won't make the flood worse than it would have been if the reservoir weren't there.

The induced surcharge, or emergency gate regulation, operation described in the water control manual for most USACE reservoirs usually specifies a release that ***must be made***. In general, a release that ***must be made*** can usually be interpreted as a *minimum* release requirement. However, under the conditions that call for induced surcharge operation, the release requirement is usually interpreted as a *specified* flow, i.e., no more and no less. Since specified flow rules can be more heavy-handed than necessary, the **Induced Surcharge** rule in ResSim was designed to produce a desired minimum release limit. However, it was *not* designed to stand alone, it needs a partner.

To model induced surcharge operation in HEC-ResSim, you MUST specify *TWO* rules:

1) An **Induced Surcharge** rule placed at the ***top*** of the rule stack in each operating zone from which the induced surcharge operation could be triggered. Remember, the trigger is a combination of pool elevation *and* rapidly rising inflow or elevation so the rule *could* potentially be activated even if the pool is relatively low but inflow is high and rising fast.

2) An accompanying **maximum** limit rule (**Release** or **Downstream Control Function**) placed ***below*** the **Induced Surcharge** rule in the rule stack of each zone that includes the Induced Surcharge rule. This maximum limit rule should correspond to the maximum discharge that would occur if surcharge operations were not in effect; as such, the maximum limit rule(s) you select to partner with the induced surcharge rule may vary per zone.

It is important to understand *why* the two rules listed above are necessary to model induced surcharge operation. If you don't understand why, you will find it difficult to build an operation set that can smoothly transition from normal flood control operation to induced surcharge and back again.

First, the **Induced Surcharge** rule is a **minimum limit** release rule. ***And***, it should be placed as the ***highest priority*** rule so that, when it determines that the dam is at risk, *it can force higher releases than normal flood control operations would allow*.

The **maximum** limit rule is the rule (or set of rules) that describe *normal flood operation.* The maximum limit rule must be placed at a ***lower priority*** so that it can prevent the reservoir from releasing more than it should while still allowing the higher priority Induced Surcharge rule to override the more restrictive normal flood control operation when necessary.

When the **Induced Surcharge** rule becomes active, the two rules function together as a *specified limit* rule. To explain this, let's see how the two rules will affect the *allowable release range*:

- For simplicity, assume the rule stack has only two rules, an **Induced Surcharge** rule followed by a **Maximum Release** rule representing channel capacity.
- When the **Induced Surcharge** rule becomes active, *it will produce a desired release that is usually **equal to or greater than** the limit of the **Maximum Release** rule*.

- The desired **Induced Surcharge** (minimum) release is then applied to the allowable range. If the desired release is greater than the current minimum limit of the allowable range, the minimum limit will be assigned the desired release value.
- Next, the lower-priority **Maximum Release** rule will be evaluated, and its value will be applied to the allowable range.
- As the allowable range attempts to lower its maximum limit to the new value called for by the **Maximum Release** rule, it will bump into the bottom (minimum limit) of the range which has been set by the higher-priority **Induced Surcharge** rule. Since the lower-priority *maximum* rule cannot reduce the *minimum* limit of the allowable range, the maximum limit of the range will be assigned the same value as the minimum limit, resulting in a single-valued allowable range equal to the value of the desired **Induced Surcharge** release. Thus, combined, the two rules effectively produce a *specified* limit.

11.4.1 Defining an Induced Surcharge Rule

An **Induced Surcharge** rule is defined by three sets of attributes:

The Induced Surcharge Computation Method—Two methods by which the **Induced Surcharge** rule can determine the release requirement are available and must be selected as part of the rule definition:

Induced Surcharge Function—This method uses an equation or formula to determine the required release. The data needed to define this method includes the **Induced Surcharge Envelope Curve** and the **Time of Recession** of the project design storm. With this information and the current pool elevation, the equation is used to estimate the remaining space in the reservoir. Then, using the assumption that the current inflow is the *peak* of the inflow hydrograph, the method uses the current inflow and the remaining space estimate to determine the minimum required release to "just fill" the reservoir. EM 1110-2-3600 (USACE, 1987) provides a discussion of this function and its parameters.

ESRD Curves—This is a tabular representation of the **Emergency Spillway Release Diagram**, **Gate Regulation Curves**, or **Induced Surcharge Curves** that are depicted in the water control manual, as shown in the example in Figure 11.38. One of these will be present for each reservoir that includes induced surcharge operation in its regulation plan. The table is used to look-up the minimum required release.

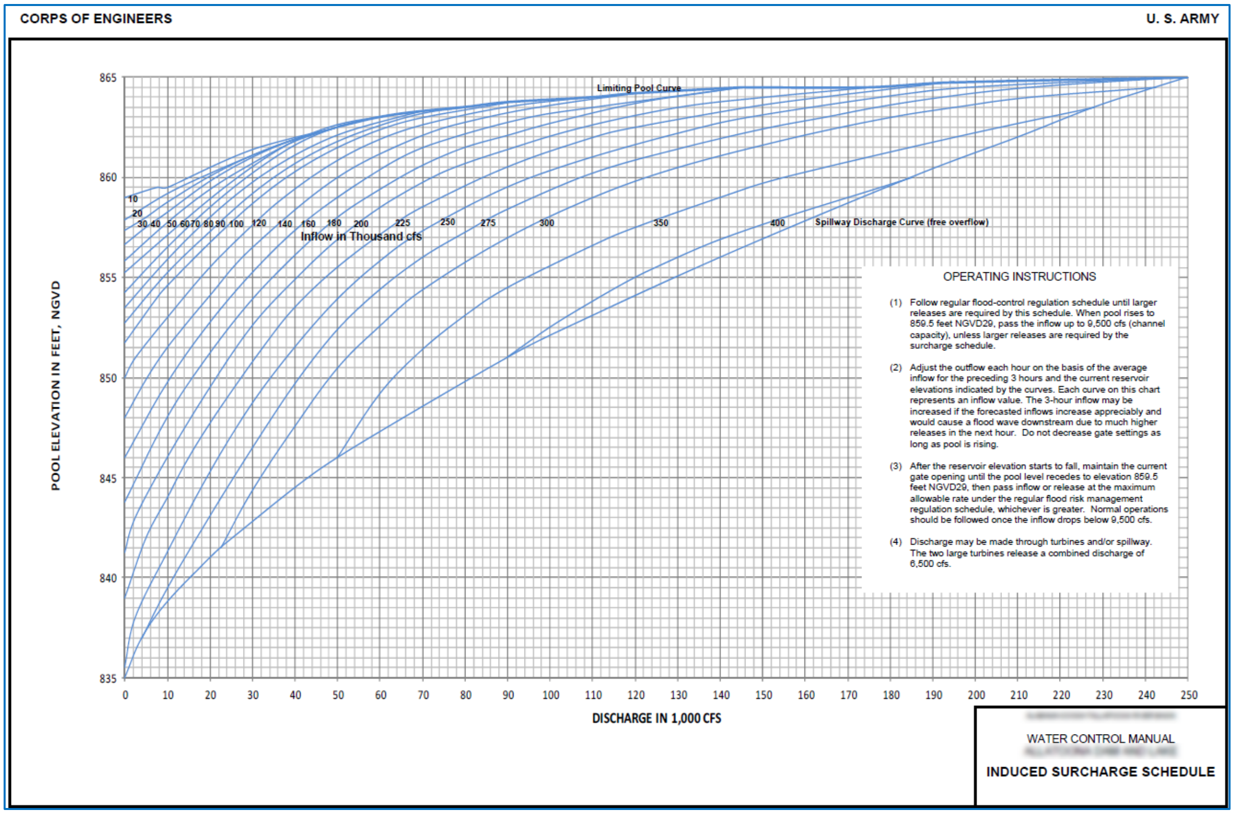


Figure 11.38 Example ESRD Curves

The Falling Pool Options—The Induced Surcharge computation method is used to determine the minimum release required by the rule while the pool is still rising. But once the pool starts to fall, that means that inflow is less than the release and the induced surcharge operation must transition to new scheme for determining the minimum release that should be made to draw the pool down. The Falling Pool Options described this scheme and the parameters needed to trigger this operation.

The Inflow Time Series Options—The *current* inflow to the pool is a necessary parameter for the induced surcharge operation. However, just what constitutes *current* inflow is debatable. Most regulation plans recommend using an average of the "observed" inflow over a specified number of hours as the input to the rule or operation. The Inflow Time Series Options dialog allows you to set identify the "time series function" for specifying the inflow to be used by the rule.

To define or edit an **Induced Surcharge** rule:

- Create a new rule as described in Section 11.1.1. Be sure to:
  - Select the reservoir as the release element (**Operates Release from:**). Although surcharge operations are conceptually tied to the operation of a gated spillway, the **Induced Surcharge** rule type is *only available when the selected release*

element is the ***reservoir*** so that it can utilize the total *available* release capacity of the reservoir.

- Select **Induced Surcharge** for the **Rule Type** in the **New Operating Rule** dialog.

- Or, if you want to edit a rule you have already created, select the **Induced Surcharge** rule from the **Zone-Rules Tree**.
- The edit panel of the **Operations** tab of the **Reservoir Editor** will display the **Induced Surcharge Rule Editor** (Figure 11.39). The name and description of the rule will appear in the **Induced Surcharge Rule** and **Description** fields.
- The rest of the rule editor has two potential views, one for each of the Induced Surcharge computation methods described above. Each view as well as the attributes that are common to both will be described in the following sections.

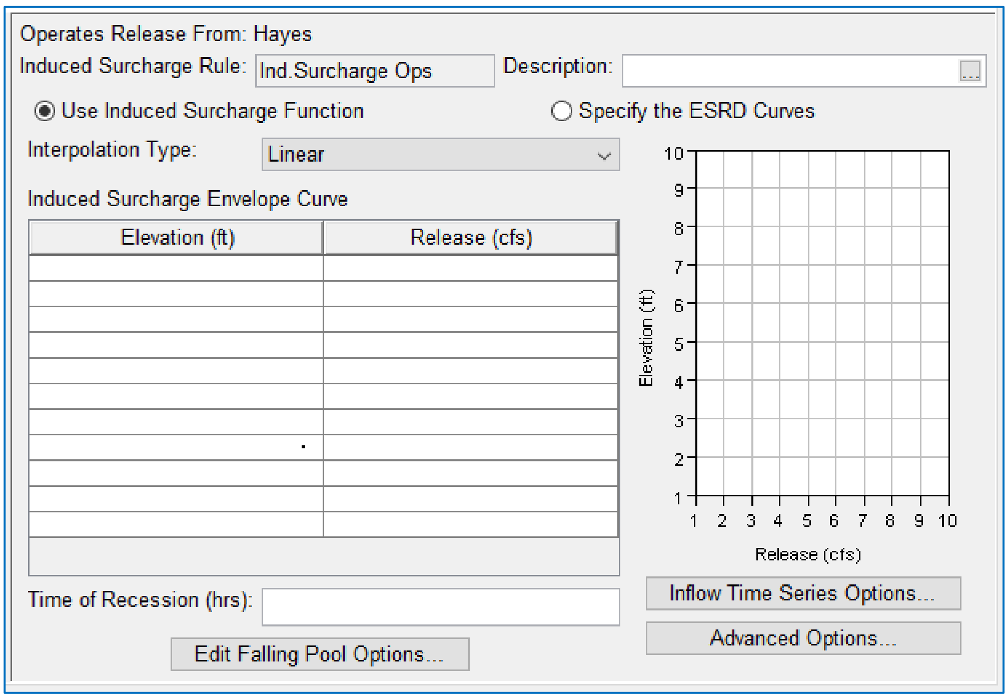


Figure 11.39 Induced Surcharge Rule Editor—Use Induced Surcharge Function Option

11.4.2 Use Induced Surcharge Function

Two parameters are needed to define the **Induced Surcharge Function**:

The **Envelope Curve** represents the maximum pool elevation the reservoir is allowed to reach while operating to manage an extreme inflow event.

The envelope curve is the top-most curve of the family of induced surcharge curves that can be found in the water control manual for any Corps reservoir that includes induced surcharge operation in its operating plan. The figure or "plate" that contains these curves goes by several names, the most common of which are: *Gate Regulate Schedule*, *Emergency Spillway Regulation Diagram*, *Induced Surcharge Schedule,* or some variation thereof.

The lowest elevation of the envelope curve is typically the top of the reservoir's Flood Control pool and its highest elevation is typically the top of the portion of the reservoir pool reserved for managing extreme inflow events. The highest elevation of the envelope curve is also the elevation at which the envelope curve intersects the spillway's maximum release capacity curve (with the gates fully open). The induced surcharge pool is defined by the range of elevations encompassed by the envelope curve.

When the induced surcharge operation is developed for a reservoir, the elevation range of the envelope curve is usually of function of the *safe* range for operating the spillway gates to induce surcharge. For reservoirs whose gated spillway is intended to be used exclusively for managing extreme events, *the spillway crest elevation* (*not* the elevation of the top of the closed gates) is used as the *starting* elevation of the envelope curve.

For reservoirs with very large spillway gates, where the gates are expected to be used to make releases for a variety of purposes under a wide range of conditions and over multiple operating zones, *the elevation of the top of the closed gates* is often used for the *starting* elevation of the envelope curve, although a lower elevation (below the top of the closed gates) may be used.

The *top* elevation of the envelope curve is the maximum pool elevation that can be reached *while passing inflow* with the gates *still* "in the water" but open to their maximum *safe* gate opening. Physical properties of the gates as well as manufacturers testing are used to identify the maximum safe gate opening. Vibration is just one of the factors used to identify an unsafe condition for the gates.

During induced surcharge operation, if the pool elevation reaches the envelope curve, the release specified by the envelope curve should equal inflow. If inflow is greater than the release capacity of the spillway at the maximum safe gate opening, the spillway gates are opened "fully" such that the bottom of the gates are "out of the water".

The **Time of Recession** is the length of time an incoming flood is expected to take to recede. It is used in the induced surcharge function to calculate the required release.

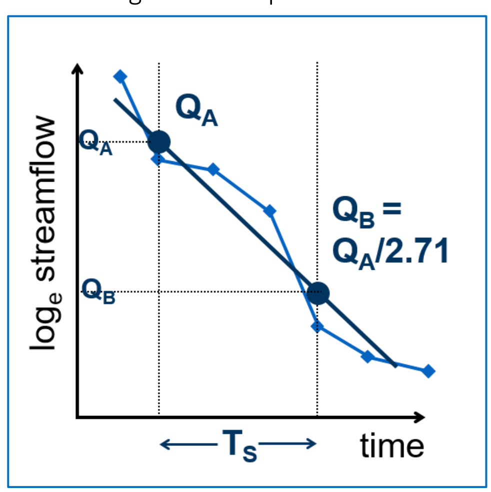


Figure 11.40 Estimating Time of Recession

The Time of Recession, T_S, is estimated using the inflow hydrograph of the spillway design storm. The process for estimating the Time of Recession includes the following steps:

- Several points are selected along the recession limb of the hydrograph and re-plotted on semi-log paper (Figure 11.40).

- A straight line is drawn through the plotted points. The time of recession, T_S, is the slope of the straight line.
- To calculate T_S, a point is selected on the straight line and its flow value, Q_A, is read from the graph.
- A second flow, Q_B, is calculated using the equation:

$$Q_B = Q_A/e = Q_A/2.718$$

- The point on the straight line whose flow equals Q_B is marked.
- The time of recession, T_S, is the difference in time between the two points placed on the straight line whose flows are Q_A and Q_B:

$$T_S = T_B - T_A$$

The **Induced Surcharge Function** logic assumes that the current inflow (from the end of the previous timestep) is the peak inflow of the event. Using that inflow and the time of recession, a recession hydrograph and the volume of water it represents is computed. Assuming the previous release is held constant, the volume represented by that release is subtracted from the volume in the recession limb to determine the volume of water that must be "managed". These volumes are illustrated in Figure 11.41.

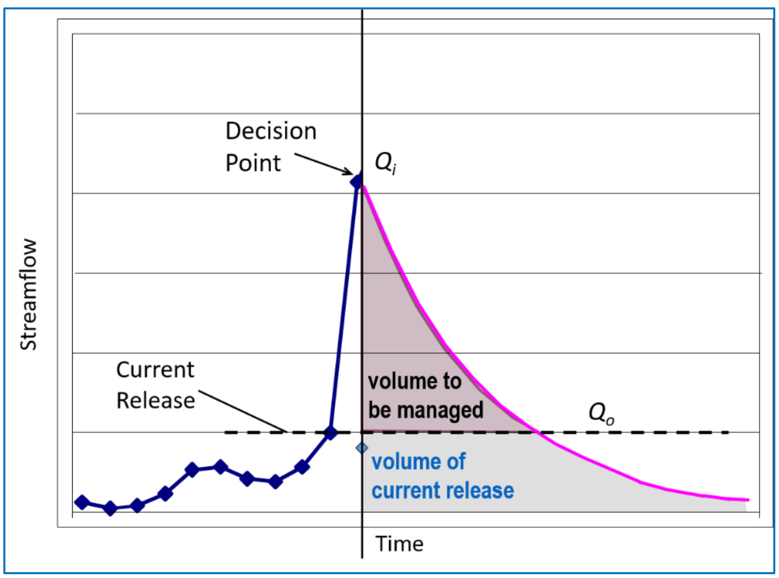


Figure 11.41 Estimating Time of Recession

To determine the *release* needed to manage the recession volume, the induced surcharge function logic must determine the remaining space in the reservoir. But the remaining space is a function of the current pool elevation and the elevation of the envelope curve for the needed *release*. Since the release is a function of the release, an iteration loop is used to converge on a release. As described above, the first iteration uses the release at the end of the previous timestep to calculate elevation of the envelope curve. Using the computed envelope curve elevation and the current pool elevation, the volume of the remaining space in the reservoir is calculated. If the *volume to be managed* (recession volume minus release volume) is greater than the *remaining volume*, a new release is calculated to make up the difference. In each successive iteration, the release estimate from the previous iteration is used to determine the new envelope curve elevation, the volume to be managed, the available space, and a new release estimate. The loop stops when the new release estimate is equal to the previous estimate (convergence), is greater than or equal to inflow, or the maximum number of iterations has been reached. If the envelope curve is not adequately detailed, is "bumpy", or is almost flat, the iteration loop may have trouble converging; messages will be sent to the console log indicating a problem and reporting that it is either holding the previous release to setting the release to inflow. Although an advanced option is available to adjust the maximum number of iterations, revising the envelope curve is usually a better approach when your model has trouble converging.

To define an **Induced Surcharge** rule using the **Induced Surcharge Function**:

- Select the radio button for **Use Induced Surcharge Function** in the **Induced Surcharge** rule editor.
- Select the **Interpolation Type.** Your options are: **Linear** or **Cubic** (**Step** is not included since the envelope should represent a smooth curve).
- Enter **Elevation** and **Release** data into the **Induced Surcharge Envelope Curve** table to describe the envelope curve. The elevation values entered in the table must be monotonically increasing. Use the finest detail you can produce, especially in the upper elevation range as the envelope curve flattens out.
- Enter the **Time of Recession** in hours. This constant describes the length of time an incoming flood is expected to take to recede. ResSim uses this time to compute the volume of water that must be evacuated to prevent overtopping the dam. See discussion in for further documentation regarding the Recession Time constant parameter (T_S).

After entering the above information, the **Induced Surcharge** rule editor should appear similar to the example shown in Figure 11.42.

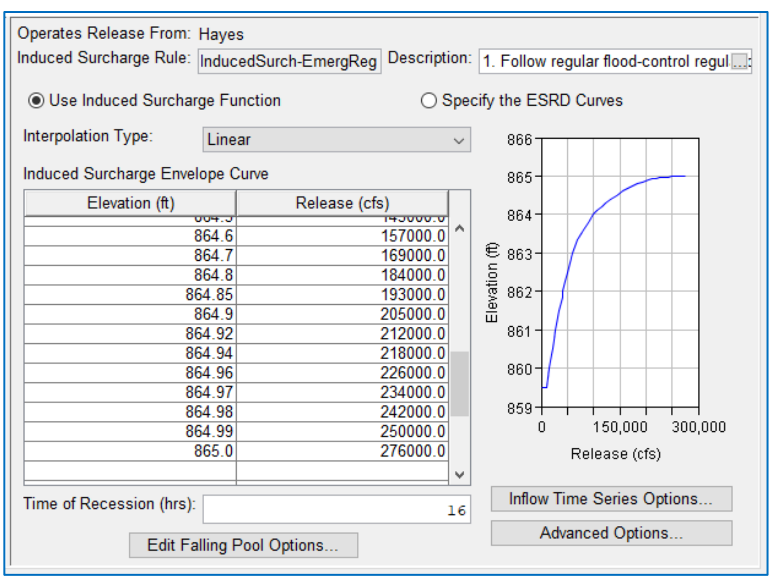


Figure 11.42 Induced Surcharge Rule Editor
Completed Example of Induced Surcharge Function

Plotting the Induced Surcharge Curves

In the **Induced Surcharge** rule editor for a rule that uses the **Induced Surcharge Function**, the thumbnail plot to the right of the Induced Surcharge Envelope Curve table shows a graph of the data entered in the table. Although the thumbnail plot can help you identify data entry errors, a full-size plot can provide more visualization options, like zoom.

To view a full-size plot of the envelope curve, *double-click* on its thumbnail plot. The full-size plot (Figure 11.43) will display the envelope curve specified in the rule as well as the reservoir's composite discharge capacity curve.

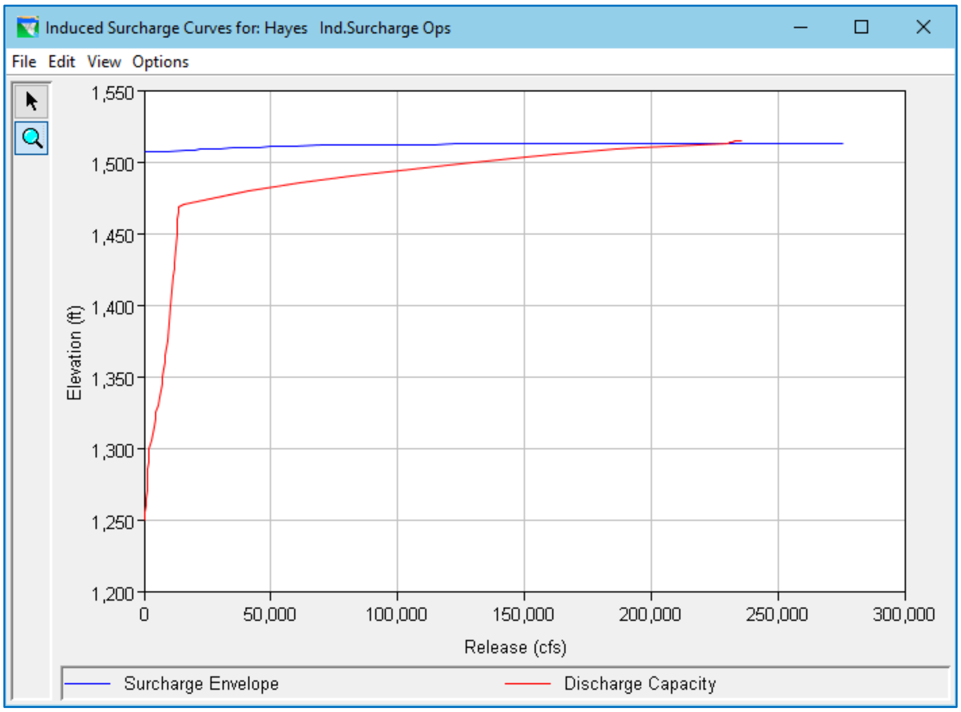


Figure 11.43 Full-Size Plot produced from the Induced Surcharge Rule's Thumbnail Plot

But, this full-size plot, opened from the thumbnail plot of the Induced Surcharge *Function's* envelope curve, is unlike the plot opened from any other thumbnail plot in ResSim. This plot has an **Options** menu…

The **Options** menu (Figure 11.44) provides access to features that will allow you to specify a set of Inflow or Rate of Rise values from which a family of induced surcharge curves will be computed and displayed in the plot. The **Options** menu contains the following options:

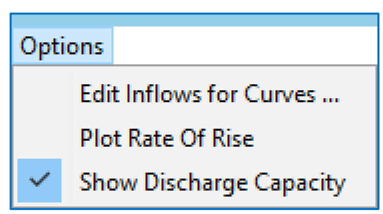


Figure 11.44 Induced Surcharge Curve Plot—Options Menu

Edit Inflows for Curves…—This option opens the **Specify Inflows for Induced Surcharge Curves** dialog (Figure 11.45) in which you can specify a list of inflow values that will be used to generate a family of induced surcharge release curves. To add more rows to the Inflows table, right-click on a cell in the table and select **Insert Rows** or **Append Row** from the context menu. Click **OK** to save your entries or changes. The inflow values you entered will be saved with the rule so that you do not need to enter them again.

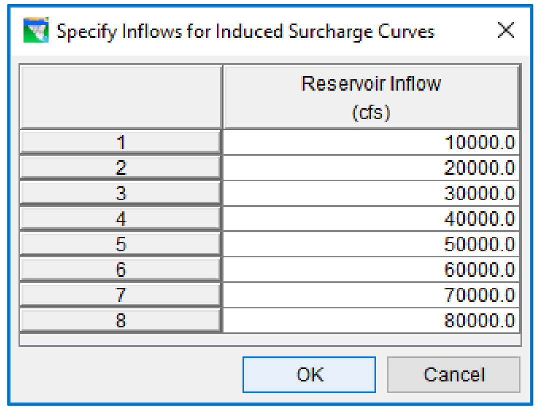


Figure 11.45 Edit Inflow for Curves…

Plot Rate of Rise—this option is a toggle switch that indicates whether the family of induced surcharge curves will be computed and plotted for a set of inflow or rate of rise values. By default, the switch is OFF (unchecked) and the plot will display the family of curves by inflow value (if inflows have been entered). By selecting this option, you can switch the plot to show the family of curves by rate of rise value.

*** When this switch is ON (checked), the first option in the **Options** menu changes to **Edit Rate of Rise for Curves…** If you haven't already done so, you will need to enter a set of rate-of-rise values in order to generate the associated family of curves.

Show Discharge Capacity—also a toggle switch. Its state (ON or OFF) indicates whether or not the plot should include the reservoir's total Discharge Capacity curve. Figure 11.46 and Figure 11.47 show the Induced Surcharge Curves Plot *with* and *without* the Discharge Capacity Curve.

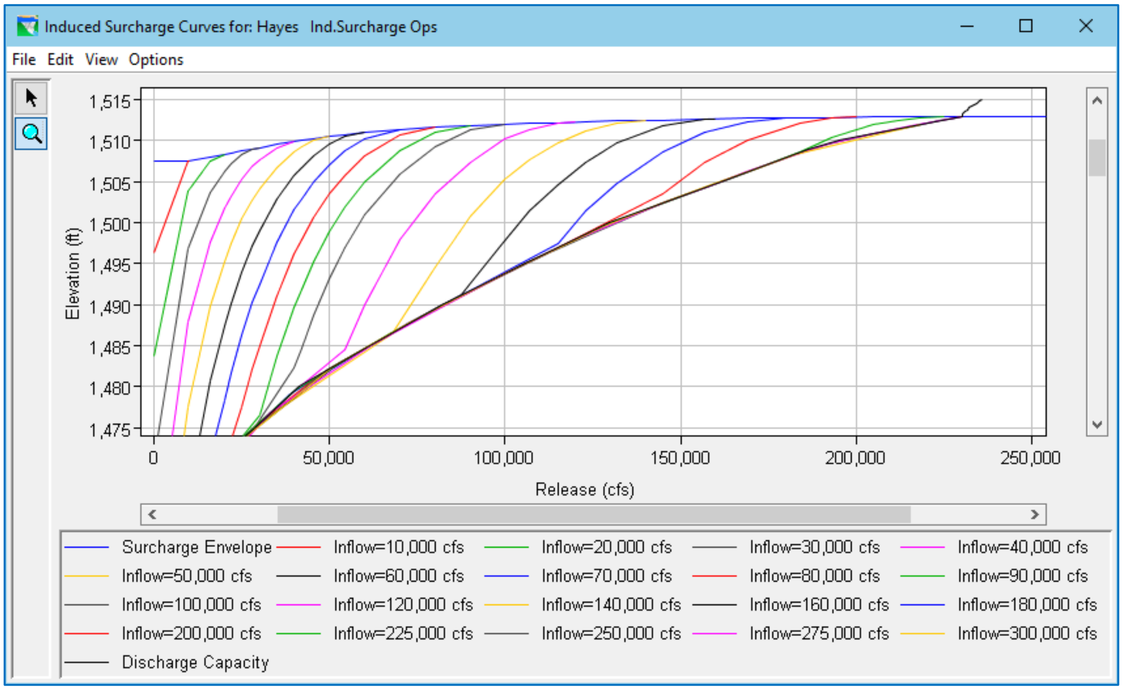


Figure 11.46 Family of Computed Induced Surcharge Curves—with the Discharge Capacity Curve

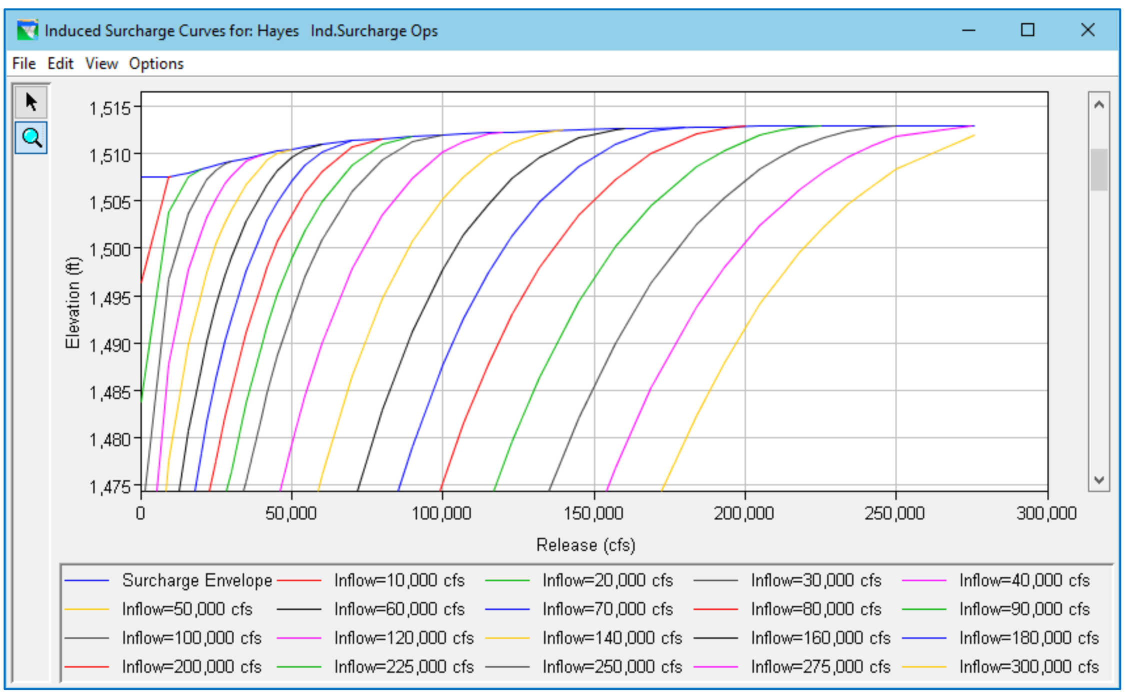


Figure 11.47 Family of Computed Induced Surcharge Curves—without the Discharge Capacity Curve

Once you have entered a set of inflows or rate of rise values, the resulting plot of the computed induced surcharge curves should resemble the diagram in the water control manual from which you estimated the envelope curve. If the plotted curves do not match the original diagram, try these adjustments:

- Add more detail to the envelope curve. Both the beginning and end of the envelope curve should be well defined. In general, a well-defined envelope curve will have 10-20 values to describe the nose of the envelope, 5-10 values to describe the middle, and another 10-20 values to describe the tail (even if all those values describe a straight line).

  > **Tip** When developing the table of values that describe the envelope curve, start with a regular interval of *release* values and find their associated elevation values in the diagram in the water control manual. This should give you a good definition of the middle and tail of the envelope. Then, using a progressively smaller interval of release, add more detail to the beginning so that the nose of the envelope looks smooth when plotted, even when zoomed in.

- Once you have an envelope curve that produces in a set of smooth induced surcharge curves, revise the time of recession.
- Continue to fine-tune the time of recession until the plotted curves are a good match to those in the manual. Think of this as a binary search. Start with a fairly large change from your original entry to determine if the new curves are going in the right direction. Then use successively smaller changes as your plotted curves get closer to those in the manual.

IF, however, you have tried various adjustments to both the envelope curve and the time of recession and have been unable to produce a set of plotted curves that closely resemble the original diagram, then either the induced surcharge curves shown in the manual were not created using the *induced surcharge function* or they were altered to meet criteria that cannot be reflected by the function. In this case, you may find it necessary to **Specify the ESRD Curves** rather than **Use the Induced Surcharge Function.**

11.4.3 Specify the ESRD Curves

The option to **Specify the ESRD Curves** was added to the Induced Surcharge rule to enable you to provide a lookup table that describes the family of induced surcharge curves from which the required release can be found or interpolated. This option should be used when the induced surcharge curves were not created using the concepts of the *induced surcharge function* or were altered to meet conditions or criteria that are not considered or accounted for in the function.

To define an **Induced Surcharge** rule by entering a table elevations and releases that describe the curves of an Emergency Spillway Regulation Diagram:

- Select the radio button for **Specify the ESRD Curves** in the **Induced Surcharge** rule editor. Figure 11.48 illustrates the initial view of the Induced Surcharge rule with the **Specify the ESRD Curves** option selected.

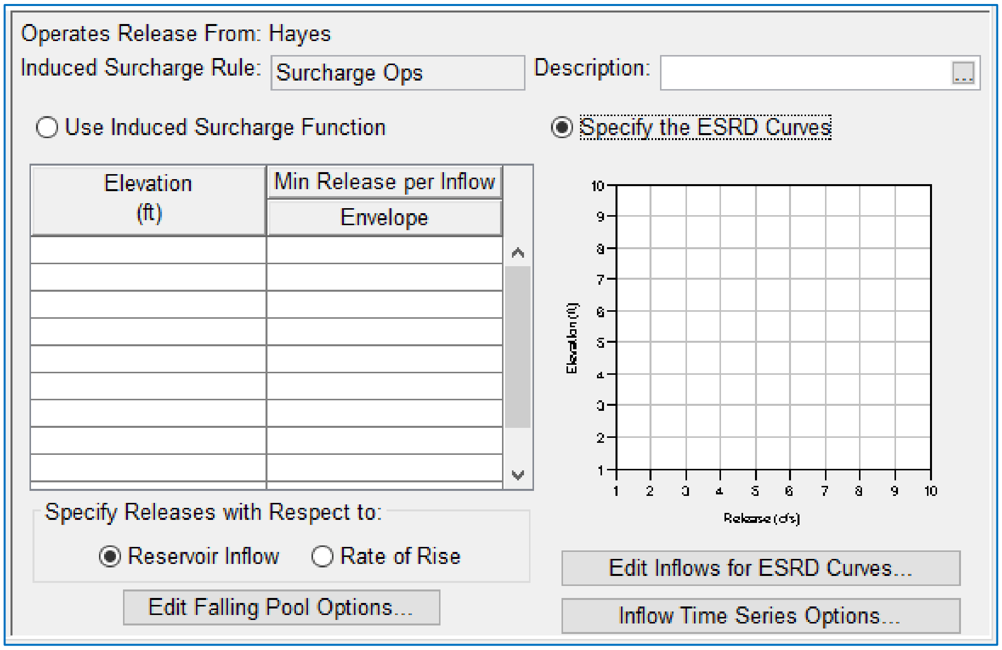


Figure 11.48 Induced Surcharge Rule Editor—Specify the ESRD Curves Option

Although you may not notice immediately, the **Specify the ESRD Curves** view of the Induced Surcharge rule editor differs from the **Use Induced Surcharge Function** view in the following ways:

- The header changed for the second column of the table. This one has two rows. The second row will display the value of inflow or rate or rise for each induced surcharge curve in the table.
- A pair of radio buttons were added below the table. The selected button identifies the type of curves in the table — *inflow* curves or *rate of rise* curves.
- The thumbnail plot is bigger. Since this thumbnail plot will show the full family of curves as entered in the table, a bigger plot was used so you could see all the curves at a glance.
- An **Edit...** button was added below the thumbnail plot and the **Advance Options** button was removed. The new **Edit** button will open a dialog for specifying the Inflow or Rate of Rise values for each induced surcharge curve in the table. The Advanced Options only apply to the **Induced Surcharge Function** form of the rule.

- Identify the type of induced surcharge curves that will be entered in the ESRD table by selecting one of the two **Specify Releases with Respect to:** radio buttons located below the table. Your choices are:
  - **Reservoir Inflow**—use this option if the curves plotted (or tabulated) in your water manual are labeled with increasing values of inflow. This parameter is used when plotting the induced surcharge curves if the expected user of the diagram is a water manager or regulator. The water manager is back in the office with access to a variety of computing resources and data that he can use to estimate reservoir inflow.

- o **Rate of Rise**—use this option if the curves plotted (or tabulated) in your water manual are labeled with increasing rates of rising pool elevation. This parameter is often used when plotting the induced surcharge curves if the expected user of the diagram is a dam operator. The assumption is that the dam operator would not have time during a major inflow event to perform the calculations necessary to estimate inflow but can quickly determine the rate at which the pool is rising glancing at the pool's stage gage on a regular interval.

- Depending on your selection, the **Edit…** button below the thumbnail plot will either be labeled **Edit Inflows for ESRD Curves…** or **Edit Rate of Rise for ESRD Curves…**

  Click this **Edit…** button to open a dialog (Figure 11.49) where you will enter a list of values (of **Reservoir Inflow** or **Rate of Rise**) that identify the curves in the ESRD table. Click **OK** to save your list and close the dialog. The ESRD table will update to include a column for each value in the list.

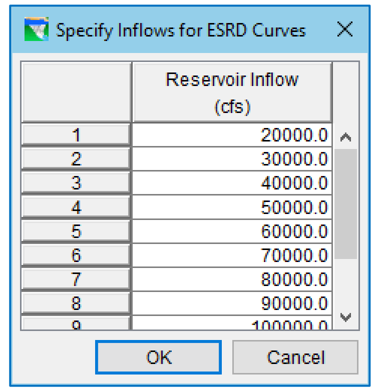
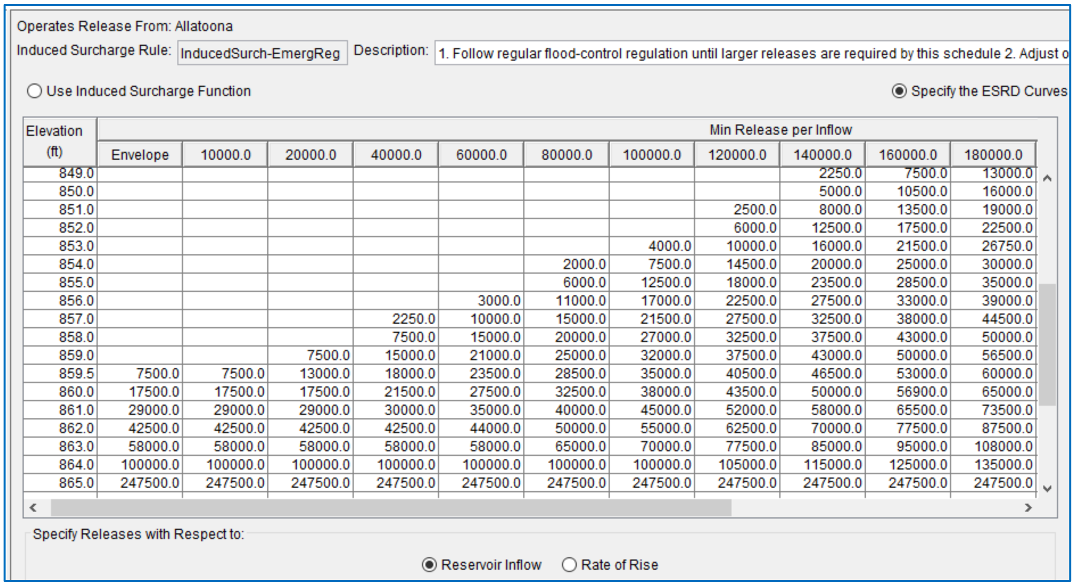


Figure 11.49 Induced Surcharge Rule—Inflows for ESRD Curves

  Note: if you change one or more values in the list of inflows or rates of rise *after* you have filled in the ESRD Curves table, the data in the columns of the table associated with the *original* values will *disappear*. However, if you insert new values (rows) into the list without changing the originally entered values, the ESRD Curves table will expand to include a new column for each new value added to the list without losing the data in the columns associated with the original values.

- Enter **Elevation** and **Release** data into the **ESRD Curves** (**Elevation** versus **Min Release per Inflow** or **Rate of Rise**) table. The first two columns in the table represents the envelope curve. When paired with the elevation column, the remaining columns represent the minimum release curve for each inflow or rate of rise values that you specified in the previous step. NOTE — the elevation data must be monotonically increasing and at the finest detail you can manage. Since ResSim will use linear interpolation to determine a minimum release for a given elevation and inflow (or rate of rise), a significant elevation range and detail will be needed in order for each curve to be adequately defined.

  An example of a complete (but not perfect) table of data defining a set of ESRD curves is illustrated in Figure 11.50 below. In this example, the cells in the upper left corner of the completed table are empty because the curves for the lower inflow are not defined for the lower elevations. In other words, when the reservoir is not too full and inflows are still low, increased releases are not required.



Operates Release From: Allatoona

Induced Surcharge Rule: InducedSurch-EmergReg Description: 1. Follow regular flood-control regulation until larger releases are required by this schedule 2. Adjust o

Use Induced Surcharge Function

Specify the ESRD Curves

Elevation (ft)	Min Release per Inflow										
	Envelope	10000.0	20000.0	40000.0	60000.0	80000.0	100000.0	120000.0	140000.0	160000.0	180000.0
849.0									2250.0	7500.0	13000.0
850.0									5000.0	10500.0	16000.0
851.0								2500.0	8000.0	13500.0	19000.0
852.0								6000.0	12500.0	17500.0	22500.0
853.0							4000.0	10000.0	16000.0	21500.0	26750.0
854.0						2000.0	7500.0	14500.0	20000.0	25000.0	30000.0
855.0						6000.0	12500.0	18000.0	23500.0	28500.0	35000.0
856.0					3000.0	11000.0	17000.0	22500.0	27500.0	33000.0	39000.0
857.0				2250.0	10000.0	15000.0	21500.0	27500.0	32500.0	38000.0	44500.0
858.0				7500.0	15000.0	20000.0	27000.0	32500.0	37500.0	43000.0	50000.0
859.0			7500.0	15000.0	21000.0	25000.0	32000.0	37500.0	43000.0	50000.0	56500.0
859.5	7500.0	7500.0	13000.0	18000.0	23500.0	28500.0	35000.0	40500.0	46500.0	53000.0	60000.0
860.0	17500.0	17500.0	17500.0	21500.0	27500.0	32500.0	38000.0	43500.0	50000.0	56900.0	65000.0
861.0	29000.0	29000.0	29000.0	30000.0	35000.0	40000.0	45000.0	52000.0	58000.0	65500.0	73500.0
862.0	42500.0	42500.0	42500.0	42500.0	44000.0	50000.0	55000.0	62500.0	70000.0	77500.0	87500.0
863.0	58000.0	58000.0	58000.0	58000.0	58000.0	65000.0	70000.0	77500.0	85000.0	95000.0	108000.0
864.0	100000.0	100000.0	100000.0	100000.0	100000.0	100000.0	100000.0	105000.0	115000.0	125000.0	135000.0
865.0	247500.0	247500.0	247500.0	247500.0	247500.0	247500.0	247500.0	247500.0	247500.0	247500.0	247500.0

Specify Releases with Respect to:

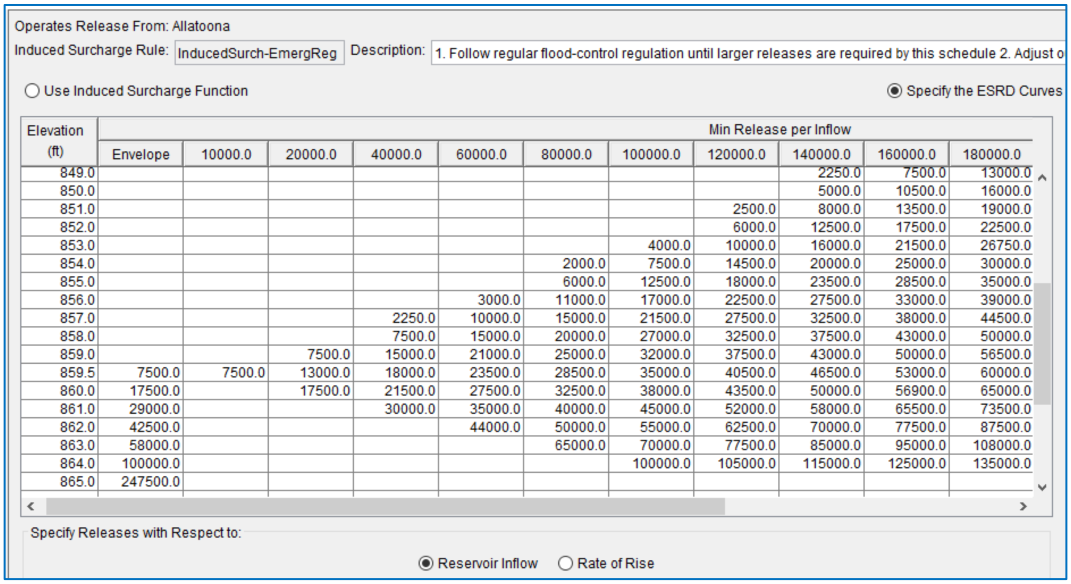
Reservoir Inflow Rate of Rise

Figure 11.50 Example of a Complete ESRD Table

Once you start entering data in a row or column, *the rest of the row or column should be filled in*. Do not leave blank cells *between* cells with data, in either rows or columns of the table.

Although ResSim will allow it, the cells in the lower left corner of the table (the last few cells of each column) should not be left blank. Figure 11.51 shows an example of an incomplete table, where the cells in the lower left corner have been left blank.

The cells in the lower left corner of the ESRD table *should* define where each curve intersects and merges with the envelope curve. Some users leave these cells blank because defining the intersection point for each curve can be troublesome and the remaining cells are viewed as redundant. For each curve's intersection point an additional elevation row may need to be added to the table; as a result, a release for those added elevations must be entered for each curve in the table; so, you can see why this effort might be considered troublesome. Since the remaining cells of the table below the intersection elevation of each curve represent the region where the curve merges (or is coincident) with the envelope curve, these cells can be filled with a copy of the data from the envelope curve.


Operates Release From: Allatoona

Induced Surcharge Rule: InducedSurch-EmergReg Description: 1. Follow regular flood-control regulation until larger releases are required by this schedule 2. Adjust o

Use Induced Surcharge Function

Specify the ESRD Curves

Elevation (ft)	Min Release per Inflow										
	Envelope	10000.0	20000.0	40000.0	60000.0	80000.0	100000.0	120000.0	140000.0	160000.0	180000.0
849.0									2250.0	7500.0	13000.0
850.0									5000.0	10500.0	16000.0
851.0								2500.0	8000.0	13500.0	19000.0
852.0								6000.0	12500.0	17500.0	22500.0
853.0							4000.0	10000.0	16000.0	21500.0	26750.0
854.0						2000.0	7500.0	14500.0	20000.0	25000.0	30000.0
855.0						6000.0	12500.0	18000.0	23500.0	28500.0	35000.0
856.0					3000.0	11000.0	17000.0	22500.0	27500.0	33000.0	39000.0
857.0				2250.0	10000.0	15000.0	21500.0	27500.0	32500.0	38000.0	44500.0
858.0				7500.0	15000.0	20000.0	27000.0	32500.0	37500.0	43000.0	50000.0
859.0			7500.0	15000.0	21000.0	25000.0	32000.0	37500.0	43000.0	50000.0	56500.0
859.5	7500.0	7500.0	13000.0	18000.0	23500.0	28500.0	35000.0	40500.0	46500.0	53000.0	60000.0
860.0	17500.0		17500.0	21500.0	27500.0	32500.0	38000.0	43500.0	50000.0	56900.0	65000.0
861.0	29000.0			30000.0	35000.0	40000.0	45000.0	52000.0	58000.0	65500.0	73500.0
862.0	42500.0				44000.0	50000.0	55000.0	62500.0	70000.0	77500.0	87500.0
863.0	58000.0					65000.0	70000.0	77500.0	85000.0	95000.0	108000.0
864.0	100000.0						100000.0	105000.0	115000.0	125000.0	135000.0
865.0	247500.0										

Specify Releases with Respect to:

Reservoir Inflow Rate of Rise

Figure 11.51 Example of an Incomplete ESRD Table

To illustrate why you should *not* leave the cells in the lower left corner of the ESRD table blank, full size plots of the ESRD curves from each example ESRD table were generated and are shown in Figure 11.52 and Figure 11.53. These plots are zoomed-in to the region where the curves should intersect or merge with the envelope curve. In Figure 11.52, you can see that the curves for the *complete* table properly intersect and merge with the envelope curve. In Figure 11.53, the curves for the *incomplete* table do NOT intersect the envelope curve. Since ResSim will not extrapolate the curves beyond the data provided, if the elevation and inflow used to lookup a release from the *incomplete* table happen to fall into one of the *undefined regions,* between the end of the curve data and the envelope, the resulting release may not be what you intended or expected.

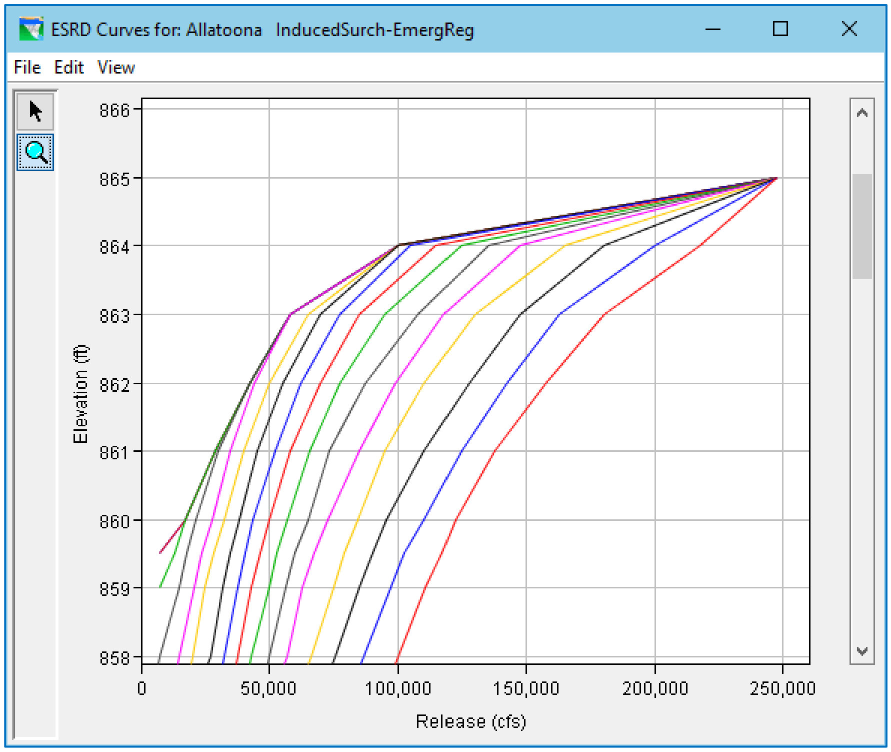


Figure 11.52 The Curves from the Complete ESRD Table Example

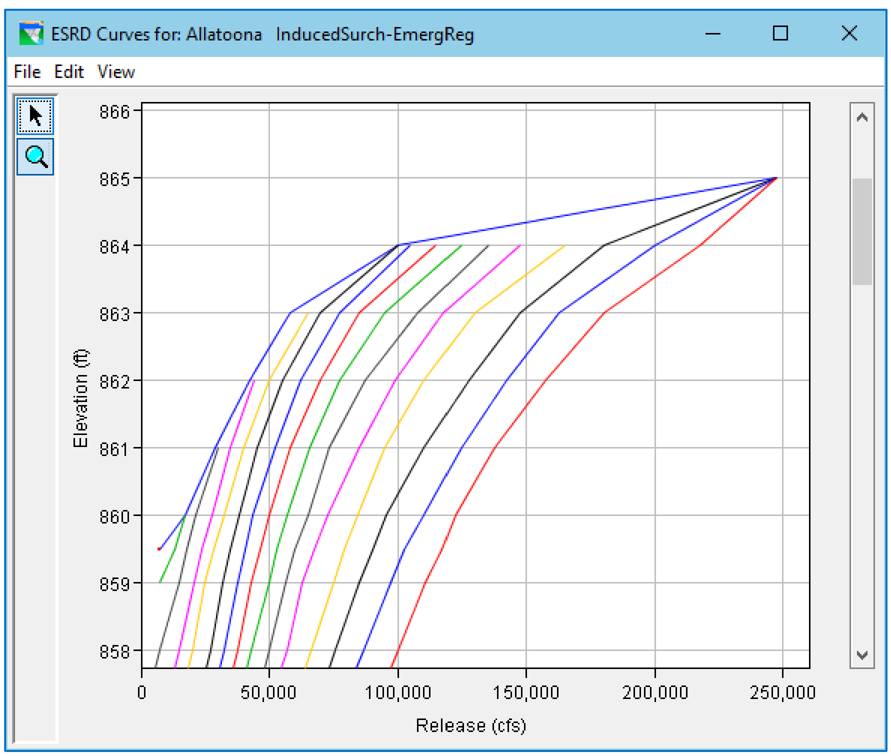


Figure 11.53 The Curves from the Incomplete ESRD Table Example

After entering the above information, the **Induced Surcharge** rule editor should appear similar to the example shown in Figure 11.54.

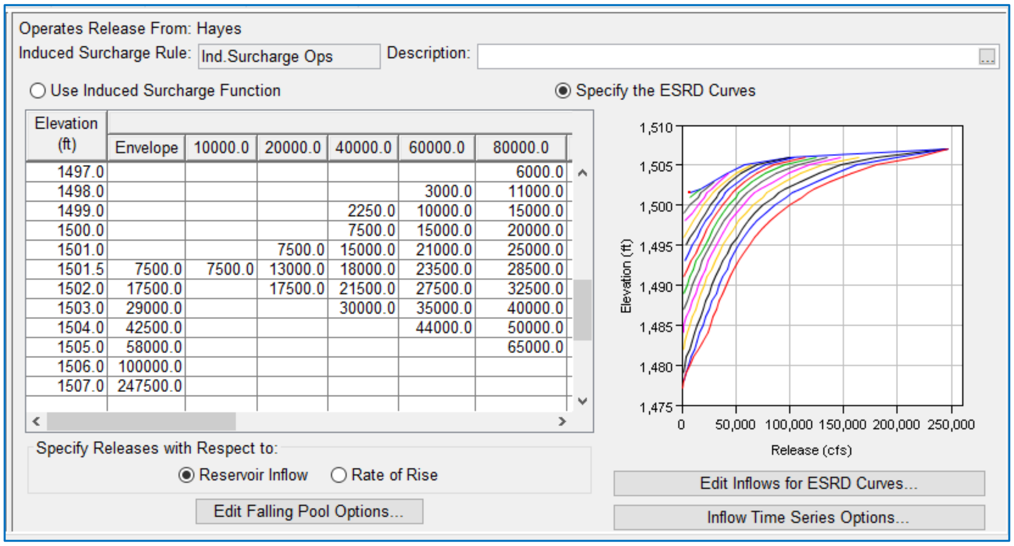


Figure 11.54 Induced Surcharge Rule Editor
Completed Example of Specifying the ESRD Curves

The thumbnail plot reflects the data entered in the **ESRD Curves** table. To see a full-size plot of the family of curves, *double-click* on the thumbnail plot (Figure 11.55).

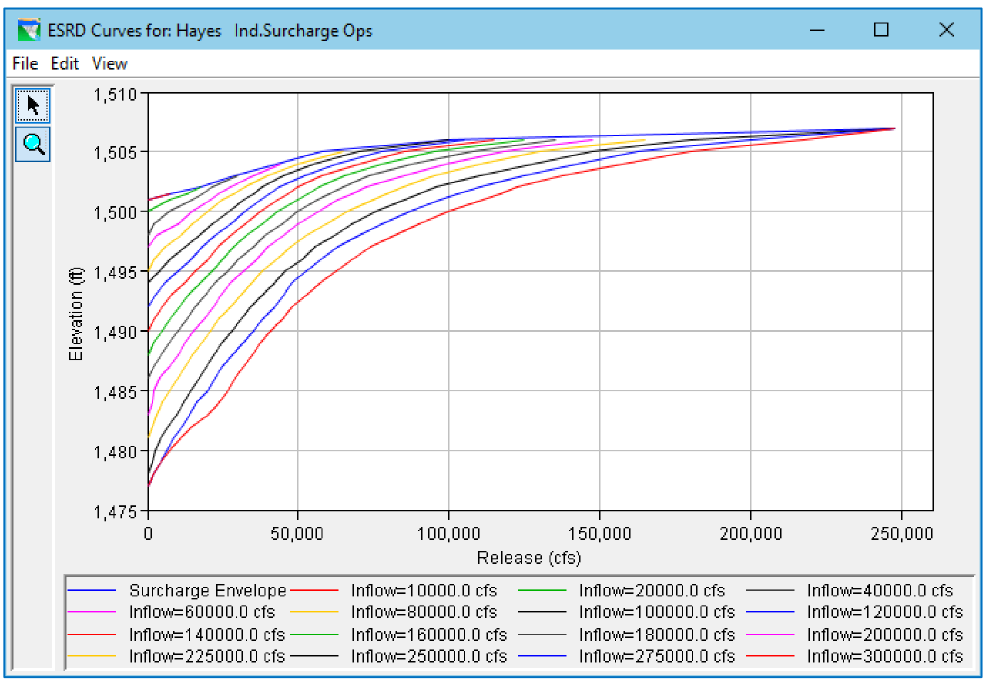


Figure 11.55 Plot of Induced Surcharge Curves for Specified ESRD Inflow Values

11.4.4 Falling Pool Options

To specify how the induced surcharge minimum release should be determined when the reservoir starts falling, you must set up the Falling Pool Options. To do so:

- *Click* the **Falling Pool Options Edit** button. The **Falling Pool Options** dialog will open (Figure 11.56).
- Specify each of the following attributes:

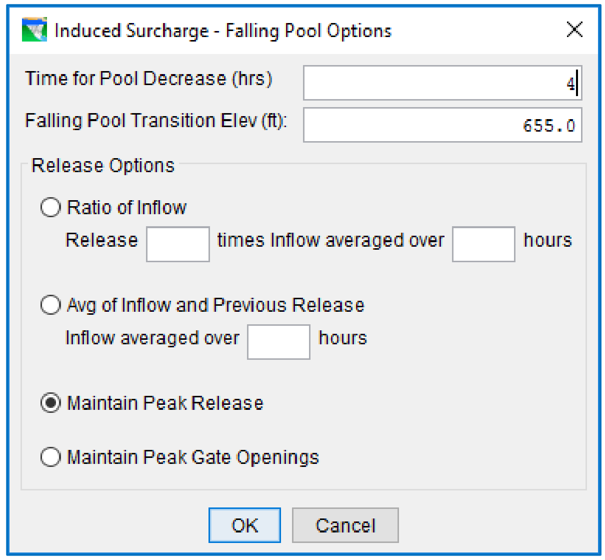


Figure 11.56 Induced Surcharge—Falling Pool Options

Time for Pool Decrease—enter a value in hours. This parameter describes the required number of successive hours the reservoir pool level must be falling before the minimum release determination transitions from rising pool emergency spillway releases to falling pool releases. During this period, the rule often appears to hold the peak rising pool release since one of the rules on the rising pool release determination is logic is to not reduce the release as long as the pool is rising.

Falling Pool Transition Elevation—enter a value in feet (meters). This parameter represents the pool elevation above which Falling Pool releases will be made. Once the pool elevation falls below this elevation, the Induced Surcharge rule will no longer operate, and ResSim will resume releases based on other rules in the active zone.

Release Options—this is the method for determining the minimum release while the pool is falling until the transition elevation is reached. Some of these options may seem counterintuitive. Just remember, the objective during falling pool is to maintain the falling pool trend (to draw down the reservoir) so the release at this point must be *greater* than inflow. Choose from:

Ratio of Inflow—this method requires two parameters:

- The *ratio value* — a multiplier of inflow. For example, to release 120% of inflow, enter a value of 1.2
- An *averaging period* — like the rising limb, you need to specify the period, in hours, over which to determine inflow for the use of this method.

Average of Inflow and Previous Release—like previous method, this method requires that you specify an averaging period for inflow, in hours.

Maintain Peak Release—select this method is you want the release to hold at the value it was releasing when the pool started to fall.

Maintain Peak Gate Openings—select this method if you want ResSim to determine the gate setting (or percent open) of the outlets at the time the pool started to fall. For each subsequent timestep until the transition elevation is reached, that gate opening (or percent open) will be used to determine the minimum release.

- *Click* **OK** to close the **Falling Pool Options** dialog.

11.4.5 Inflow Time Series Options

To specify the how the *current* value of inflow is determined from the inflow time series, follow the instructions below. Note: if you skip this step, the default of **Previous Value** will be used.

- *Click* the **Inflow Time Series Options** button. This editor defines how the reservoir inflow time series is interpreted when used in making induced surcharge operation release decisions. The editor and its options will vary depending on your Induced Surcharge computation method:
- If you are using the **Induced Surcharge Function**, Figure 11.57 shows the editor that will open.
  - The **Function** list and the associated parameters of **Offset** and **Period** are described in the Independent Variable Editor's Time Series Options section in Appendix C.

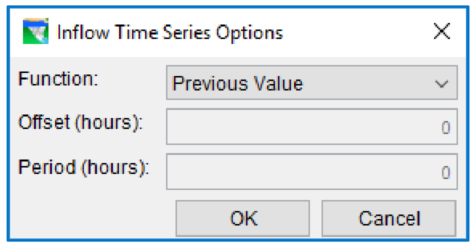


Figure 11.57 Induced Surcharge—Inflow Time Series Options

- If you are using the **ESRD Curves**, Figure 11.58 shows the more complex editor that will open. The more complex editor allows you to specify any external times series or model variable as the Inflow to be used by the rule—rather than the default, computed inflow.

  Variable for ESRD Lookup—

  Inflow—To use the default, computed inflow and the standard Inflow Time Series Options, select the first radio button, **Inflow**. The associated **Time Series Options** will be specified above the radio buttons.

  User Selected—To specify an external variable or model variable as the "inflow" for the ESRD lookup, select the second radio button, **User Selected**. The lower portion of the editor will be enabled; use it to select the variable you want to use as "inflow" and its associated **Time Series Options**.

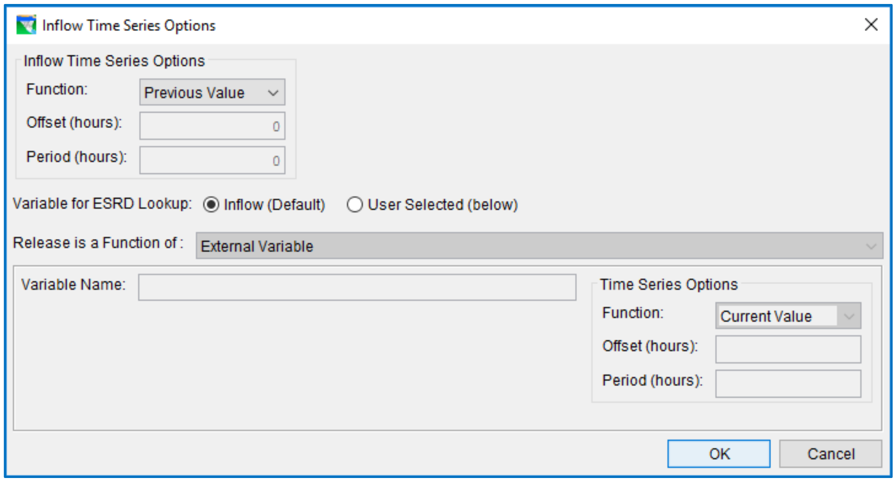


Figure 11.58 Inflow Time Series Options—External Variable

When you have finished specifying all the required attributes for your Induced Surcharge rule, press Apply to save your settings. Then, be sure the rule is added to and correctly positioned at the top of the rule stack in each zone where the rule should apply.

11.5 Flow Rate of Change Limit Rule

A **Flow Rate of Change Limit** rule specifies the allowable change when increasing or decreasing release values (a.k.a., "ramping rates"). A single rule of this type will only limit a rising release or a falling release, but not both. To describe both increasing and decreasing limits, you must define two rules and set the type of one to increasing and the other to decreasing. A rule of this type can be assigned to any release element to influence the behavior of that element.

> *Note:* **Rate of Change rules have more impact than you might think…**
>
> Unless otherwise noted, ResSim's decision logic tries to meet its objectives as fast as possible, usually within the current timestep. But rate-of-change constraints are in direct opposition to that tendency; their objective is to slow things down. To more fully address this slow-down objective, logic was added to the downstream control evaluation and guide curve release determination to account for rate of change constraints. So, in addition to acting as normal release limit rules, Rate of Change rules also impact downstream control and guide curve releases by *extending* the time window over which the two methods try to meet their objectives.

To define or edit a **Flow Rate of Change Limit** rule:

- Create a new rule as described in Section 11.1.1. Be sure to:
    - Select **Flow Rate of Change Limit** for the **Rule Type** in the **New Operating Rule** dialog.
- Or, if you want to edit a **Flow Rate of Change** rule you have already created, select it from the **Zone-Rules Tree**.
- The edit panel of the **Operations** tab of the **Reservoir Editor** will display the **Flow Rate of Change Limit** rule editor (Figure 11.59). The name and description of the rule will appear in the **Release Rate of Change Limit** and **Description** fields.

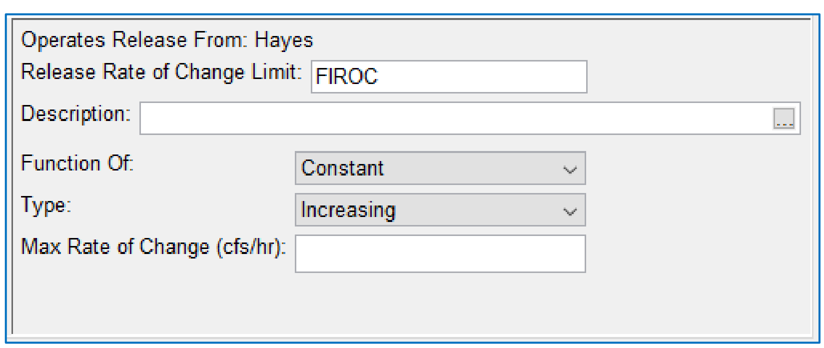


Figure 11.59 Flow Rate of Change Limit Rule Editor—Function of Constant

Function Of—A **Flow Rate of Change** rule can be **Constant**, or it can vary with:

- **Reservoir Inflow**
- previous **Release**
- **Pool Elevation**

Type—this is the **Limit** type. Select either **Increasing** or **Decreasing**. Note: when evaluated, an *Increasing* Flow Rate of Change Limit results in a *maximum* release limit and a *decreasing* Flow Rate of Change Limit results in *minimum* release limit

Max Rate of Change—depending on your **Function of** selection…

Constant—you will either be presented with a single field — for a constant value in units of *flow per hour*. Figure 11.59 shows this view of the Flow Rate of Change rule editor.

Inflow, **Release**, or **Elevation**—you will either be presented with a table. Enter the relationship between the selected independent variable and the maximum rate of change limit. The limit is described in units of flow (cms or cfs) ***per hour***, regardless of the compute interval. For example, if you enter 500 cfs/hr with a compute interval of 12 hours, then this rule describes the maximum flow change per timestep as 6,000 cfs. Figure 11.60 shows this view of the Flow Rate of Change rule editor.

Interpolate—enter the interpolation type for looking up values in the table. You can choose from the standard options of **Linear**, **Cubic**, or **Step**. See Section 11.2.2.2 for an explanation of the interpolation options.

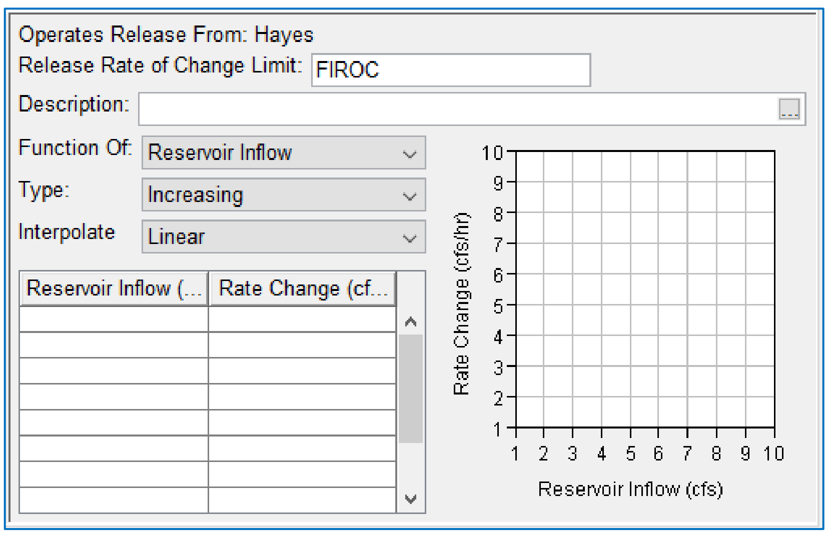


Figure 11.60 Flow Rate of Change Limit Rule Editor—Function of Time-Series

- When you have finished entering data for the Flow Rate of Change rule, be sure to click **Apply** before moving on to the next rule.

11.6 Elevation Rate of Change Limit Rule

An **Elevation Rate of Change Limit** rule describes the allowable change when increasing or decreasing pool elevation values. A single rule of this type will only limit a rising pool or a falling pool, but not both. To describe both increasing and decreasing limits, you must define two rules and set the type of one to increasing and the other to decreasing. Since this rule watches the pool elevation, this rule type is only available for the reservoir (pool) release element

To define or edit an **Elevation Rate of Change Limit** rule:

- Create a new rule as described in Section 11.1.1. Be sure to:
    - Select **Elevation Rate of Change Limit** for the **Rule Type** in the **New Operating Rule** dialog.
- Or, if you want to edit an **Elevation Rate of Change** rule you have already created, select it from the **Zone-Rules Tree**.
- The edit panel of the **Operations** tab of the **Reservoir Editor** will display the **Elevation Rate of Change Limit** rule editor (Figure 11.61). The name and description of the rule will appear in the **Elevation Rate of Change Limit** and **Description** fields.

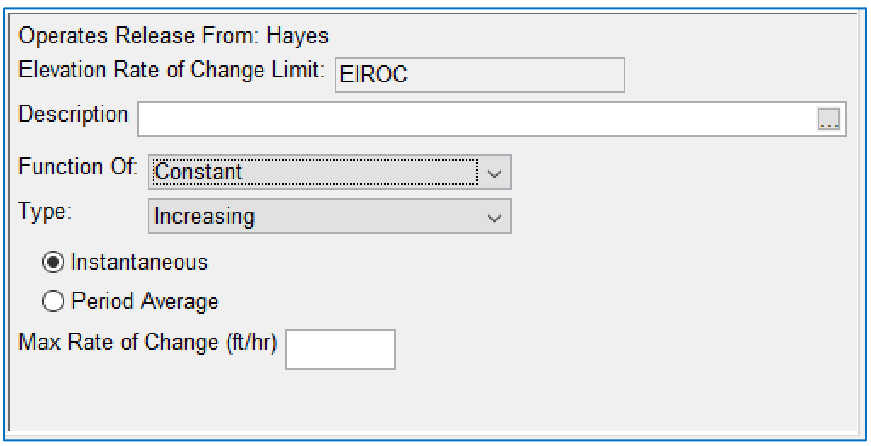


Figure 11.61 Elevation Rate of Change Limit Rule Editor—Function of Constant

Function Of—An **Elevation Rate of Change** rule can be **Constant**, or it can vary with:

- **Reservoir Inflow**
- Reservoir **Release**

Type—this is the **Limit** type. Select either **Increasing** or **Decreasing**. Note: when evaluated, an *Increasing* Flow Rate of Change Limit results in a *maximum* release limit and a *decreasing* Flow Rate of Change Limit results in *minimum* release limit

Instantaneous or **Period Average** Radio Buttons—These radio buttons allow you to choose whether or not the rate of change rule applies to the current timestep (**Instantaneous**) or to a range of timesteps (**Period Average**). When **Period Average** is selected, you must enter the time period, in hours, over which the change in elevation applies.

Max Rate of Change—depending on your **Function of** selection…

Constant—you will either be presented with a single field — for a constant value in units of elevation (ft or m) ***per hour***. Figure 11.61 shows this view of the **Flow Rate of Change** rule editor. This view will change to include a field for the period if the **Period Average** option is selected.

Inflow or **Release**—you will either be presented with a table. Enter the relationship between the selected independent variable and the maximum rate of change limit. The limit is described in units of elevation (ft or m) ***per hour***, regardless of the compute interval. Figure 11.62 shows this view of the **Flow Rate of Change** rule editor. This view changes to include a field for the period if **Period Average** is selected.

Interpolate—enter the interpolation type for looking up values in the table. You can choose from the standard options of **Linear**, **Cubic**, or **Step**. See section 11.2.2.2 for an explanation of these options.

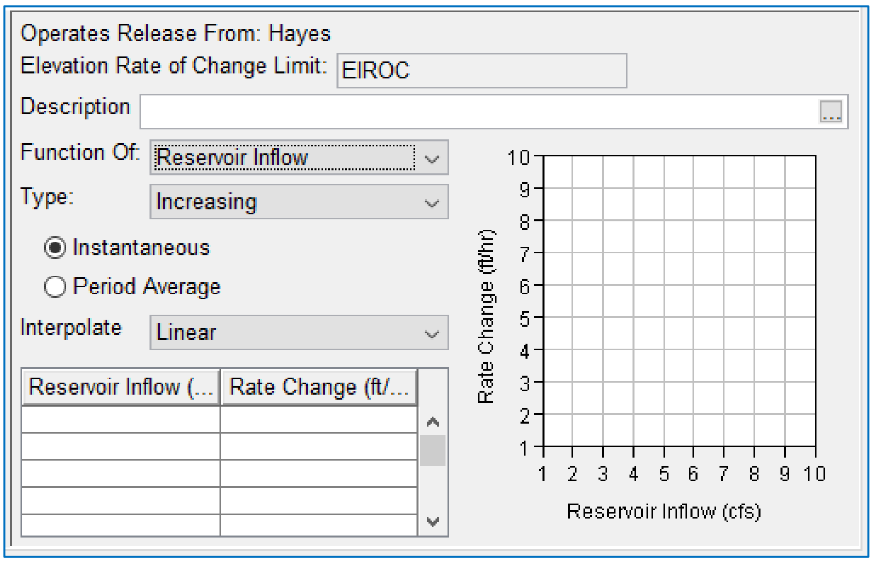


Figure 11.62 Elevation Rate of Change Limit Rule Editor—Function of Time-Series

- When you have finished entering data for the **Elevation Rate of Change Limit** rule, be sure to click **Apply** before moving on to the next rule.

11.7 The Hydropower Rules

Hydropower rules specify a power generation requirement from the power plant at reservoir or a system of reservoirs. When evaluated, the hydropower rules determine the minimum flow that must be released through the power plant to produce the required energy given the plants generating capacity, the hydraulic head, and the generation pattern. The various hydropower rules (Figure 11.63) each provide a different way of specifying the generation requirement:

The **Power Guide Curve** rule allows you to specify the required generation as a function of storage.

The **Schedule** rule allows you to specify the requirement as a function of date.

The **Time Series Requirement** lets you specify the requirement directly through an external time series.

And the **System Schedule** rule allows you to define the generation requirement in the same manner as the **Schedule** rule but the **System Schedule** rule is applied to multiple reservoirs so that together they can operate to meet the generation requirement specified in the rule.

The data requirements for each of these rules will be described in the following sections.

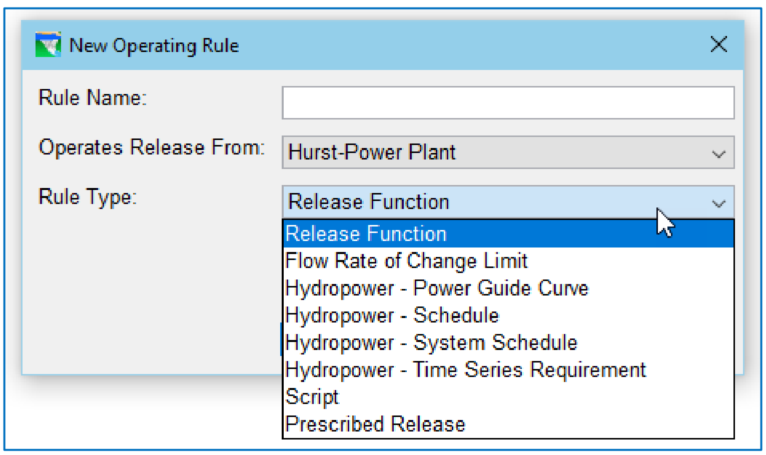


Figure 11.63 Hydropower Rule Types

To define or edit a **Hydropower** rule:

- Create a new rule as described in Section 11.1.1. Be sure to:
  - Select the **Power Plant** as the release element in the **Operates Release From:** selector.
  - Select the appropriate **Hydropower** rule type from the **Rule Type** selector.
- Or, if you want to edit a **Hydropower** rule you have already created, select it from the **Zone-Rules Tree**.
- The edit panel of the **Operations** tab of the **Reservoir Editor** will display the **Hydropower** rule editor for the specific Hydropower rule type. The name and description of the rule will appear in the **Hydropower-*Type* Rule** and **Description** fields.

11.7.1 Hydropower—Power Guide Curve Rule

The **Hydropower—Power Guide Curve** rule allows you to define a function that describes the hydropower generation requirement with respect to the available storage in the *power pool*. The power pool is defined by identifying two top-of-zone curves—one that marks the top power pool and one that marks the bottom.

The power requirement must be described in units of percent of plant factor. Plant factor is a fraction of the capacity of the plant to generate; so, percent plant factor is a percentage of the capacity of the plant to generate. Plant factor can also be interpreted to mean the fraction (or percentage) of the day the plant generates at full capacity. Thus, a plant factor of .25 (or 25%) may mean that the plant generates at 25% of capacity all day long, or the plant generates at full capacity for 25% of the day (or 6 hrs a day). No matter how you phrase it, this specification is a quantity of energy in MWHs. How the plant actually generates is a function of the power generation pattern.

The **Power Guide Curve Rule** editor is shown in Figure 11.64 and its attributes are described below.

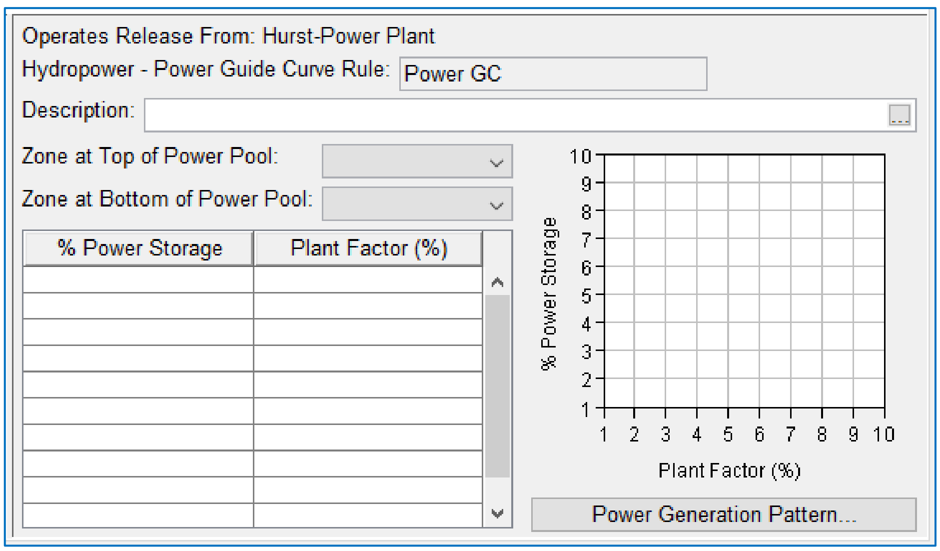


Figure 11.64 Hydropower—Power Guide Curve Rule Editor

Zone at Top of Power Pool—select the zone whose top of zone curve will define the **top** of the *power pool.*

Zone at Bottom of Power Pool—select the zone whose top of zone curve will define the **bottom** of the *power pool*.

Power Guide Curve table—The table in the Power Guide Curve rule editor represents the relationship between storage and the hydropower generation requirement. The first column, the independent variable, is the *percent of storage* available in the power pool. These values should increase as you move down in the table (monotonically increasing). The second column is the *plant factor (in units of percent)*; this is the power requirement as a percentage of the plant's capacity to generate (or the percent of the day the plant should generate at full capacity). The thumbnail plot will display the data in the table.

Power Generation Pattern—use this button to open the **Power Generation Pattern** editor which is described in detail in Section 11.7.5.

11.7.2 Hydropower—Schedule

The **Hydropower**—**Schedule** rule allows you to define a regular *monthly* or user specified *seasonally varying* hydropower requirement. The various options on this rule editor (Figure 11.65) allow you to define each month's or season's power generation requirement, the type of the requirement (megawatt-hours or plant factor), and the hours of the day and days of the week during which the plant can generate.

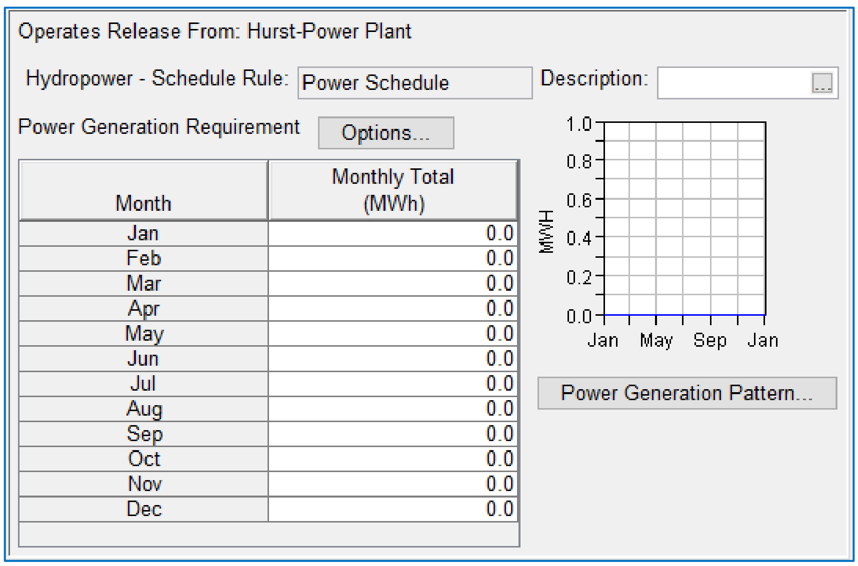


Figure 11.65 Hydropower—Schedule Rule Editor

Power Generation Requirement—click the **Options** button to open the **Power Generation Requirement** editor (Figure 11.66).

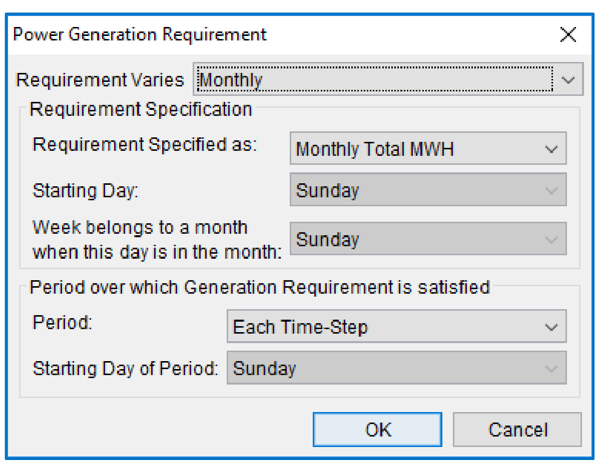


Figure 11.66 Power Generation Requirement Options

Requirement Varies—the option for interval in which you want to specify the requirement. Your options are: **Monthly** and **Seasonally**. The remaining attributes on this editor define what the requirement values mean (Requirement Specification) and how they are to be met (Period over which Generation Requirement is satisfied).

<u>Requirement Specification</u>

Requirement Specified as:—the list of options for describing the values in the Power Generation Requirement table. The options include:

Plant Factor—**Monthly or Seasonal Total**—this means that the requirement values in the table are in units of plant factor and represent a total energy requirement for the Month or Season assuming that the plant ran all the time at the specified fraction of capacity (or ran at full capacity for the specified fraction of the total time). No matter how you phrase it, this specification can be translated into a quantity of energy in MWHs. How the plant actually generates is a function of the power generation pattern.

Daily Total MWH—this option indicates that the values in the requirement table identify how much energy must be generated each day of the month or season.

Weekly Total MWH—this option indicates that the values in the requirement table identify how much energy must be generated each week of the month or season. If you select this option, you must also specify the attributes: **Starting Day** and **Week belongs to a month (or season) when this day is in the month.**

Monthly (or Seasonal) Total MWH—this option indicates that the values in the requirement table identify how much total energy must be generated over the current month (or season).

Starting Day—this is the first day of the week, when Weekly Total is selected for **Requirement Specified as.**

Week belongs to a month (season) when this day is in the month (season)—This field identifies a day of the week that must be in the season so that the whole week can belong to the current month (season) or to the next month (season)—when Weekly Total is selected for **Requirement Specified as.**

For example, if the Requirement is specified as a **Weekly Total** that **varies Monthly**, and the week **Starts** on *Sunday* and the **Week belongs to**…day is *Wednesday*, then if the current month is the one shown in Figure 11.67, then the week that starts on Sunday the 28th belongs to the next month because the following Wednesday is in the next month.

April

Su	Mo	Tu	We	Th	Fr	Sa
	1	2	3	4	5	6
7	8	9	10	11	12	13
14	15	16	17	18	19	20
21	22	23	24	25	26	27
28	29	30				

Figure 11.67 An Example Month

<u>Period over which Generation Requirement will be satisfied</u>

Period—all of the options for **Requirement Specified as** are *period total* values. That means that if your timestep is less than the period total, which it probably is, then the requirement is divided up across the timesteps of the total period based on the power generation pattern, and each timestep is assigned is shared of the total requirement. As each timestep is computed, if that timestep cannot meet its allotted portion of the total requirement, your selection for the **Period over which Generation Requirement will be satisfied** will determine how *or if* the unmet requirement will be satisfied. The **Period** options include:

Each Timestep—with this option, the energy requirement is distributed to all timesteps of the total period based on the power generation pattern. If a given timestep doesn't meet its allotted requirement, the unmet portion will remain unmet.

Daily—with this option, if your timestep is less than 1DAY, then the total energy requirement is distributed to the days across the total period, then across the day to the timesteps of the day—all based on the power generation pattern. If a given timestep cannot meet its allotted requirement, the unmet portion of the day's requirement is redistributed to the remaining timesteps of the day, per the generation pattern. If the remaining timesteps cannot meeting that day's allotted requirement, the requirement will remain unmet.

Weekly—with this option, the total energy requirement is distributed to the weeks across the total period, then the week's allotment is distributed to the timesteps of the week—all based on the power generation pattern. If a given timestep cannot meet its allotted requirement, the unmet portion of the week's requirement is redistributed to the remaining timesteps of the week, per the generation pattern. If the remaining timesteps cannot meeting that week's allotted requirement, the requirement will remain unmet. If you select Weekly, you must also specify the **Starting Day of Period**

Starting Day of Period—this is the starting day of the *week* over which the requirement is to be satisfied. This option is only required if the **Period over which Generation Requirement will be satisfied** is **Weekly.**

Power Generation Requirement table—Complete the power requirement table as appropriate, depending on the options selected **Power Generation Requirement** editor. For example, if **Monthly Total MWH** was selected, then enter the total monthly requirement for each month of the year in the table.

Power Generation Pattern—this button will open the Power Generation Pattern editor. The Power Generation Pattern Is used by the rule to distribute the power requirement to the weeks, days, and timesteps in the period. Please see Section 11.7.5 for a full description of the purpose of the Power Generation Pattern and options for defining it using the Power Generation Pattern editor.

11.7.3 Hydropower—System Schedule

The **Hydropower**—**System Schedule** rule allows you to specify a hydropower generation requirement that a set of reservoirs (a reservoir system) will operate to try to meet. This rule describes the hydropower generation requirement in the same manner as the **Hydropower**—**Schedule** rule (as described above). The System Schedule rule, however, has some *additional options* for specifying the power generation pattern and for identifying reservoirs whose power generation and/or storage can be used to meet the system requirement.

To cause a reservoir to operate to meet a system power requirement, it must have the System Schedule rule in the currently active zone of its operation set. The easiest way to do this is to create the rule in one of the reservoirs, then add the rule to the operation set(s) of the other reservoirs in the system by selecting it from the "Use Existing" list.

The **Hydropower**—**System Schedule** rule editor is shown in Figure 11.68 and its options and attributes are described below:

System Generation Requirement—to specify what values of the system generation requirement table mean, what their units are, and how the requirement is to be met, click the **Options** button to open the **Power Generation Requirement** editor which is the same editor used by the **Hydropower**—**Schedule** rule, so please refer to Section 11.7.2 above for its description.

Power Generation Requirement table—Complete the power requirement table as appropriate, depending on the options selected in the **Power Generation Requirement** editor. For example, if **Monthly Total MWH** was selected, then enter the total monthly requirement for each month of the year in the table.

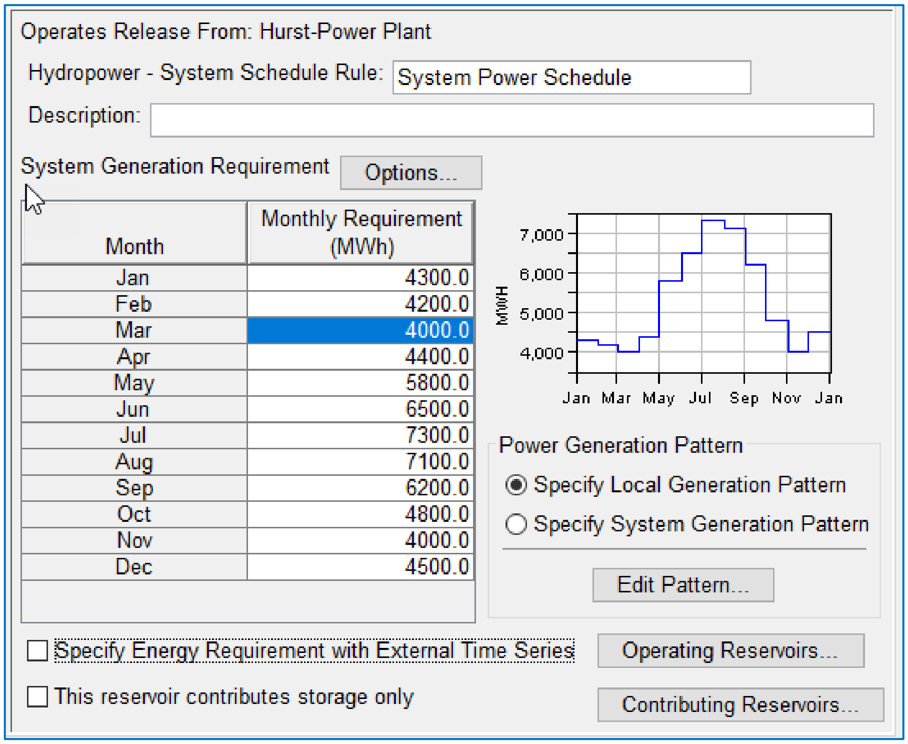


Figure 11.68 Hydropower—System Schedule Rule Editor

Power Generation Pattern—as with the **Hydropower**—**Schedule** rule, the power generation pattern specifies how the power requirement is distributed across the weeks, days, and timesteps of the requirement period. However, with the System Schedule, you can specify for each reservoir (that operates to meet the system requirement) whether the **Power Generation Pattern** that that reservoir should use is specific to that reservoir (**Local**) or is the pattern defined to be shared by one or more reservoirs in the system (**System**). In either case, the **Power Generation Pattern** editor is the same and is described in detail in Section 11.7.5.

Specify Local Generation Pattern—this option allows you to specify a different generation pattern for each reservoir that operates to meet the system power requirement. If you choose this option, you will need to edit the power pattern in the system hydropower rule from each reservoir that includes the rule in its operation set.

Specify System Generation Pattern—this option allows you to specify a single generation pattern that will be used by all the reservoirs that operate to meet the system power requirement.

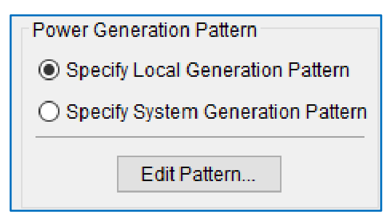


Figure 11.69
Power Generation Pattern

Edit Pattern—use this button to open the **Power Generation Pattern** editor, where you will specify a (local or system) pattern for meeting the system hydropower requirements. Please see Section 11.7.5 for a full description of the purpose and use of the Power Generation Pattern editor.

Specify Energy Requirement with External Time Series—with this option *checked*, you are telling ResSim to use an **External Time Series** to be identified in the alternative as the specification of the Generation Requirement for this rule; this disables the **System Generation Requirement** Options and table. This option makes the **Hydropower—System Schedule** rule very similar to the **Hydropower—Time Series Requirement** rule rather than the **Hydropower—Schedule** rule.

Operating Reservoirs…—use this button to open the **Hydropower-System Rule: Operating Reservoirs** list. This list shows the reservoirs that have this rule in their operation set(s) and are, therefore, operating to meeting the power generation requirement called for by this rule.

Contributing Reservoirs…—use this button to open the **Hydropower-System Rule: Reservoir List** dialog (Figure 11.70). The purpose of this dialog is to allow you to identify those power-producing reservoirs in your network that do not actively operate for this system hydropower requirement but whose generation can be counted toward meeting it. You need not add reservoirs which will *actively* operate for this requirement, they will automatically be added to the "Selected" list when you add this rule to their operation set(s).

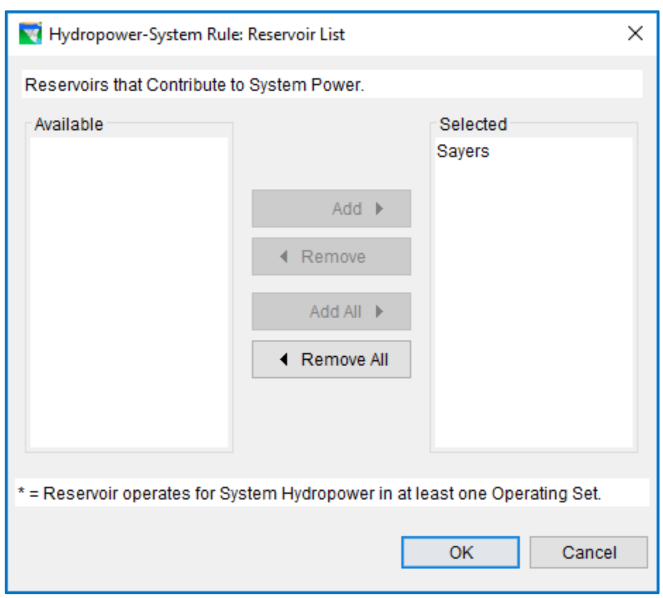


Figure 11.70 Hydropower System Rule—Reservoir List Editor

11.7.4 Hydropower—Time Series Requirement

The **Hydropower—Time Series Requirement** rule allows you to define an *irregular schedule* of Hydropower requirements through the use of a DSS time-series record. Except for the Description, there are options or attributes to be selected or edited in this rule editor, not even a Power Generation pattern since the time series directly indicates the generation requirement for each timestep. Instead, a message is included in the editor (Figure 11.71) to remind you that you will need to specify a time series of required power *in units of megawatts* when you setup an alternative that uses this operation set for this reservoir.

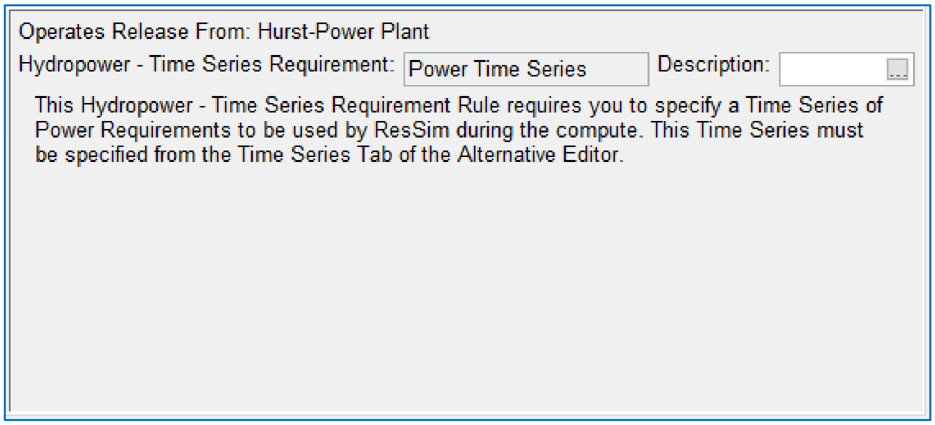


Figure 11.71 Hydropower—Time Series Requirement Rule Editor

11.7.5 Power Generation Pattern

The **Power Generation Pattern** allows you to specify a weekly, daily, and hourly distribution of the specified energy requirements. The pattern covers a week and each day is described with a set of 24-hourly weighting factors. By default, the weekly pattern repeats throughout the year unless you specify a set of seasons and a different pattern for each season.

Weekly Distribution

Three options are available for specifying the weekly power pattern: All Week, Weekdays and Weekend, and Each Day. These options are available from the **Specify Pattern for** list.

All Week—this option applies the same hourly power generation pattern for every day of the week. See Figure 11.72 for an illustration of the editor for this pattern.

Weekdays and Weekend—this option allows you to specify a different power generation pattern for Weekdays (Monday through Friday) than for Weekend days (Saturday and Sunday). See Figure 11.72 for an illustration of the editor for this pattern.

Each Day—this option allows you to specify different power generation patterns for every day of the week. See Figure 11.73for an illustration of the editor for this pattern.

All Week *Weekdays & Weekends*

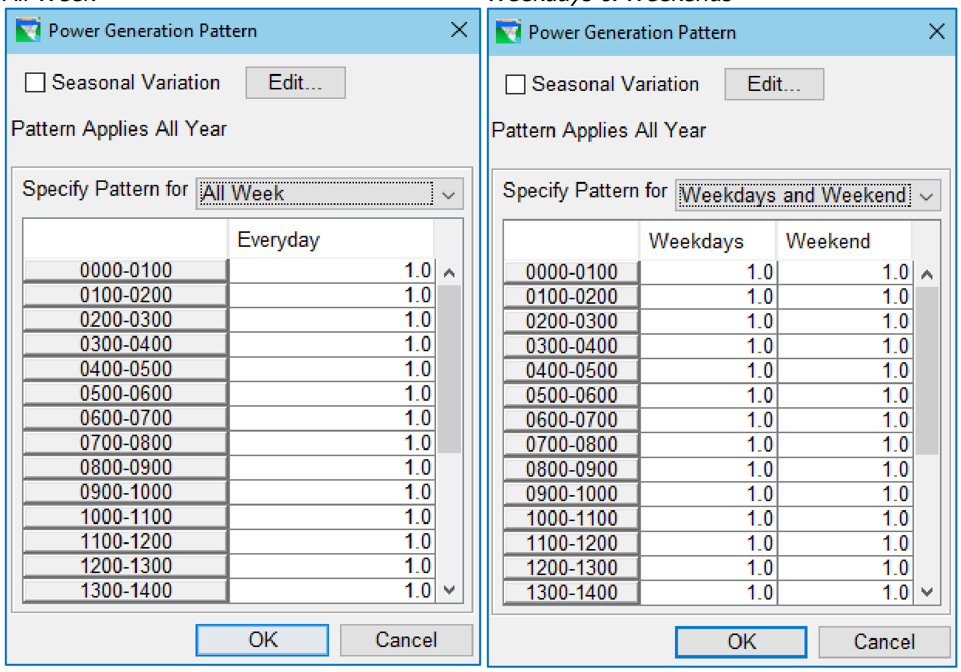


Figure 11.72 Power Generation Patterns—All Week or Weekdays and Weekends

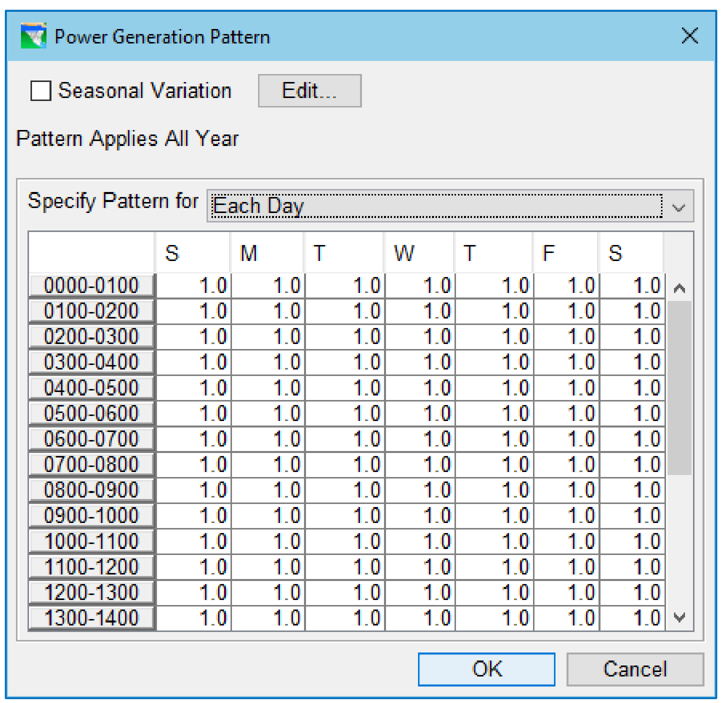


Figure 11.73 Power Generation Pattern (Each Day)

Hourly Distribution

Each day of the **Power Generation Pattern** table is divided into 24 hourly values. Each value represents a weighting factor that you can use to distribute the energy requirement throughout the day and throughout the week. The default value for each hour of the day is 1.0 (Figure 11.72) which will attempt to evenly distribute the energy requirement across the day.

However, if you want the power plant to generate only during a portion of the day, you can set the factor for those hours to 1.0 and set all other hours to 0.0. Basically, the values of 1.0 and 0.0 turn the hourly generation "on" and "off", respectively.

> For example, to specify that generation should *only occur from 8:00 a.m. to 5:00 p.m*. 7 days a week, use the All Week pattern and change the value of 1.0 to 0.0 for hours 0000-0800 and 1700-2400 (Figure 11.74).

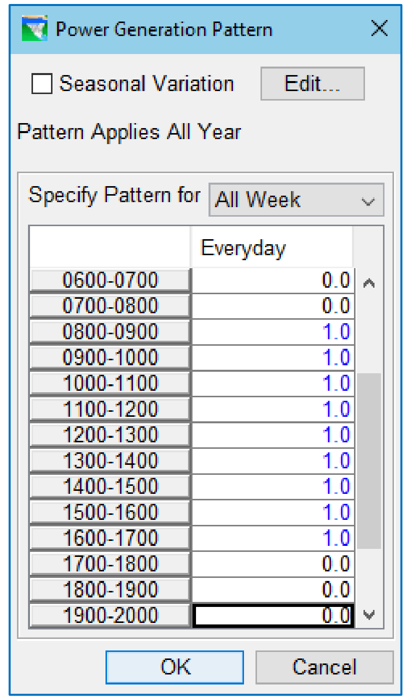


Figure 11.74 Example Generation Pattern—On from 8 am–5 pm

The *weighting factor* aspect of the values in the table comes into play when the values used in the pattern are not just 0's and 1's; in other words, when the generation should be distributed *unevenly* over the "on" hours.

> For example, to indicate that the generation between 10:00 a.m. and 1:00 p.m. should be *twice* the generation of the other generating hours, you could change the value of 1.0 to 2.0 for hours 1000-1300 (Figure 11.75). The sum of the weighting factors for each day in this example is 6(1.0) + 3(2.0) = 12.0. If each day's energy requirement was 12 megawatt-hours, then the "unit" generation requirement would be 1 megawatt (12 MWh/12 units). Thus, from 8am to 10am and from 1pm to 5pm, the plant would generate at a rate of 1 megawatt (1 MW * 1.0 weighting factor), but from 10am to 1pm, the plant would generate at a rate of 2 megawatts (1 MW * 2.0 weighting factor).

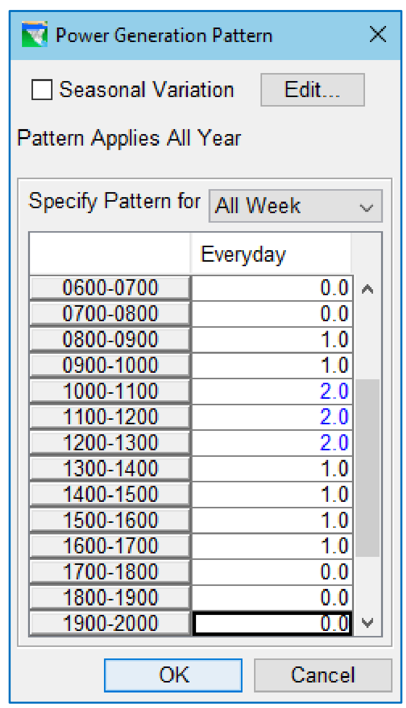


Figure 11.75 Example Pattern—Varied Hour of Day Weighting

Daily Distribution

Another aspect of the *weighting factors* that should be understood is the impact of a pattern that changes from day to day throughout the week. If the pattern is specified for **Each Day** or for **Weekdays and Weekend**, then the weighting factors are summed for the entire week and applied to the generation requirement for the whole week.

For example, if the weekly generation requirement is 20MWh and the daily pattern is for Each Day, then the following power requirement would be specified as shown in Figure 11.76:

- No generation on Tuesday, Thursday, Saturday and Sunday;
- Generation on Monday and Friday is from 10am to 2pm;
- Generation on Wednesday is for the same hours as Monday and Friday but at twice the rate.

In this example, the sum of the weighting factors is 4(1.0) + 4(2.0) + 4(1.0) = 16 which produces a unit generation of 20/16.0 = 1.25. Thus, for each hour on Monday and Friday, 1.25 MWh of energy will be produced for a total of 5 MWh each day and for each hour on Wednesday, 2.5 MWh of energy will be produced for a total of 10 MWh, resulting in the total of 20 MWh for the week.

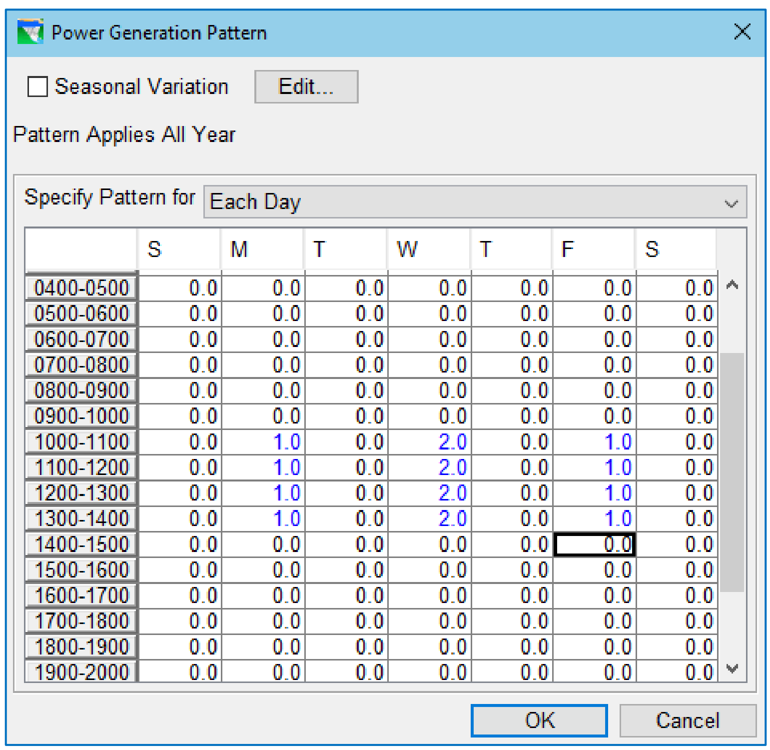


Figure 11.76 Weighting Factors throughout the Days of the Week

Seasonal Variation

If the weekly pattern varies throughout the year, then you can click the **Seasonal Variation Edit…** button to open the **Seasonal Variation** editor (Figure 11.77). This editor will allow to you define the seasons for which different generation patterns apply. To define the seasons, enter the start date of each season. The first season always starts on the first of January, so if you have a season that begins before the end of the year and ends after the first of January, then you must define the first half of the season as the last season in the table and the second half as the first season in the table.

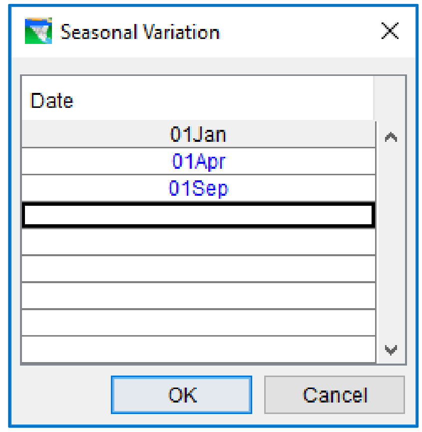


Figure 11.77 Seasonal Variation Editor for Specifcation of Seasons

If you define two or more seasons, the Power Generation Pattern editor will show:

- a checkmark in the checkbox in front of the **Seasonal Variation** label
- a season selector and a set of navigation buttons so that you can specify the pattern that should apply to each season (Figure 11.78).

The power generation pattern you define for one season will apply to all days and weeks of that season until the start date of the next season.

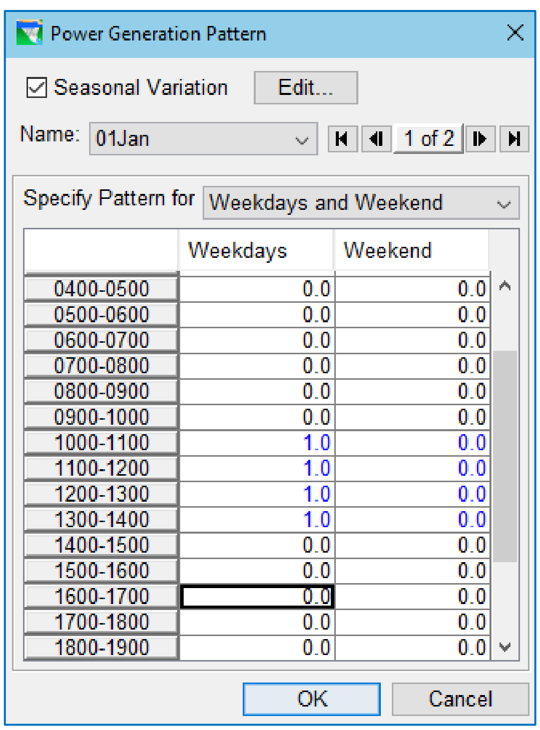


Figure 11.78 Power Generation Pattern Editor with Season Selector

11.8 Defining a Pump Schedule Rule

A **Pump Schedule** rule is an operation rule that specifies the pumping schedule and requirements for a pump outlet. Unlike other outlet types, pump outlets will not "move water" without a rule, and that rule is the **Pump Schedule** rule.

A **Pump** outlet provides a reservoir with the ability to pump water *from a downstream reservoir* into the current reservoir (i.e., pumpback storage operation). This means that the flow direction for water moving through a **Pump** is opposite to the flow direction of any other outlet; a **Pump** moves water into the current reservoir while all other outlets move water out of the current reservoir.

The **Pump Schedule** rule type is only available when a **Pump** outlet is selected as the release element in the **Operates Release From** selector; and, it is the only rule type that can be applied to a **Pump**.

The **Pump Schedule Rule** editor is shown in Figure 11.79. The **Pump Schedule** rule has a number of options and attributes, which are described below; pay careful attention to the description of each option and attribute to learn how each selection you make could change what appears in the editor and how it will affect the way the pump operates.

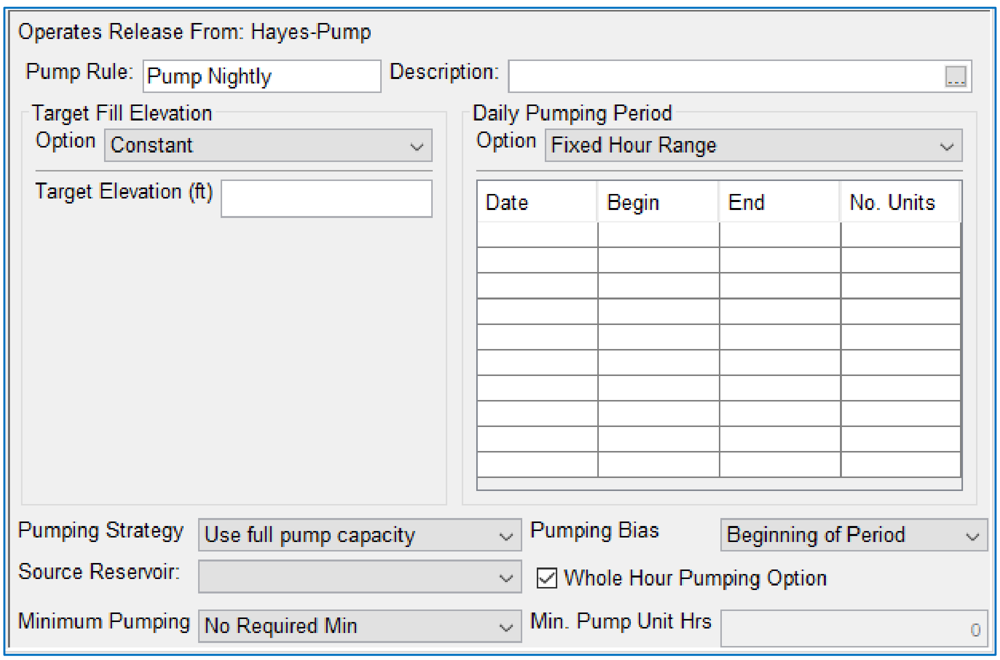


Figure 11.79 Pump Rule Editor

Target Fill Elevation—the pump operation needs to know *why* it should pump. The *why* is the **Target Fill Elevation**. When the Pump Schedule rule is evaluated, its first determination is whether the reservoir pool is below the target elevation. If so, then the rule decides it needs to pump and proceeds to determine if it is *allowed* to pump and how much it should pump.

Option—there are three options (Figure 11.80) for specifying the **Target Fill Elevation**. Each option will change the lower portion of the **Target Fill Elevation** section of the rule editor as you will see below.

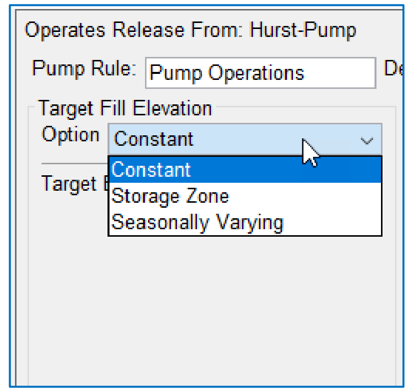


Figure 11.80 Pump Rule Editor—Target Fill Elevation Options

Constant—if selected, a Target Elevation text field will be displayed in the lower portion of the Target Fill Elevation section. Enter an elevation above which the reservoir should *not* pump.

Storage Zone—if selected, a **Zone** selector will be displayed (Figure 11.81). Choose the zone whose top of zone curve represents FULL for the current reservoir.

> Note: each zone in the list of zones shown in the Zone selector is identified by its name and the *operation set* it is defined in. Select the zone associated with the *current* operation set. Unfortunately, this will make the rule applicable to only to the current operation set. If the same rule is needed in a different operation set, you should re-create the rule and select the appropriate zone for that operation set.

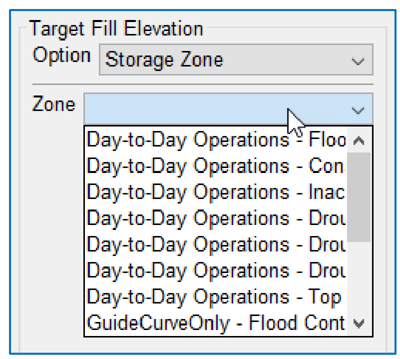


Figure 11.81 Pump Rule Editor—Target Fill Elevation Option—Storage Zone

Seasonally Varying—if selected, a seasonal table relating the Target Fill Elevation to a Date or season will be displayed (Figure 11.82). Enter the data to define the seasonally varying target fill elevation in the table.

Daily Pumping Period—This is the schedule for which the rule gets its name. Since pumps often run only at night when the power to run them is less expensive, a schedule identifying the pumping window is necessary. In this table you must specify the portion of the day during which pumping is allowed. This is a seasonal table that should start on 01Jan, like all other seasonal tables. You have two options for defining the **Daily Pumping Period**:

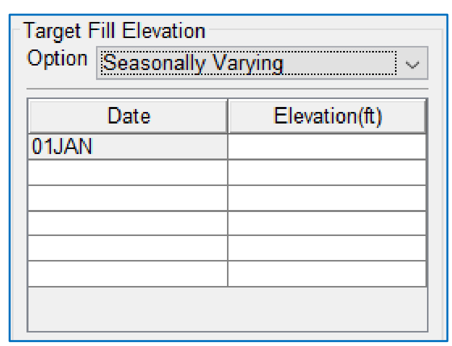


Figure 11.82 Pump Rule Editor—Target Fill Elevation Option—Seasonally Varying

Fixed Hour Range—This option displays a seasonal table in which you must enter the start date of each season, the Begin and End times of that season's pumping window, and the number of pump units that may be used. The times must be entered as times on a 24-hour clock and may cross over midnight. For example, to specify a pumping period starting at 9:00 pm and ending at 3:30 am, enter 2100 and 0330 into the **Begin** and **End** columns, respectively. The data in each row of the schedule table is treated as a step function and remains in effect until the start of the next season.

Between Sunset and Sunrise—If selected, this option displays a seasonal table in which you must define the start and end times of the pumping window for each season as a number of hours after sunset and before sunrise (**After SS** & **Before SR**), respectively. In order for this schedule to determine when sunrise and sunset occur on any particular day, you are required to provide the approximate **Latitude** and **Longitude** of the dam. Note, blank entries for Latitude and Longitude will be translated as 0 values.

Pumping Strategy—The pumping strategy identifies how and when the pumps are to be used during the pumping window, assuming that there is more time available than is needed to get the reservoir to its target elevation. Your options are:

Use full pump capacity—This strategy will minimize the time spent pumping. With this option, you must select the Pumping Bias. Your options include:

Beginning of Period—The pumps will start pumping at the beginning of the pumping window.

Middle of Period—The hours needed to pump to get the reservoir to the target elevation will be centered within the pumping window.

End of Period—the hours needed to pump will be pushed toward the end of the pumping window.

Use entire pump period: This strategy will run the pumps at a rate which may be less than full capacity in order to *just* reach the target at the end of the pumping period.

Regardless of your **Pumping Strategy** selection, if the target cannot be reached in the pumping period, the pumps will be operated at full capacity over the entire pumping period in order to get the reservoir elevation as close as possible to the target.

- Identify the **Source Reservoir**. This should be the next reservoir downstream of the current reservoir although it may be a reservoir *connected* to the current one through a diverted outlet. ResSim's pump outlets are designed to only pump from a storage pool, not directly from a stream. NOTE: This field *should* be defined in the physical data of the pump outlet, not in the operation rule—so don't be surprised if this setting gets moved to where it belongs in a future version of ResSim.
- Determine if **Whole Hour Pumping** should be activated. When checked, if the pumps needed to run for a fraction of an hour, then they will run for the full hour, even if their pumping causes the reservoir to exceed the target pool elevation. Note—ResSim really doesn't want to run the pumps longer than necessary, so if it can, it will

minimize the total number of pump hours. For example, if you have 2 available pumps and the number of pump unit hours needed to get the reservoir to target is 4.5, then the pumps will only run for a total of 5 pump unit hours—one pump will run for 3 hours and the other for 2 hours (or just one pump for 5 hours).

- Select the **Minimum Pumping** requirement. Depending on your selection, this feature identifies the minimum number of hours the pumps must run—*if* they turn on at all during the pumping window. Your options include:

  No Required Min—The default. The pumps will only operate if and for how long they need to in order to get the reservoir pool elevation back up to the target within the pumping window.

  At Least Min—The pumps will operate for at least the specified minimum duration even if the target elevation has been reached or exceeded. With this option, you must enter a value in the **Min. Pump Unit Hrs** field.

  Pump unit hours are the number of hours a single unit operates. If you have 4 pump units and you want all four pumps to operate for at least 1 hour each during the pump window, you might enter 4 in the **Min. Pump Unit Hrs** field. ResSim, however, probably won't interpret your entry as a requirement to run all four pumps. It may turn on just one pump and run it for four hours.

  At Least Min if blw Target—With this option, the pumps will only operate if the reservoir is below the target elevation at the start of the pumping window. And, if they do operate, they will run for at least the specified duration. With this option, you must enter a value in the **Min. Pump Unit Hrs** field.

  Only if Req'd >= Min—With this option, the pumps will operate only if the time needed to pump (at maximum capacity) is greater than or equal to the specified minimum. In other words, if the time needed is less than the min, don't pump. With this option, you must enter a value in the **Min. Pump Unit Hrs** field.

11.9 Defining a Tandem Operation Rule

A **Tandem Operation** rule establishes a tandem system operation in which an upstream reservoir operates for a downstream reservoir to achieve a *storage balance*. Unlike the **Downstream Control** and **System Hydropower** rules, which must be included in the operation sets of all the reservoir that operate together as a system, the **Tandem Operation** rule is created and included in the operation set at the upstream reservoir only. The **Tandem Operation** rule simply identifies the downstream reservoir for which the current reservoir must operate to balance storage. Refer to Chapter 12 to learn more about defining system operations.

Because the tandem operation must account for all releases from the reservoir that could reach the downstream reservoir, the only valid release element for a **Tandem Operation** rule is the reservoir (pool), not an outlet or outlet group.

When evaluated, the **Tandem Operation** rule returns a desired *specified* release. That means the Tandem rule is saying to the allowable range that the release can be no more than AND no less than the release value calculated by the rule. Because a specified release limit is very aggressive, it is recommend that you place the **Tandem** rule at the bottom of the rule stack in each zone it applies to; by doing so, you allow other rules to take precedence over the Tandem rule, hopefully preventing the Tandem rule from "taking over".

To create a **Tandem Operation** rule:

- Follow the instructions for creating a new rule as described in Section 11.1.1.
- Be sure to select the *reservoir* from the **Operates Release From** selector and **Tandem Operation** from the **Rule Type** selector.
- After giving the rule a name, click **OK** to finish creating the new rule and close the **New Operating Rule** dialog. The new rule will appear at the bottom of the rule stack in the selected zone of the **Zone-Rules Tree** and the **Tandem Rule** editor (Figure 11.83) will appear in the edit panel.

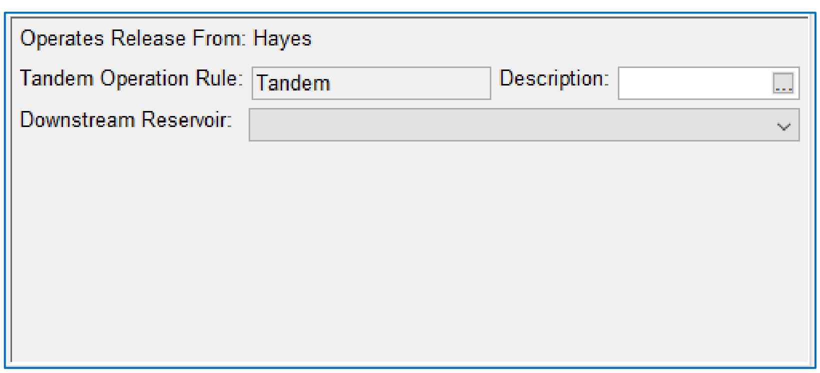


Figure 11.83 Tandem Operation Rule Editor

To define the data for a **Tandem Operation** rule:

- If necessary, select the **Tandem** rule from the **Zone-Rules Tree**. The **Tandem Operation** rule editor will be displayed in the edit panel (Figure 11.83).

- Select the **Downstream Reservoir** for which the current (upstream) reservoir is operating. This is usually the next reservoir downstream of the current reservoir.
- Check the position of your **Tandem** rule with respect to other rules in the zone in which it is placed. If necessary, use the **Move to Bottom** option from the rule's context menu to force the rule to the bottom of the rule stack.

11.10 Defining a Prescribed Release Rule

A Prescribed Release rule is a rule that can be used to "hand regulate" a reservoir. That means that you can tell the reservoir how to operate over a specific time window with a specific operating objective. The operating objective is defined by selecting an **Operator** and an accompanying value. For example, if the Operator is "**Rel % Inflow**", the **Value** is the percentage of inflow that should be released.

Although similar in concept to a **Release** or **Elevation Override**, the **Prescribed Release** rule can be *prioritized* with other rules; while an override is just that—it overrides whatever the rules may have decided was the appropriate release.

To create a **Prescribed Release** rule:

- Follow the instructions for creating a new rule as described in Section 11.1.1.
- Be sure to select an appropriate release element from the **Operates Release From** selector and **Prescribed Release** from the **Rule Type** selector.
- After giving the rule a name, click **OK** to finish creating the new rule and close the **New Operating Rule** dialog. The new rule will appear at the bottom of the rule stack in the selected zone of the **Zone-Rules Tree** and the **Prescribed Release Rule** editor (Figure 11.84) will appear in the edit panel.

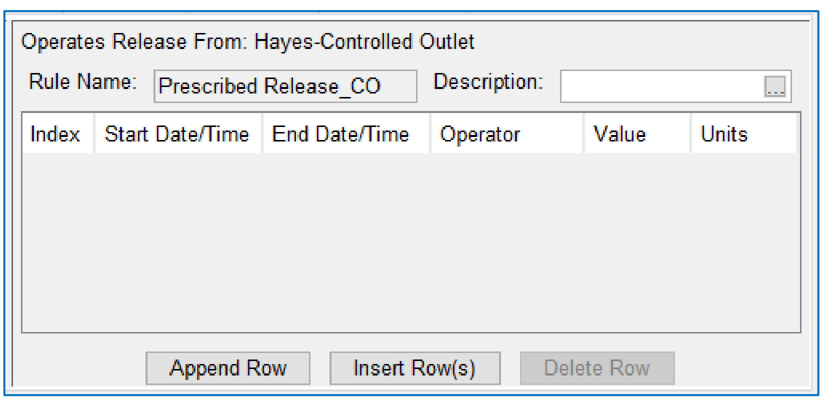


Figure 11.84 Prescribed Release Rule Editor

To define a **Prescribed Release** rule:

- Select the **Prescribed Release** rule from the **Zone-Rules Tree**. The **Prescribed Release** rule editor will be displayed in the edit panel (Figure 11.84).
- Use the buttons at the bottom of the editor to add rows to the table for each prescribed release you want to define.

  Append Row—adds one row at a time to the bottom of the table.

Insert Row(s)—allows you to specify the number of rows to insert above the selected row in the table.

Delete Row—will delete the selected row.

- In each row…
  - Define the time window over which the prescribed release will apply by entering a **Start Date/Time** and an **End Date/Time**
  - Choose an **Operator** to define the type of prescribed release you are defining (Figure 11.85).

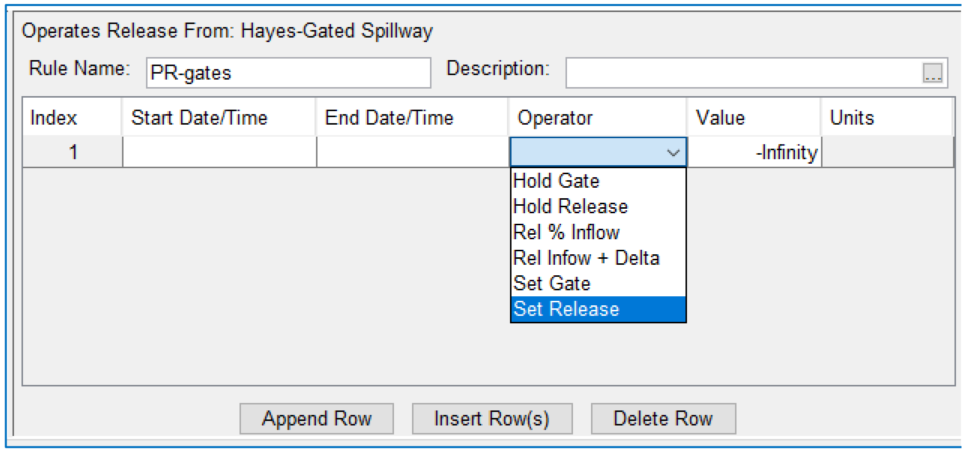


Figure 11.85 Prescribed Release—Setting Operator

The list of available **Operators** will vary depending on the release element you identified for this rule to **Operate Release From**. Table 11.2 shows the list of available **Operators** that you'll find for the different release elements.

Table 11.2 The Available Prescribed Rule Operators by Release Element Type

Reservoir	Dam or Group	Controlled outlet	Controlled Outlet with Gate Settings
Delta Elev	Delta Elev	Hold Release	Hold Gate
Elev ROC	Elev ROC	Rel % Inflow	Hold Release
Elev Target	Elev Target	Rel Infow + Delta	Rel % Inflow
Hold Release	Hold Gate	Set Release	Rel Infow + Delta
Pass Infow	Hold Release		Set Gate
Rel % Inflow	Pass Infow		Set Release
Rel Infow + Delta	Rel % Inflow		
Set Release	Rel Infow + Delta		
	Set Gate		
	Set Release		

  - And, as needed, enter an appropriate **Value** for the selected **Operator** to use.
- Check the position of your new rule with respect to other rules in the zone it is placed. To raise or lower the priority of a rule within the rule list for a particular zone, use the **Increase/Decrease Priority** or **Move to Top/Bottom** options in the rule's context menu or from the **Rule** menu.

When you have finished setting up your **Prescribed Release** rule, be sure to click **Apply** before moving on to the next rule.

Chapter 12
Advanced Features

In Chapter 10 and Chapter 11, you learned the basics about developing an **Operation Set** to guide how your reservoir should make its release decisions. In this chapter, you will learn about the additional features that can be added to your network and to its reservoirs' operation sets to allow you to specify more complex objectives and constraints on the operation of your reservoirs. You will also learn about the network data importer that can be used to import or copy model data from one network to another.

The first part of the chapter describes some advanced tools that you can use to define your reservoir and diversion operations. These tools provide additional flexibility and control over the conditions that activate one or more rules, influence the desired release from a rule (or diversion), or even define a zone. These tools are **IF_Blocks**, **State Variables**, and **Scripted Rules**.

The next part of the chapter describes the advanced operational features that can be added to the operation set of a reservoir. These features include: **Capacity Outages**, **Storage Credit**, **Release Decision Schedule**, and **Projected Elevation**. Each of these features has a unique influence on the overall reservoir operation; their influence is not zone-based and doesn't fit into the zone-rules portion of the operation set. These features are specified on the remaining sub-tabs of the **Reservoir Editor**'s **Operations** tab; but each tab must be activated by selecting the feature from the **Operations** menu of the **Reservoir Editor**.

The last part of this chapter steps back from the reservoir operations to describe the two network properties ***importers*** that are available in ResSim. One of these importers will copy a whole reservoir network from another watershed into the current watershed. The other importer will copy data from an existing network into the current (open) network.

12.1 IF_Blocks

Reservoir operations can be influenced by several conditions that determine the applicability of specific operational goals and constraints. So far, you have learned that, in ResSim, the determination of the applicable set of rules is related to the prioritization and presence of the rules in the active operation zone.

Using the ResSim **IF_Block** feature, you can define one or more conditional expressions that determine which rules should be used in the reservoir release decision process. IF_Blocks are highly customizable. You can create simple **IF_Blocks** that contains only a single **IF** statement or you can build more complex **IF_Blocks** by adding one or more **ELSE IF** statements and/or an **ELSE** statement. For each **IF**, **ELSE IF**, or **ELSE** statement used in an **IF_Block**, you can associate a prioritized set of rules that is only applicable when its conditional expression evaluates to *TRUE*.

12.1.1 IF_Block Terminology

To understand how to create and use **IF_Blocks**, you should understand the terminology that was developed related to IF_Blocks. Below are some important terms and their definitions:

IF_Block—a feature that you can add to a zone in which you can define one or more conditional expressions that, when evaluated to a value of TRUE, activates one or more operation rules. An **IF_Block** contains an ordered list of conditional blocks.

A complex (or compound) conditional expression strings together two or more simple conditional expressions using the logical operators: AND and OR.

Conditional Block—an object made up of a conditional expression and a rule set that will be activated when the conditional expression evaluates to true.

IF, **ELSE IF**, and **ELSE** are the three *types* of conditional blocks that may be members of an **IF_Block** in ResSim. By default, all **IF_Blocks** are created with an **IF** in their list of conditional blocks; you can add one **ELSE** and one or more **ELSE IF**s to the IF_Block list.

Conditional Expression—a logical or comparative statement that can be evaluated to TRUE or FALSE. A simple conditional expression compares two objects or values using one of the comparison (or Boolean) operators: **=**, **>=**, **>**, **<=**, **<**, and **!=**.

IF—one of the three types of conditional blocks that may be found in an **IF_Block**.

> For simplicity, an IF conditional block will usually be referred to in this document as an IF. The feature that an IF *is contained in* will always be referred to as an IF_Block.
> And, for consistency, the ELSE and ELSE IF conditional blocks will also be referred to as just ELSE and ELSE IFs.

An **IF** conditional block is always created as part of an **IF_Block**; it cannot be deleted nor can another **IF** be added. The **IF** is always *first* in the list of conditional blocks that belongs to an **IF_Block** and, as such, its conditional expression is always evaluated first. If the **IF** block's conditional expression evaluates to true, the **IF**'s rule set will be added to the zone's set of active rules and the conditional expressions for the remaining conditional blocks contained in the *parent* **IF_Block** will be evaluated; however, if there are any *nested* **IF_Blocks** in the **IF** block's rule set, the conditional expression evaluation will move into the nested **IF_Block**(s).

ELSE IF—one of the three types of conditional blocks that may be found in an **IF_Block**. If an **ELSE IF** has been added to an **IF_Block**, the **ELSE IF** is evaluated only when the conditional expression of the preceding **IF** evaluated to FALSE. If more than one **ELSE IF** has been added to the **IF_Block**, the **ELSE IF**s will form an ordered list between the **IF** block and the **ELSE** block (if there is one); the conditional expressions of the **ELSE IF** blocks will be evaluated in order, from top down, until one of them evaluates to

TRUE, at which point the evaluation of the conditional expressions will cease and the rule set of the *TRUE* **ELSE IF** block will be added to the current zone's list of active rules. If the conditional expressions of all the **ELSE IF** blocks evaluate to FALSE, the rules in the **ELSE** block (if there is one) will be added to the current zone's list of active rules.

ELSE—one of the three types of conditional blocks that may be found in an **IF_Block**. Only *one* **ELSE** may be added to an **IF_Block**. When added, the **ELSE** will always be placed at the *bottom* of the ordered list of conditional blocks in the **IF_Block**. Although an **ELSE** does *not* have a conditional expression that you can specify, think of it as having a conditional expression that is always TRUE. As a result, an **ELSE**'s rule set will be activated when the conditional expressions of the **IF** and all the **ELSE IF**s that precede it evaluate to FALSE.

Nested **IF_Block**—this is an **IF_Block** that is added to the rule set of a conditional block belonging to a higher-level (or parent) **IF_Block**.

A ResSim Truth: *wherever you can add a rule, you can add an IF_Block*.

Nested **IF_Blocks** can be useful, especially when you need to conditionally activate the same rule set in multiple conditional blocks of a single **IF_Block** or in multiple other **IF_Blocks**. However, do not use a nested **IF_Block** as a way to avoid defining a conditional block with a compound or complex conditional expression and/or using multiple **ELSE IF** blocks.

Rule Set**—**the prioritized list of rules and **IF_Blocks** belonging to a zone or conditional block.

During the compute of each timestep and after the current zone has been determined, ResSim needs to assemble the zone's list of active rules *before* it can evaluate those rules. To do so, ResSim processes each member of the zone's rule set, from top to bottom—if a member of the rule set is a…

Rule—the rule is added to (the bottom of) the active rule list.

IF_Block—ResSim evaluates the conditional expression of each conditional block in the **IF_Block** until it encounters one that evaluates to TRUE, at which point ResSim processes the members of the conditional block's rule set, adding its rules to the zone's active rule list and evaluating its **IF_Blocks**.

12.1.2 Managing IF_Blocks

Since you can add an **IF_Block** wherever you can add a rule, the functions for managing them are similar to those for managing rules. You can create, rename, and delete **IF_Blocks**, as well as add or remove existing **IF_Blocks** from one or all zones. But there's one additional function that is available for **IF_Blocks** that is not available for rules—*duplicate*.

Most of the **IF_Block** management functions are available from the **IF_Block** menu in the Menu bar of the **Reservoir Editor** when the **Operations** tab is active. And, except for **New… and Delete…**, the same functions are available from the **IF_Block** context menu (see Figure 12.1). Because of this duplication, the following sections will focus on the use of the context menu; however, you should use the menu you are most comfortable with—the context menu from the tree nodes or the menu in the menu bar. If you are more comfortable with the menu in the **Menu** bar, replace the words "*right-click* on" with "select" and "context menu" with the appropriate menu for the operation (**IF_Block**).

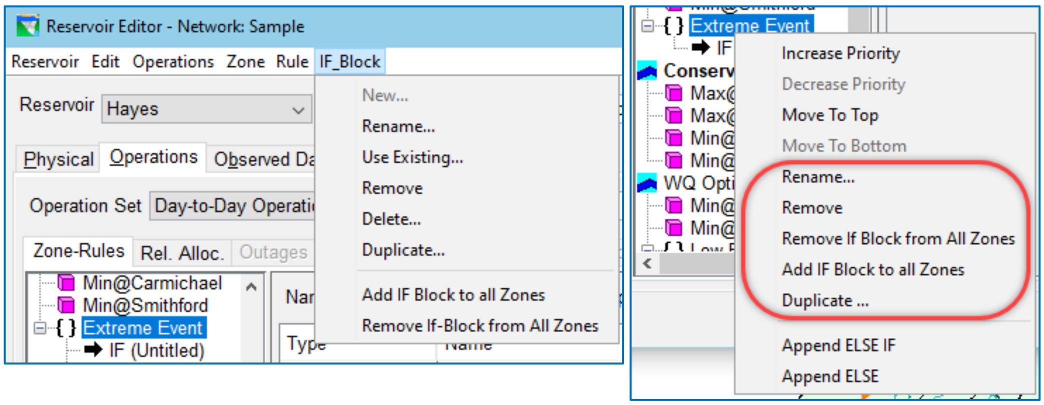


Figure 12.1 Reservoir Editor—Operatons Tab—IF_Block Menu and Context Menu

The **Zone** menu and the zone context menu include the remaining the **IF_Block** management functions. These menus and their **IF_Block** functions are shown in Figure 12.2.

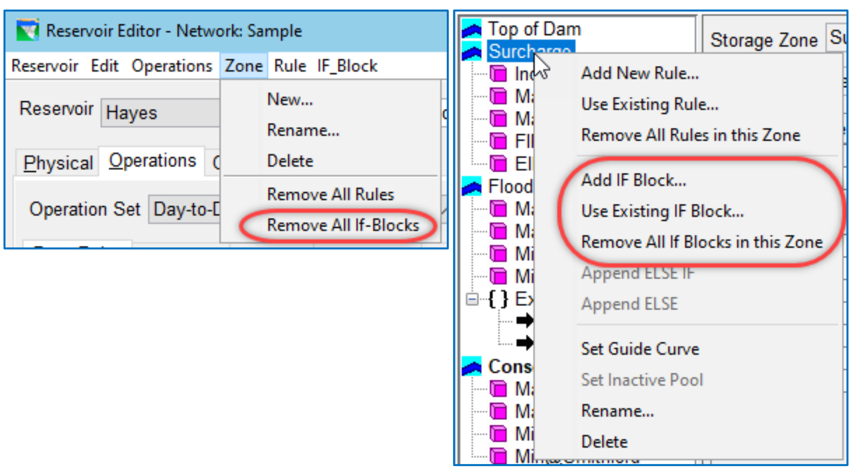


Figure 12.2 Reservoir Editor—Operatons Tab—Zone Menu and Context Menu

The contents in the **IF_Block** and **Zone** menus are sensitive to the currently selected node in the **Zone-Rules** tree. For example, the **New...** function in the **IF_Block** menu is only active when the currently selected node in the tree is a zone or conditional block.

Sections 12.1.2.1—12.1.2.6 describe each of the **IF_Block** management features. Editing **IF_Blocks** is covered in Section 12.1.3.

12.1.2.1 Creating an IF_Block

To create a new **IF_Block**:

- Open the **Reservoir Editor**, select the **Operations** tab, and select the appropriate operation set.
- *Right-click* on a zone in the **Zone-Rules** tree in which you want to place the new **IF_Block.**
- Select **Add IF_Block** from the zone's context menu *or* select **New** from the **IF_Block** menu. The **New IF_Block** dialog will open (Figure 12.3).

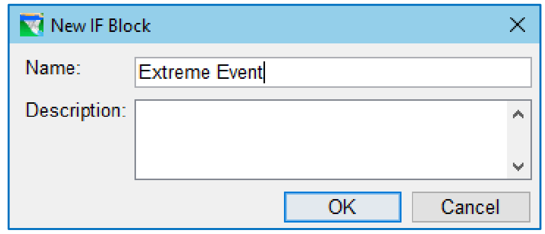


Figure 12.3 New IF_Block Dialog

- Enter a name (and optional description) for the new **IF_Block** and *click* **OK**.
- A *branch* in the **Zone-Rules** tree will appear at the bottom of the selected zone (Figure 12.4). This branch represents the **IF_Block** you created. The root node of the branch will display the **IF_Block**'s name preceded by a pair of curly brackets { }. Below the root node of the **IF_Block**, indented, is the node for the **IF** conditional block belonging to the **IF_Block**; this node displays the word **IF** preceded by a thick black arrow ➡ and followed by a label in parentheses () and. *Untitled* is used as the default label to remind you to give each conditional block a more appropriate label—one that describes the condition(s) defined in the conditional expression or the set of operations that will be added if the conditional expression evaluate to TRUE.

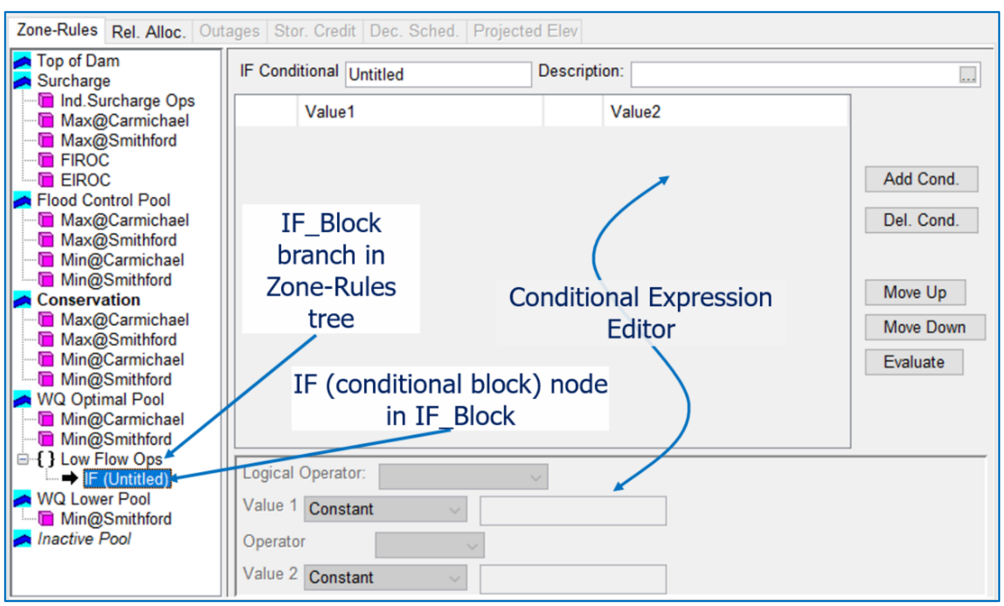


Figure 12.4 New IF_Block and the Conditional Expression Editor

- When the new **IF_Block** is added to the **Zone-Rules** tree, its **IF** becomes the *selected* node and the **Conditional Expression Editor** for the **IF** is displayed in the Edit Pane (Figure 12.4). Section 12.1.3.3 describes how to use the

Conditional Expression Editor for defining the conditional expressions of your **IF** and **ELSE IF** conditional blocks.

12.1.2.2 Using an Existing IF_Block

Like a rule, an existing **IF_Block** can be added to the rule set of any zone or conditional block.

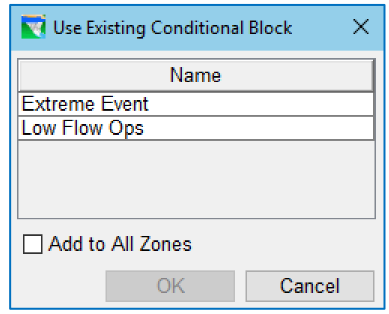


Figure 12.5 Use Existing IF_Block Dialog

To add an existing **IF_Block** to a zone:

- *Right-click* on the zone in the **Zone-Rules** tree.
- Select **Use Existing IF_Block...** from the node's context menu.
- The **Use Existing Conditional Block** dialog will open (Figure 12.5). Select the **IF_Block** you want from one of the white rows in the table and *click* **OK**. [Grey rows in the table are **IF_Blocks** that are already being used in the rule so are not available to be added again.]
- The **IF_Block** will appear at the bottom of the selected zone's rule set.

To add the **IF_Block** to *all* the zones in the current operation set:

- Activate the checkbox in front of **Add to All Zones** before *clicking* **OK** in the **Use Existing Conditional Block** dialog—OR—
- *Right-click* on the IF-Block in the Zone-Rules tree and select Add IF_Block to all Zones from the context menu.

12.1.2.3 Copying an IF_Block

Making a copy of an **IF_Block** is not the same as using an existing **IF_Block**. When you use an existing **IF_Block** in multiple zones, a change made to the **IF_Block** in one zone will show up in all instances of the **IF_Block**. However, when you make a copy of an **IF_Block**, you give the copy a new name and the copy can now evolve independently of the original **IF_Block** it was copied from.

Two methods are available for making a copy of an IF_Block; these methods differ depending on where you start:

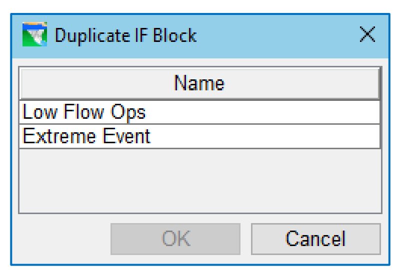


Figure 12.6 Duplicate IF_Block Selection Dialog

<u>From a **Zone** in the **Zone-Rules** tree:</u>

- Select a zone in the **Zone-Rules** tree.
- From the **IF_Block** menu, select **Duplicate.**
- The **Duplicate IF_Block** selection dialog will open (Figure 12.6). Select the **IF_Block** you want to copy from the list and click **OK**.
- The **Name Duplicate Conditional** dialog will open (Figure 12.7). Enter a new name for the new **IF_Block** and click **OK**.
- The new (copied) **IF_Block** will appear in at the bottom of the selected zone.

From an **IF_Block** in the **Zone-Rules** tree:

- Select an **IF_Block** in the **Zone-Rules** tree.
- From the **IF_Block** menu, select **Duplicate**.
- The **Name Duplicate Conditional** dialog will open (Figure 12.7).
- Enter a new name for the new **IF_Block** and click **OK**.
- The new (copied) **IF_Block** will *not* appear in the **Zone-Rules** tree, but it *will* be available for use from the **Use Existing IF_Block** dialog.

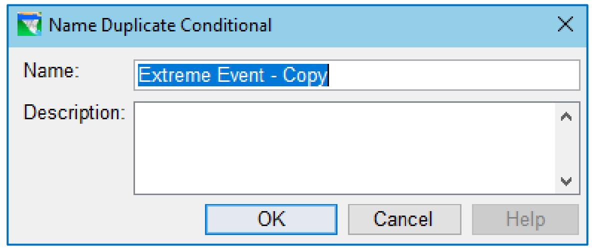


Figure 12.7 Name Duplicate IF_Block Dialog

12.1.2.4 Prioritizing an IF_Block

Since **IF_Blocks** are intended to *conditionally* add a set of rules into the list of active rules for a zone, **IF_Blocks** can and should be prioritized with respect to the other rules and **IF_Blocks** in a zone's (or conditional block's) rule set. The context menu of an **IF_Block** (Figure 12.8) contains the move functions needed for prioritizing an **IF_Block** within a rule set:

Increase Priority—use this function to move the whole **IF_Block** above the rule or **IF_Block** right above it in the rule set.

Decrease Priority—use this function to move the whole **IF_Block** below the rule or **IF_Block** right below it in the rule set.

Move to Top—use this function to move the whole **IF_Block** to the top of the rule set.

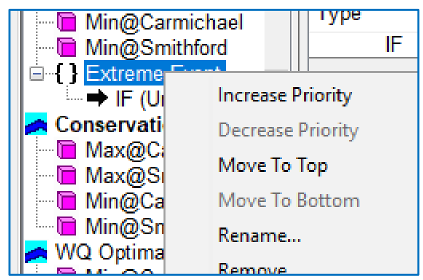


Figure 12.8 IF_Block Context Menu—Move/Prioritize Functions

Move to Bottom—use this function to move the whole **IF_Block** to the bottom of the rule set.

12.1.2.5 Removing an IF_Block

Removing an **IF_Block** from the rule set of a zone or conditional block is the same as removing a rule. To do so:

- *Right-click* on the **IF_Block** node in the **Zone-Rules** tree
- Select **Remove** from the context menu.
- The **Remove IF_Block** confirm dialog will open (Figure 12.9).
- Click **OK** to proceed with removing the selected **IF_Block** from the rule set.

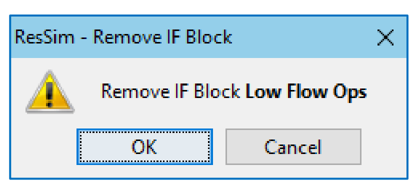


Figure 12.9 Remove IF_Block Confirm Dialog

To remove an **IF_Block** *from all zones* in the current operation set:

- *Right-click* on the **IF_Block** node in the **Zone-Rules** tree
- Select **Remove IF_Block from All Zones** from the context menu.
- The **Remove IF_Block** confirm dialog will open (Figure 12.10).
- Click **OK** to proceed with removing the selected **IF_Block** from the rule sets of all the zones of the current operation set.

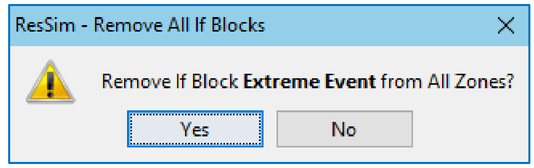


Figure 12.10 Remove IF_Block from All Zones Confirm Dialog

To remove *all* **IF_Blocks** *from a zone* in the current operation set:

- *Right-click* on the zone in the **Zone-Rules** tree
- Select **Remove all IF_Block from this Zone** from the context menu.
- The **Remove All IF_Blocks...** confirm dialog will open (Figure 12.11).
- Click **OK** to proceed with removing the selected **IF_Block** from the rule sets of all the zones of the current operation set.

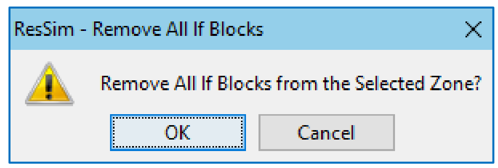


Figure 12.11 Remove All IF_Blocks... Confirm Dialog

12.1.2.6 Deleting an IF_Block

Deleting an *IF_Block* removes all uses of the **IF_Block** from the reservoir's operation sets, even if the **IF_Block** is nested inside another **IF_Block**. After removing all reference to the IF_Block in the operation sets, the **IF_Block** is then deleted from the reservoir. Since **Delete** is such an encompassing operation, it is only available from the **IF_Block** menu.

To delete an **IF_Block**:

- Select an **IF_Block** in the **Zone-Rules** tree. [Any **IF_Block** will do in order to activate the **Delete** option in the **IF_Block** menu.]
- The **Select Conditional Block to Permanently Delete** dialog will open (Figure 12.12).
- Select the **IF_Block** you want to delete from the list of existing **IF_Blocks** and click **OK**.
- A **Delete IF_Block** confirm dialog will open (Figure 12.13). Click **Yes** to complete the delete process. **No** will abort the delete process.

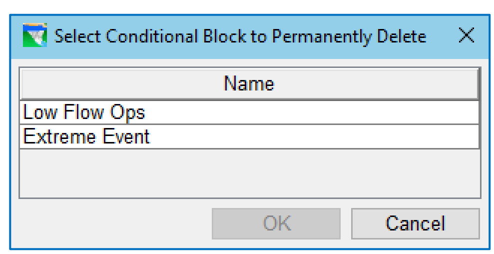


Figure 12.12 Delete IF_Block Dialog

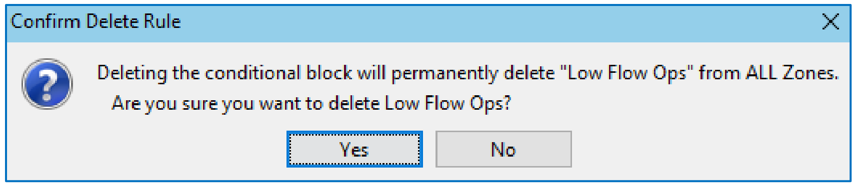


Figure 12.13 Delete IF_Block Confirm Dialog

12.1.3 Editing IF_Blocks

There are three steps to editing **IF_Blocks**, although they can be addressed in almost any order:

- Adding (and/or Deleting) **ELSE IF** and **ELSE** Conditional Blocks
- Defining the Conditional Expressions of the Conditional Blocks
- Specifying the Rule Set for each Conditional Block

The following sections describe these steps.

12.1.3.1 Adding a Conditional Block to an IF_Block

Since an **IF** conditional block is created by default as part of the **IF_Block** and cannot be deleted, only **ELSE** and **ELSE IF** conditional blocks can be added to an **IF_Block.** The functions for adding **ELSE** and **ELSE IF** conditional blocks are available from the context menus of an **IF_Block** and a conditional block; they are *not* available from the **IF_Block** menu in the Menu bar. These functions are:

Append ELSE—this function will add an **ELSE** conditional block to the bottom of the list of conditional blocks in the **IF_Block**. If the **IF_Block** already has an **ELSE**, the **Append ELSE** function will be unavailable (greyed-out) in the context menu.

Append ELSE IF—this function will add a new **ELSE IF** conditional block to the *bottom* of the list of **ELSE IF** blocks in the **IF_Block**, even if you selected **Append ELSE IF** from the context menu of a specific conditional block in the **IF_Block**.

12.1.3.2 Deleting a Conditional Block from an IF_Block

To delete an **ELSE** or **ELSE IF** from an IF_Block:

- *Right-click* on the **ELSE** or **ELSE IF** conditional block to be deleted.
- The **Remove Rule** confirm dialog will open (Figure 12.14).
- Click **OK** to confirm that you want to delete the selected conditional block or **Cancel** to abort the delete process.

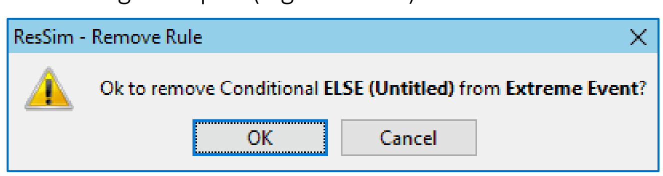


Figure 12.14 Delete ELSE (or ELSE IF) Confirm Dialog

12.1.3.3 The Conditional Expression Editor

The **Conditional Expression Editor** (Figure 12.15) is displayed in the Edit Panel of the **Zone-Rules** tab whenever the selected node in the **Zone-Rules** tree is an **IF** or **ELSE IF** conditional block of an **IF_Block**.

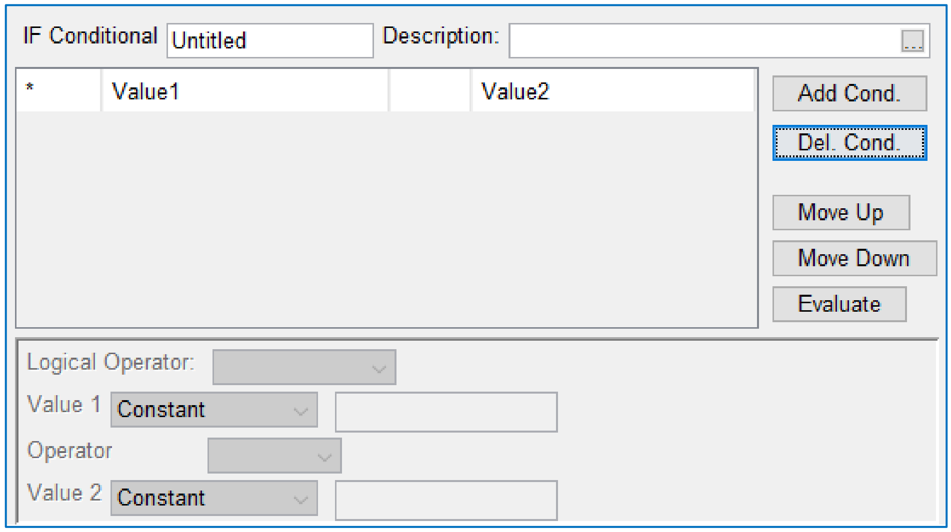


Figure 12.15 Conditional Test Editor—Pick Value

The **Conditional Expression Editor** is divided into three parts—a name and description row and two panels:

The name field is labeled with the conditional type (e.g., IF Conditional). Unlike the rule editors, this name field is editable. Enter a name or label for the conditional block in this name field. Each conditional block was given the label *Untitled* by default in order to remind you to give each conditional block a more appropriate label. Use a label that describes the condition(s) defined in the conditional expression or the set of operations that will be added if the conditional expression evaluates to TRUE. And, if you change the conditions, be sure to change the label.

The **upper panel** gives you the ability to create one or more **conditions** or comparisons. Each **condition** is a row in the table. A single condition (row) forms a *simple* conditional expression. Two or more conditions (rows) can be joined with logical operators to form a *compound* conditional expression.

The function buttons to the right of the table are for managing the rows, in the table:

Add Cond.—adds a row to the bottom of table. Since the table starts out empty, you must add at least one row to specify a simple conditional expression.

Del. Cond.—deletes the selected row from the table.

The order in which conditional expressions are evaluated is similar to that for mathematical expression. The evaluation order for conditional expressions follow these three basic principles:

- left to right,
- comparison operators before logical operators, and
- if parentheses are used, innermost parenthetical expressions first, moving out from there.

Since the order of the conditions in a compound conditional expression may be important when it is evaluated, functions to rearrange the order of the conditions in the table are provided:

Move Up—swaps the selected row with the row above it.
Move Down—swaps the selected row with the row below it.

When using these functions, think of the first row in the table as the left-most comparison in the compound conditional expression and the last row as the right-most comparison. Like the thumbnail plots in the rule editors, the last function button will help with visualizing what you define in the table:

Evaluate—assembles the rows in the Conditions table into a string and displays the string in a message box so that you can verify that your table entries describe the conditional expression you intended.

The **lower panel** is for specifying each condition that forms the conditional expression. A **condition** is formed by two operands, **Value 1** and **Value 2**, and the **comparison operator** between them:

Value 1 and **Value 2**—The two operands can be defined as being one for the variable types listed in Table 12.1. Table 12.1 indicates which variable types can be used by each operation and the data required to define them.

Table 12.1 Available Variable Types and Their Required Data

Value 1	Value 2	Data Required
Constant	**Constant**	Enter a numeric value in the text field
Date/Time	**Date/Time**	Enter a specific calendar date (e.g., 31Oct1949) and a clock time (0000—2400) in **Date** and **Time** fields. A calendar tool can be opened using the button in the **Date** field to assist in specifying a date.
Current Time Step	**Current Time Step**	- No user input required -
Time Series	**Time Series**	Use the **Pick Value** button to open the Independent Variable Definition Editor. Use it to select the specific model variable, state variable, or external variable you need.
Seasonal	**Seasonal**	Enter a specific day of the year (e.g., 01Apr) in the **Date** field.
	Time of Day	Enter a clock time (0000—2400)
	Day of Week	Enter a weekday (e.g., Sun, Mon, etc.)

The ***operators*** that can be used to compare **Value 1** to **Value 2** are listed in Table 12.2.

Table 12.2 The Comparison Operators

=	equal to
>	greater than
>=	greater than or equal to
<	less than
<=	less than or equal to
!=	not equal to

Logical Operators—**AND** and **OR**. Each condition after the first must be joined to the prior condition(s) with a logical operator.

You can include parentheses in your conditional expression by setting **Value 1** to the required parenthesis. If you set **Value 1** to an open parenthesis (**(**), the operator and **Value 2 selectors** will be greyed-out (deactivated); however, you must specify the logical operator that precedes the parenthesis. If you set **Value 1** to a close parenthesis (**)**), all other fields of the lower panel will be greyed-out (deactivated) since the closed parenthesis forms the end of a condition. Figure 12.16 shows an example of a compound conditional expression and illustrates the use of parentheses and logical operators.

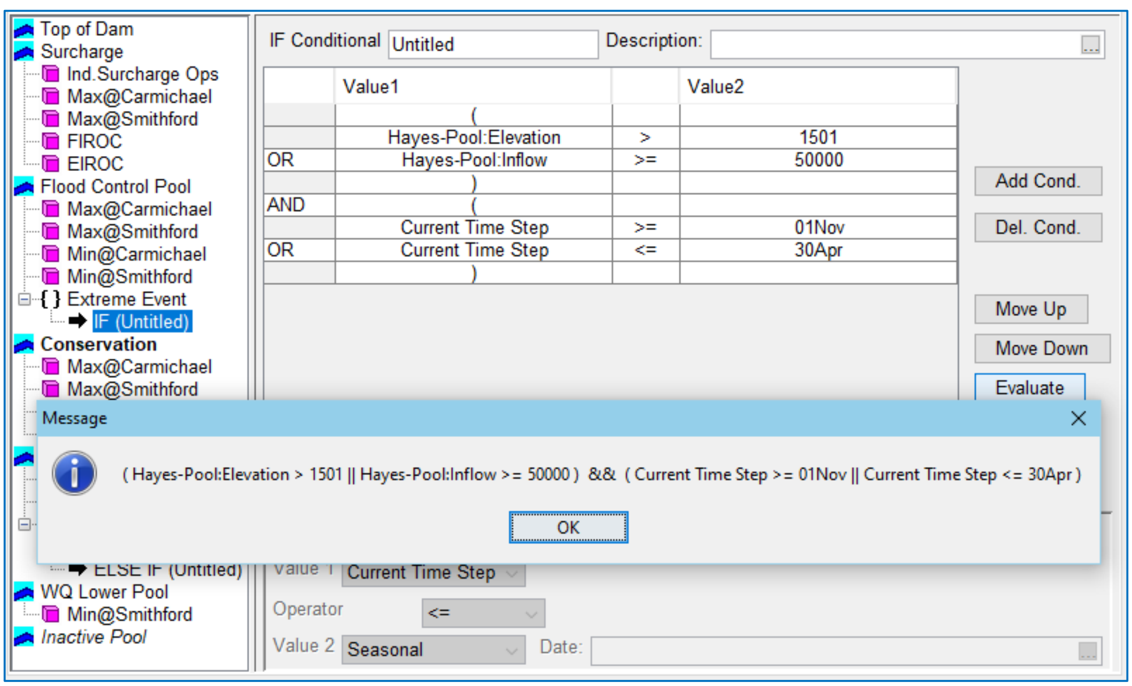


Figure 12.16 Example of a Compound Conditional Expression and the Associated Evaluate *String*

12.1.3.1 Defining the Rule Set of a Conditional Block

The steps for adding rules and **IF_Blocks** to the rule set of a conditional block is the same as adding them to a zone. The functions you need are all available from the context menu of a conditional block in the **Zone-Rules** tree (Figure 12.17).

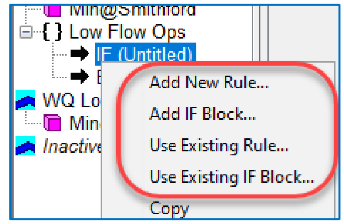


Figure 12.17 Conditional Block Context Menu

To add a new or existing rule to the rule set of a conditional block:

- *Right-click* on the conditional block (**IF**, **ELSE IF** or **ELSE**) in the **Zone-Rules** tree.
- Select Add New Rule... or Use Existing Rule... from the context menu
- For a *new* rule, select the release element, set the rule type, and give the rule a name. For an *existing* rule, select the rule from the **Use Existing Rule** dialog. *Click* **OK** to complete the add rule process.

Figure 12.18 illustrates a completed conditional block rule set.

Figure 12.18 Completed Conditional Block with Rule Set

12.2 State Variables

Definition: A state variable is one of a set of variables used to describe the state of a dynamical system. Since a watershed is a dynamic system, this definition applies to all the variables that can be computed for an object or a group of objects in the watershed as it varies through time.

ResSim places state variables into three categories:

Model Variables—state variables that ResSim computes natively for each element in a ResSim network. Examples include: reservoir pool elevation, release from an outlet, or flow through a diversion. A Model Variable is computed for each timestep of the simulation and stored in a *Time Series object*.

External Variables—variables that have been computed externally (outside of ResSim) and provided as input time series. Examples include: a time-series of forecasted snowmelt inflow volume or water supply demand. An External Variable is also stored in a *Time Series object.*

(User-Defined) State Variables—variables that a user defines by writing a Jython script that will be used to compute the value of the variable for each timestep of the simulation. A State Variable is an object that belongs to a reservoir network and contains a *Time Series object* that holds the state values computed by the script.

In order to minimize confusion, outside of the descriptions above, the term *state variable* will only be used in this manual to describe user-defined or scripted State Variables. For the most part, the modifiers: *user-defined* and *scripted* will be omitted unless they are needed for clarity.

Since many reservoirs are operated based on the *state* of the watershed (e.g., dry or wet hydrologic conditions) or some abstract state of a reservoir or system of reservoirs (e.g., a system drought level), **State Variables** can be developed to compute these watershed conditions.

State Variables can be used wherever a **Model Variable** or **External Variable** can be used to control or influence the operation of a reservoir. This includes (but is not limited to):

- Function-of Rules
- IF_Block Conditional Expressions
- Zones
- Diversions

A **State Variable** is a special form of a *TimeSeries Object*. It is defined and computed through the execution of a *user-defined Jython script*. Jython is a Java implementation of the Python programming language. The script may perform calculations referencing any *TimeSeries object* in the ResSim network and alternative, including all model variables and other **State Variables**.

Python and Jython tutorials and references can be found at www.python.org and www.jython.org.

12.2.1 The State Variable Editor

State variables are created, edited, and managed through the ResSim **State Variable Editor**. The **State Variable Editor** can be accessed by selecting **State Variables…** from the **Edit** menu of either the **Reservoir Network** module (Figure 12.19) or the **Simulation** module.

ResSim's **State Variable Editor** (Figure 12.20) is designed to help you create and manage your state variables. It includes a built-in scripting interface for editing your state variable scripts and accessing needed objects from your ResSim model.

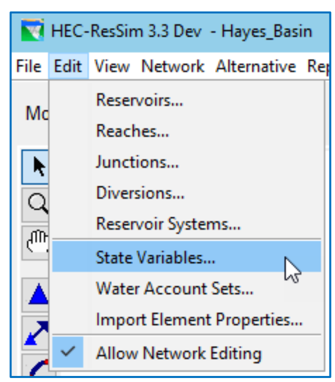


Figure 12.19 Reservoir Network Module—Edit Menu

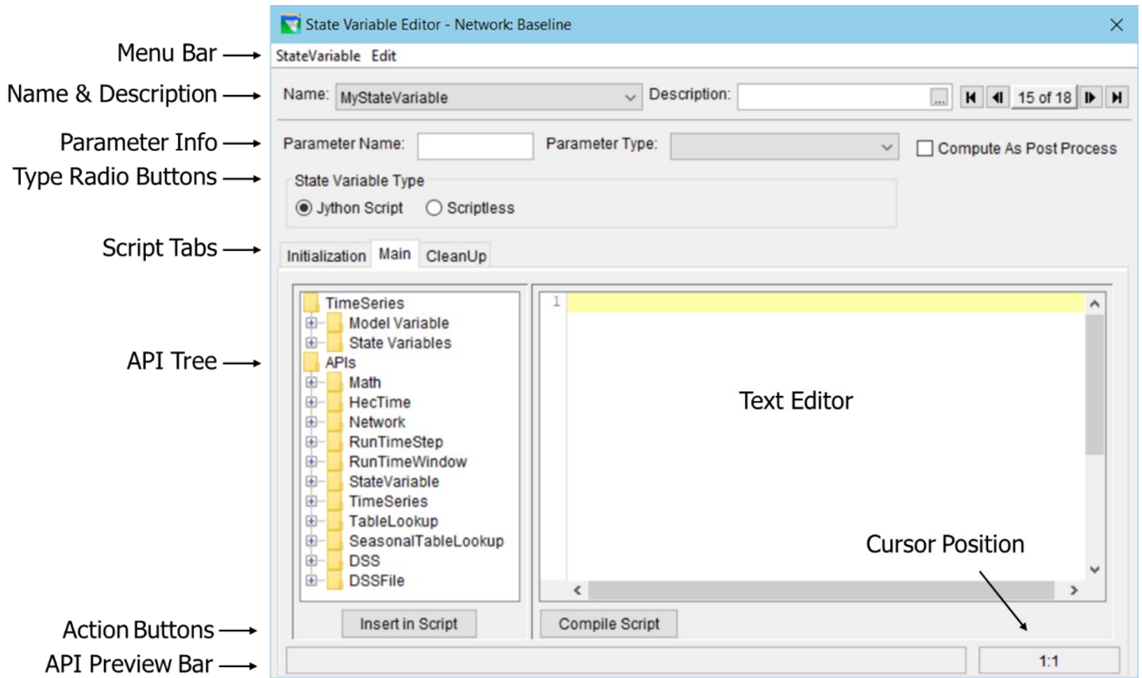


Figure 12.20 The State Variable Editor—Annotated

The **State Variable Editor** consists of several important regions and widgets:

Menu bar—this bar contains the StateVariable and Edit menus.

StateVariable menu—like a File menu, this menu provides access to the functions for creating and managing your state variables.

Edit menu—this menu provides access to the standard text editing functions including **Cut**, **Copy**, **Paste**, and **Undo**. **Redo** will also appear in this menu if there's anything that has been Undone since the last save of the current state variable.

Name & Description—this bar contains the **Name** and **Description** fields as well as a set of navigation buttons. The **Name** field is a selector; use it to select the specific state variable you want to edit. The **Description** field can be used to add a description of the selected state variable. The navigation buttons, like the **Name** selector, give you the ability to cycle through all the network's state variables by moving forward and backward through the list.

Parameter Info—this bar contains the fields for specifying the current state variable's **Parameter Name** and **Type**. For example, if your state variable is going to compute basin inflow, then the parameter type is probably **Flow** and the parameter name you enter is up to you—you might call it flow, or inflow, or flow-in or something descriptive of the data you are computing.

Note: when the compute results are written to the simulation.dss file at the end of the compute, the B and C parts of the DSS pathname for each state variable's output are formed from its Name and its Parameter Name, respectively. The units of the data are based on the parameter type and the current unit system for your watershed.

☐ Compute As Post Process—this checkbox, located to the right of the Parameter Info bar, is provided to force the computation of a State Variable that would otherwise not be computed. State variables that are not referenced in the current operations (e.g., used in a rule) will not be calculated during the simulation unless the **Compute As Post Process** box is checked. When this box is checked, the state variable will be computed after the Regulated Compute has finished running through the time window but has not yet started its CleanUp process. This option might be used when a State Variable has been created for the purpose of post-processing results (e.g., water accounting, system balancing, etc.). *Note: this option to compute the State Variable as a Post Process should only be used when needed since it will increase the compute time.*

State Variable Type Radio Buttons—these radio buttons allow you to choose between two types of state variable—**Jython Script** or **Scriptless**. While all user-defined state variables must be calculated and written by a script, the default **Jython Script** state variable type is calculated by *its own* Jython script, whereas a **Scriptless** state variable has no script of its own and must be calculated by another state variable. You may wish to use a single script to calculate several different but related state variables. In such a case, you would create one Jython Script state variable and one or more Scriptless state variables, all of which would be calculated and populated during the compute of the Jython Script state variable. When a state variable is denoted as Scriptless, the Script tabs and Text Editor will be disabled and ResSim will not attempt to launch a script to compute the variable. A lookback entry will be created for the Scriptless state variable and a DSS record will be defined, however the record will not be populated with data unless a Jython Script state variable explicitly writes values to that Scriptless state variable.

Note: The functionality of the Scriptless state variable will be replaced by Global Variables in future versions of ResSim.

Script Tabs—the **Tabs** in the **State Variable Editor** separate the three scripts that define the three compute functions of each state variable:

Initialization—this tab holds the initStateVariable function for your state variable. Use this tab to initialize the variables and objects that your state variable needs. The initialization function is executed only once, at the start of the Regulated Compute loop.

Main—this tab holds the script that is executed at least once per timestep through the simulation portion of the time window in order to compute the value of the state variable for each timestep.

CleanUp—this tab holds the script that is executed only once, at the end of the Regulated Compute loop. Use this tab to clean up the variables you created in your state variable and to write to disk any local time series or other objects that you need for analysis and/or debugging.

API Tree—a tree widget is used to provide access to the ResSim Application Programming Interface (API) available to the script. The tree is divided into two major branches:

TimeSeries branch—Each node in this branch gives you access to the TimeSeries objects for the various elements in your model (Figure 12.21).

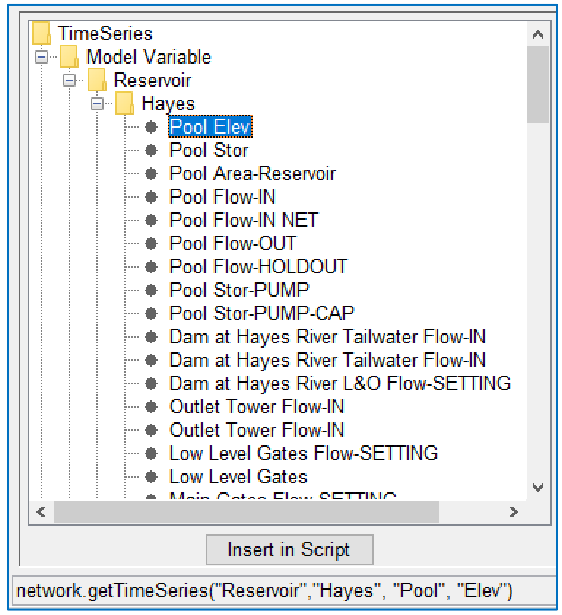


Figure 12.21 Model Variable Node in the Time Series Branch of the API Tree

APIs branch—each node in this branch provides you with easy access to the method calls for accessing and manipulating the objects in your model that you are most likely to need in your scripts. Among other things, it contains API entries for:

- Accessing methods for manipulating the **TimeSeries** objects you have retrieved from the TimeSeries branch.
- Determining the date and time, as well as the index, of the timestep being evaluated.
- Accessing data directly from HEC-DSS file.

By *double-clicking* on a node in the tree, the code needed to retrieve an element's time series or to access a method for manipulating a particular object type can be placed in your script at the current location of your cursor in the editor. This action can also be performed by selecting a node in the API tree and clicking the **Insert in Script** button.

By hovering your cursor over a node in the tree, a tooltip (Figure 12.22) will appear with the Javadoc entry for the method call that is represented by that node. The Javadocs are class-level documentation of the public classes and their methods that you can use in your scripts.

Most method calls (or functions) that you may need require arguments (or parameters) as input. If the method you select requires arguments, the text of that method call that is added to your script will include the arguments between the parentheses ().

Method arguments given as literals (i.e., actual numbers, or strings enclosed by " ") are already complete and require no additional input. The text inserted from items under the **Time Series** branch of the **API Tree** typically fall into this category; for these calls, you do not need to modify the argument list.

Method arguments given as java classes are placeholders for variables that you must supply in the script (Figure 12.22). The method call text inserted from nodes in the **APIs** branch usually requires that you replace the placeholders in the argument list with appropriate variables/objects that you created within your script.

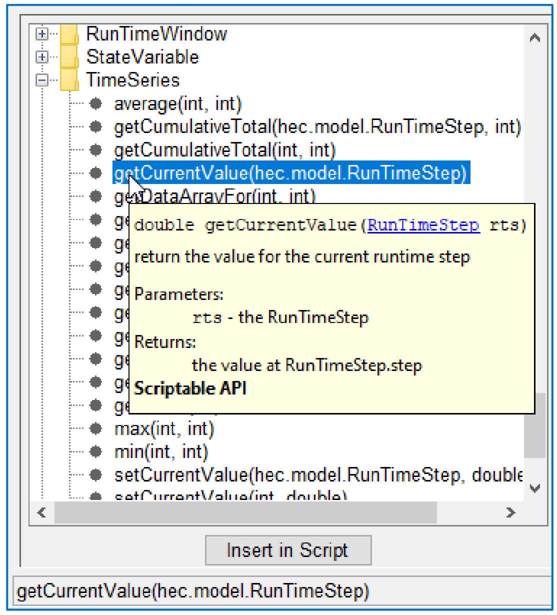


Figure 12.22 Method Node in the APIs Branch of the API Tree—Showing Tooltip

> **Tip** If a method call has: RunTimeStep rts as an argument, you should almost always use the variable: currentRuntimestep for that argument. You will learn more about currentRuntimestep in the next section.

The list of ResSim classes and their methods that are included in the **APIs** branch of the **API Tree** is not complete or all-inclusive. If you are looking for a class or method you think you need from ResSim and it isn't in the list, click on a branch or node in the **APIs** section and press **F1**. The Javadoc Viewer (Figure 12.23) will open showing the Javadoc pertinent to the node or branch you selected. From there, you can search through the Javadocs to find the method you need to access the information you need.

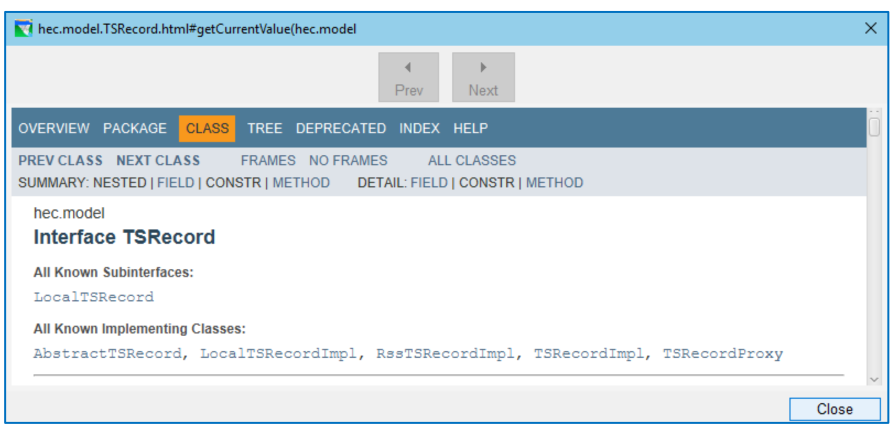


Figure 12.23 The Javadoc Viewer

API Preview Bar—When you select a node in the **API tree**, the text that would be generated by that node will appear in the **Preview Bar**.

Text Editor—This pane of the State variable Editor uses a text editor that interprets the text of your script as Jython source code and colorizes it according to the type of statement or key word it recognizes. This editor supports the standard Windows® shortcut-keys for *select all* (Ctrl+A), *cut* (Ctrl+X), *copy* (Ctrl+C), and *paste* (Ctrl+V), and *undo* (Ctrl+Z) functions. These functions can also be accessed from the **Edit** menu or by *right-clicking* in the **Text Editor** to open a context menu. The *find* (Ctrl+F) function is also supported but can only be access from the shortcut keys.

Cursor Position—the position (row and column number) of the cursor in the Text Editor is shown in the small box to right of the API Preview Bar.

12.2.2 State Variable Scripting Concepts

Here are some basics you should know about scripting state variables in ResSim:

The Scripts—State Variable have three scripts, each of which represents all or part of a compute function of the state variable object in ResSim. These scripts are referred to as:

Init—The **Initialization** script is called by the initialization function of the state variable, which is executed only once per run, early in the regulated compute. This script is where you should create the persistent local variables that you may need to hold data between executions of the Main script.

Main—The **Main** script is executed by the evaluate function of the state variable, which is executed at least once per timestep of the regulated compute. This script is where your code to determine the value of the state variable for the current timestep should be located.

CleanUp—The **CleanUp** script is executed by the function of the state variable that tidies up after the compute. This method is only executed once per compute, at the end of the regulated compute. This script is where you should perform the tasks that should be

Templates—each of the three scripts you can write for your state variables gets started with a template for you to write you script into.

Init—The Initialization script template includes an import statement and the function definition for the initStateVariable method, and some instructive comments describing the arguments that ResSim will send into the function (Figure 12.24).

```
from hec.script import Constants
#
# initialization function. optional.
# set up tables and other things that only need to be performed
# once at the start of the compute.
#
# variables that are passed to this script during the compute initialization:
#   currentVariable - the StateVariable that holds this script
#   network - the ResSim network
#
def initStateVariable(currentVariable, network):
    # return Constants.TRUE if the initialization is successful and
    #        Constants.FALSE if it failed.
    # Returning Constants.FALSE will halt the compute.
    return Constants.TRUE
```

Figure 12.24 The Initialization Script Template

Main—The Main script template is all comments describing the three variables (arguments) that ResSim will provide to the main script and the method call you need to include in your script in order to store the computed value for the state variable for each timestep (Figure 12.25).

```
# no return values are used by the compute from this script.
#
# variables that are available to this script during the compute:
#   currentVariable - the StateVariable that holds this script
#   currentRuntimestep - the current RunTime step
#   network - the ResSim network

# The following represents an undefined value in a time series
#   Constants.UNDEFINED

# add your code here...


# to set the StateVariable's value use:
#   currentVariable.setValue(currentRuntimestep, newValue)
# where newValue is the value you want to set it to.
```

Figure 12.25 The Main Script Template

CleanUp—The CleanUp script template also has an import statement and some comments describing the two variables (arguments) that ResSim will provide to your CleanUp script (Figure 12.26).

```
from hec.script import Constants
#
# script to be run only once, at the end of the compute. optional.

# variables that are available to this script during the compute:
#   currentVariable - the StateVariable that holds this script
#   network - the ResSim network

# The following represents an undefined value in a time series:
#   Constants.UNDEFINED

# add your code here...

```

Figure 12.26 The Cleanup Script Template

Arguments—each script receives some variables that are provided by ResSim so that it can access model elements and their objects for use in computing the value of the state variable. The three variables that may be provided are described below. All three scripts are provided the first two. The third is provided only to the Main script.

currentVariable—is a reference to the current state variable object itself. **currentVariable** is conceptually equivalent to the *this* keyword in Java or the *self* keyword in Python.

network—is a reference to the object that holds the reservoir network. It can be used to access to all the elements of your network and their model variables such as pool elevations, inflow, diversions, etc.… It can also be used to access the current alternative and its properties.

currentRuntimestep—is a reference to the object that holds the timestep being evaluated. The actual date & time of the timestep, as well as the sequence number of the timestep and the run time window are accessed through this object. This object is used, among other things, to determine where (or is that *when*) to store the computed value of the state variable.

The *StateVariable*.setValue method—the Main script template of your state variable instructs you to include the following line in your script to store the compute value of your state variable for the current timestep:

currentVariable.setValue(currentRuntimestep, *newValue*)

Where *newValue* is the value, or local variable that holds the value, to be stored.

12.2.3 Creating a State Variable

To *create* a *new* state variable:

- Open the **State Variable Editor** by selecting **State Variables…** from the **Edit** menu in the **Reservoir Network** or **Simulation** module.
- Select **New** from the **StateVariable** menu (Figure 12.27) of the **State Variable Editor**.
- The **New State Variable** dialog (Figure 12.28) will open. Enter a **Name** (and an optional **Description**) and *click* **OK**.

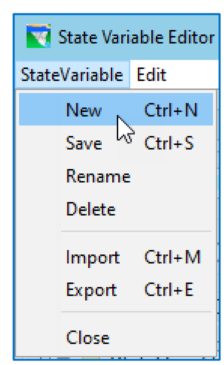


Figure 12.27 StateVariable Menu

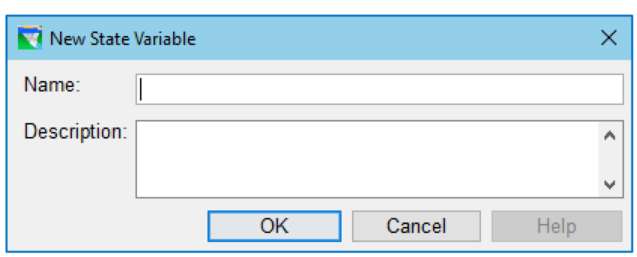


Figure 12.28 New State Variable Dialog

- The **State Variable Editor** will add the new variable to the **Name** selector and make it the selected variable. As such, the name of the new state variable will appear in the **Name** selector, the **Parameter Name** and **Parameter Type** fields will be blank, and the three script tabs will be filled with their relevant templates. Figure 12.29 illustrates the how the **State Variable Editor** will appear immediately after a new state variable is created.
- Enter a **Parameter Name** for the state variable that you will be computing. This name will be used for the **C** part of the HEC-DSS pathname that will be written to the simulation.dss file for this state variable.
- Select a **Parameter Type** from the selection list. The parameter type is used to determine the units of the state variable. If your state variable should be unitless, then select **Code** or **Count** for your variable's parameter type.

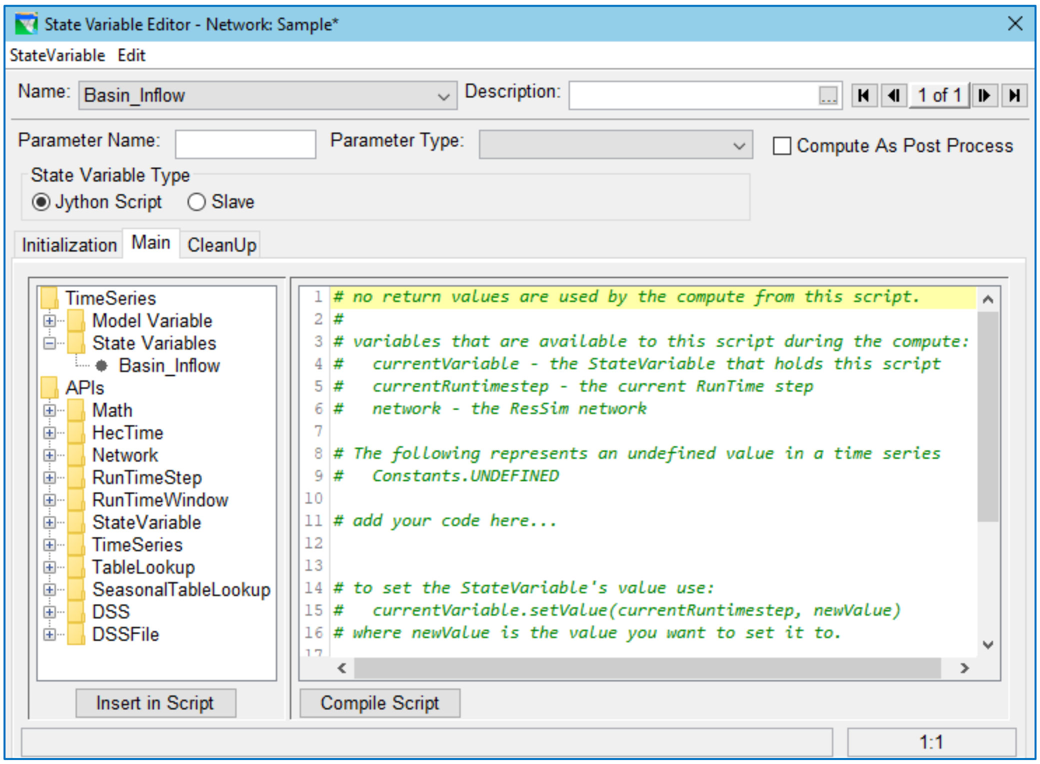


Figure 12.29 New State Variable in the State Variable Editor

12.2.4 Compiling your State Variable

The State Variable Editor provides a syntax checker that you run your script through to check for errors. To do so:

- Click the **Compile Script** button below the Text Editor
- If you get a message box that says "Compile failed":
  - Select **OK** to dismiss the message box.
  - Correct the mentioned syntax error.

- Repeat the above steps until the process results in a message box that says "Script Compiled Successfully". Select **OK** to close the message box; then save the state variable.

12.2.5 Saving a State Variable

Although the **State Variable Editor** will prompt you to save if you try to close the editor after having made changes to the current state variable, it is good practice to save your work regularly. And, you should always save changes before switching to another state variable within the editor.

To save your current state variable:

- Select **Save** from the **StateVariable** menu, or type Ctrl+S.

12.2.6 Renaming a State Variable

To *rename* the current state variable:

- Select the state variable you want to rename from the **Name** selector in the **State Variable Editor**.
- Select **Rename** from the **StateVariable** menu.
- A **Rename State Variable** dialog will appear (Figure 12.30). Enter a new name for the state variable and click **OK**.
- The name shown in the **Name** selector for the current state variable will change to the new name.

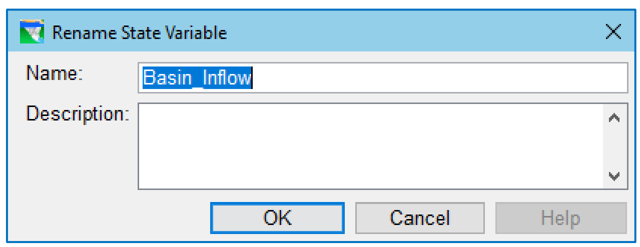


Figure 12.30 Rename State Variable

12.2.7 Deleting a State Variable

Deleting a state variable merely deletes the state variable object and its scripts. It does NOT delete any existing references to the state variable in the rules, IF_Blocks, zones, or diversion; these references will be broken until you go find them and fix or removed them.

To *delete* the current state variable:

- Select the state variable you want to delete from the **Name** selector in the **State Variable Editor**.
- Select **Delete** from the **StateVariable** menu.
- A **Confirm Delete** dialog will appear (Figure 12.31). Click **Yes** to proceed with the delete process or **No** to abort the delete.

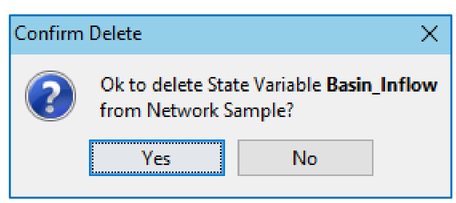


Figure 12.31 State Variable—Confirm Delete Dialog

- The state variable will be removed from the **Name** selector and the first variable in the selection list will become the current variable in the **State Variable Editor**.

12.2.8 Importing and Exporting a State Variable

You may find it useful to save the three parts of your script to a single file so that you can use a more sophisticated editor for building and editing your scripts. The **Export** function is very useful for this purpose. It will add comment lines to the file to label each section script code for the tab it belongs to. If you leave those comment lines intact during your external editing, you can use the **Import** function to bring your edited scripts back into the state variable's tabs in ResSim. This process is much easier and less error prone than using Ctrl+A to select all, then Ctrl+C to copy the contents of a tab to the Windows clipboard, then Ctrl+V to paste the clipboard contents into the external editor. Then you would have to reverse those steps to return the edited code back into the state variable ResSim.

To write the three scripts of your state variable to a file:

- Select the state variable you want to export from the **Name** selector in the **State Variable Editor**.
- Select **Export** from the State Variable menu.
- An **Export State Variable Script** file browser will open. Use the browser to select the folder where you want to save the file and enter a filename in the **File name** field at the bottom. Note—although the browser has a field for **Files of type:** which defaults to **Python Scripts**, it does not use this setting to append an extension to the filename you entered; instead, it uses the file type to filter the list of files it shows. For convenience when you later want to **Import** your script file, we recommend that you add the extension *.py* to your exported script filenames.
- Click the **Export** button to complete the export process and close the browser.

To read the three scripts of your state variable from a file that you had previously exported and place them into their respective tabs in the **State Variable Editor**:

- Select the state variable you want to import from the **Name** selector in the **State Variable Editor**.
- Select **Import** from the State Variable menu.
- An **Import State Variable Script** file browser will open. Use the browser to locate and select the file you want to import. Note—the browser has a field for **Files of type:** which defaults to **Python Scripts**. It uses the file type to filter the list of files it shows; Python scripts use *.py* as the file extension.
- Click the **Import** button to complete the import process and close the browser.
- The contents of the three tabs of your current state variable will be replaced with the contents of the associated section from selected file.

12.3 Scripted Rules

A scripted rule is an advanced operation rule that provides you with the ability to write your own **Release Function** rule so that you can perform complex calculations or address a complex set of constraints to arrive at a desired release. The scripted rule must be written in Jython, a Java implementation of the Python programming language. Python and Jython tutorials and book references can be found at www.python.org and www.jython.org.

To create a scripted rule:

- Open the **Reservoir Editor** and select the **Operations** tab.
- Select the operation set in which you want to use the scripted rule.
- *Right-click* on a zone in the **Zone-Rules** tree and select **Add New Rule…** from the context menu.
- The **New Operating Rule** dialog will open (Figure 12.32).
- Select the release element for which you want the scripted rule to influence.
- Set the **Rule Type** to **Script**.
- Enter a name for your new rule and click **OK**.

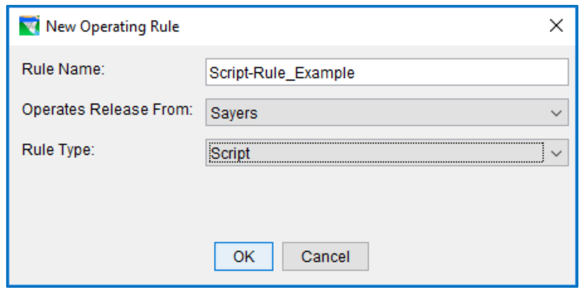


Figure 12.32 New Operating Rule Dialog—Script

- The new **Scripted Rule** will appear at the bottom of the selected zone. The rule will be the selected node in the **Zone-Rules** tree and the edit panel will display the **Scripted Rule Editor** (Figure 12.33). And, as with state variables, the Text Editor pane of the Scripted Rule Editor will include the text of a template that provides the framework for your new rule script.

Since a scripted rule is just another rule type to ResSim, all the rule management functions described in Chapter 11, Section 0, like renaming, deleting, and prioritizing, apply to scripted rules.

12.3.1 The Scripted Rule Editor

The **Scripted Rule** editor (Figure 12.33) has the same scripting interface as the **State Variable Editor** including the API tree and the Text Editor:

Name & Description—Like any other rule, you can enter a **Description** for the rule in the Name & Description section.

API Tree—A tree widget is used to provide access to the ResSim Application Programming Interface (API) available to the script. The tree is divided into two major branches:

The **TimeSeries** Branch—Each node in this branch gives you access to the TimeSeries objects for the various elements in your model.

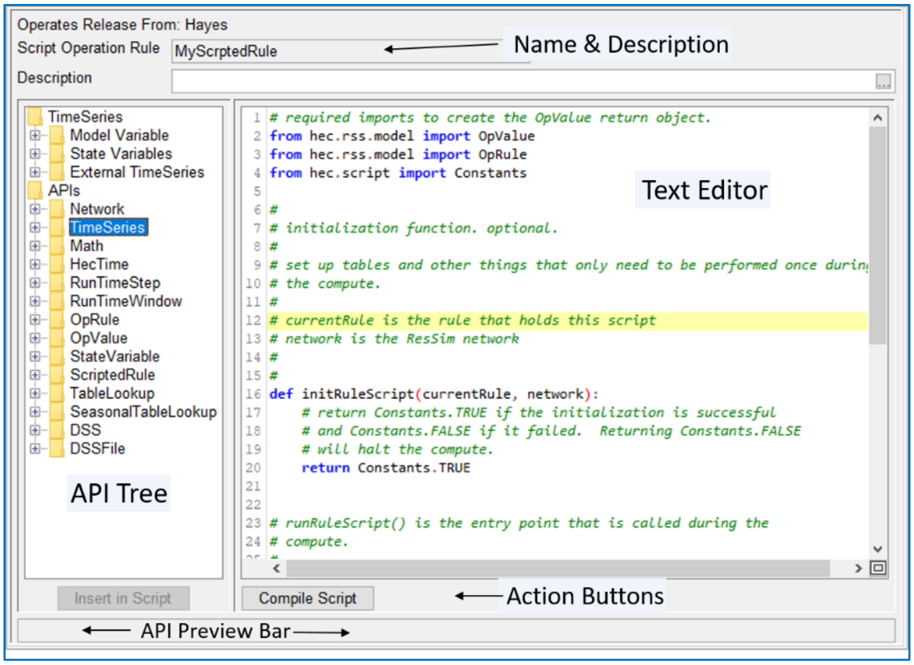


Figure 12.33 Scripted Rule Editor—"Default" Template

The **APIs** Branch—Each node in this branch provides you with easy access to the method calls for accessing and manipulating the java objects in your model that you are most likely to need in your scripts.

By *double-clicking* on a node in the tree, the code needed to retrieve an element's time series or to access a method for manipulating a particular object type can be placed in your script at the current location of your cursor in the editor. This action can also be performed by selecting a node in the API tree and clicking the **Insert in Script** button.

By hovering your cursor over a node in the tree, a tooltip will appear with the Javadoc entry for the method call that is represented by that node. The Javadocs are class-level documentation of the public classes and their methods that you can use in your scripts.

Most method calls (or functions) that you may need require arguments (or parameters) as input. If the method you select requires arguments, the text of that method call that is added to your script will include the arguments between the parentheses ().

Method arguments given as literals (i.e., actual numbers, or strings enclosed by " ") are already complete and require no additional input. The text inserted from items under the Time Series branch of the API tree typically fall into this category; for these calls, you do not need to modify the argument list.

Method arguments given as java classes are placeholders for variables that you must supply in the script (Figure 12.34). The method call text inserted from nodes in the **APIs** branch usually requires that you replace the placeholders in the argument list with appropriate variables/objects that you created within your script.

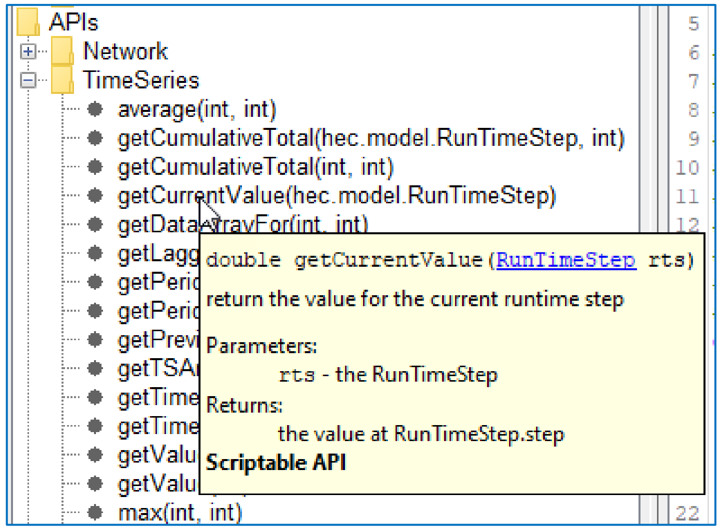


Figure 12.34 Scripted Rule Editor—API Tree—Java Object Classes and Methods

Tip If a method call has: RunTimeStep rts as an argument, you should almost always use the variable: currentRuntimestep for that argument. You will learn more about currentRuntimestep in the next section.

The list of The ResSim classes and their methods that are included in the **APIs** branch of the **API Tree** is not complete nor all-inclusive. If you are looking for a class or method you think you need from ResSim and it isn't in the list, click on a branch or node in the **APIs** section and press **F1**. The Javadoc Viewer (Figure 12.35) will open showing the Javadoc pertinent to the node or branch you selected. From there, you can search through the Javadocs to find the method you need.

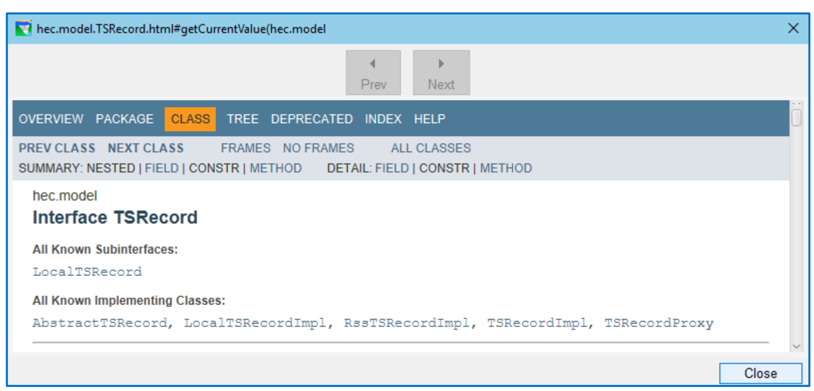


Figure 12.35 The Javadoc Viewer

API Preview Bar—When you select a node in the **API tree**, the text that would be generated by that node will appear in the **Preview Bar**.

Text Editor—This pane of the State variable Editor uses a text editor that interprets the text of your script as Jython source code and colorizes it according to the type of statement or key word it recognizes. This editor supports the standard Windows® shortcut-keys for *select all* (Ctrl+A), *cut* (Ctrl+X), *copy* (Ctrl+C), and *paste* (Ctrl+V), and *undo* (Ctrl+Z) functions. These functions can also be accessed from the **Edit** menu or by *right-clicking* in the **Text Editor** to open a context menu. The *find* (Ctrl+F) function is also supported but can only be access from the shortcut keys.

12.3.2 Development Concepts for Scripted Rules

Developing a scripted rule involves writing Jython code to create the data objects and execute the methods and other calculations that will produce a desired release limit to be returned by your rule. It is important to note that a scripted rule is essentially a Release Function rule; thus, the rule must return a desired "release" value and an associated "Limit Type" for that release value. The limit types are Maximum, Minimum, and Specified.

The code development process has been simplified somewhat by two things: the scripted rule template and the API tree. The API tree is described in the previous section and the template is described below:

12.3.2.1 The Scripted Rule Template

Like State Variables, each new Scripted Rule will be created with a template. Figure 12.36 displays the Scripted Rule template. This template has some very important lines of code as well as several comment statements which are marked with the number sign (#).

The template basically gives you instructions on three key elements that you must provide in your script:

The *two function statements for the* initRuleScript and runRuleScript functions. You should use these statements as-is, do not edit them.

The *opValue* object, its *init* (or set) method, and the constants that represent the available limit types.

The *return statements* that will cause ResSim to correctly receive the result of your script's functions and utilize that information within the release decision logic.

Looking at the template in more detail, you will notice that it starts with some import statements. Import statements are needed to "teach" your script what it needs to know about the java objects your script will be using. The three import statements in the template are required for your scripted rule to function properly. It is recommended that you add any additional import statements that your script may need below these lines.

Below the imports are the two function definitions mentioned above. Each function's def statement is preceded by several comment lines that describe the function and its arguments. Since the two functions are called by ResSim during the Regulated compute, the arguments are objects that are passed into your script from your model. A more comprehensive description follows:

```
# required imports to create the OpValue return object.
from hec.rss.model import OpValue
from hec.rss.model import OpRule
from hec.script import Constants

#
# initialization function. optional.
#
# set up tables and other things that only need to be performed once during
# the compute.
#
# currentRule is the rule that holds this script
# network is the ResSim network
#
#
def initRuleScript(currentRule, network):
    # return Constants.TRUE if the initialization is successful
    # and Constants.FALSE if it failed.  Returning Constants.FALSE
    # will halt the compute.
    return Constants.TRUE


# runRuleScript() is the entry point that is called during the
# compute.
#
# currentRule is the rule that holds this script
# network is the ResSim network
# currentRuntimestep is the current Run Time Step
def runRuleScript(currentRule, network, currentRuntimestep):

    # create new Operation Value (OpValue) to return
    opValue = OpValue()

    # add your code here

    # set type and value for OpValue
    #  type is one of:
    #  OpRule.RULETYPE_MAX  - maximum flow
    #  OpRule.RULETYPE_MIN  - minimum flow
    #  OpRule.RULETYPE_SPEC - specified flow
    opValue.init(OpRule.RULETYPE_MAX, 1000)

    # return the Operation Value.
    # return "None" to have no effect on the compute
    return opValue
```

Figure 12.36 The Scripted Rule Template

initRuleScript—this function is where you should put the setup code for your rule. This function is called by the rule object only once, at the start of the regulated compute loop. Use the function definition statement as-is, do not edit it. It is the content of the function that you are expected to edit. The function must return a value of Constants.TRUE (to indicate success) or Constants.FALSE (to indicate failure). It is up to you to determine if your script requires special logic that might cause it to return FALSE. The return value of the initRuleScript function should be True for successful and False for unsuccessful. If this function returns false, the ResSim compute will be aborted.

runRuleScript—this is the "main" function of the script. This function is called by the rule's *evaluate* method and is called at least once per timestep. Use the function definition statement as-is, do not edit it. It is the content of the function that you are expected to edit. This is where you should put the code that calculates the desired release limit of your rule. The return value of the **runRuleScript** function is an **OpValue** object. In this object, you are expected to store the value of the desired release limit that you calculated and the limit type.

The function arguments—the two default functions of your scripted rule receive some variables (arguments) that are provided by ResSim so that your function can access model data needed to compute the value of the rule's desired release limit. The three variables that may be provided are described below. Both functions receive the first two variables. Only the **runRuleScript** function receives the third variable.

currentVariable—is a reference to the current state variable object itself. **currentVariable** is conceptually equivalent to the *this* keyword in Java or the *self* keyword in Python.

network—is a reference to the object that holds the reservoir network. It can be used to access to all the elements of your network and their model variables such as pool elevations, inflow, diversions, etc. It can also be used to access the current alternative and its properties.

currentRuntimestep—is a reference to the object that holds the timestep being evaluated. The actual date & time of the timestep, as well as the sequence number of the timestep and the run time window are accessed through this object. This object is used, among other things, to determine where (or is that *when*) to store the computed value of the state variable.

The **opValue** object—this object is used to carry each rule's desired release limit value and the limit type back to ResSim (through the return statement) for use in determining the allowable release range (see the introduction to Chapter 11 for details on the release range). As described in the template, the limit types are:

- OpRule.RULETYPE_MAX—maximum flow
- OpRule.RULETYPE_MIN—minimum flow
- OpRule.RULETYPE_SPEC—specified flow

The template also shows how to create an OpValue object and how to initialize it:

```
# create new Operation Value (OpValue) to return
opValue = OpValue()
# set type and value for OpValue
opValue.init(OpRule.RULETYPE_MAX, 1000)
```

Remember, for these lines to work, the imports at the top of the template must be included in your script.

12.3.2.2 Compiling your Scripted Rule

The **Scripted Rule** editor provides a syntax checker that you can run your script through to check for errors. To do so:

- Click the **Compile Script** button below the Text Editor
- If you get a message box that says "Compile failed":
  - Select **OK** to dismiss the message box.
  - Correct the mentioned syntax error.
- Repeat the above steps until the process results in a message box that says "Script Compiled Successfully". Select **OK** to close the message box then save the state variable.

Compiling the Script simply checks for basic syntax errors. It does not guarantee that the programming logic of the script is error free.

12.4 Water Account Sets

Reservoirs used for water supply may reserve a particular volume of their conservation pools for the use of a water storage account holder. The water storage account holders may draw water from their accounts to use for water supply, and their currently available volume of water is determined by tracking credits and debits from their storage account.

ResSim has a water accounting feature that tracks reservoir storage accounts. Water Account Sets are used to define storage accounts within your reservoirs. In order to use the ResSim water accounting feature, you will need to set up a water account set with water account(s) and create reservoir release rules for the water account holder(s). The lower limit of the conservation pool must also be defined. This setup allows ResSim to track the volume in the water account(s). The ResSim Yield alternative type can also use the water accounting feature to calculate water account yield or to size a water account to meet a particular yield. See Appendix F for details on the ResSim Yield feature.

To create a Water Account Set for your network:

- Select **Water Account Sets…** from the Network Module's **Edit** menu. The **Water Account Set Editor** will open (Figure 12.37).

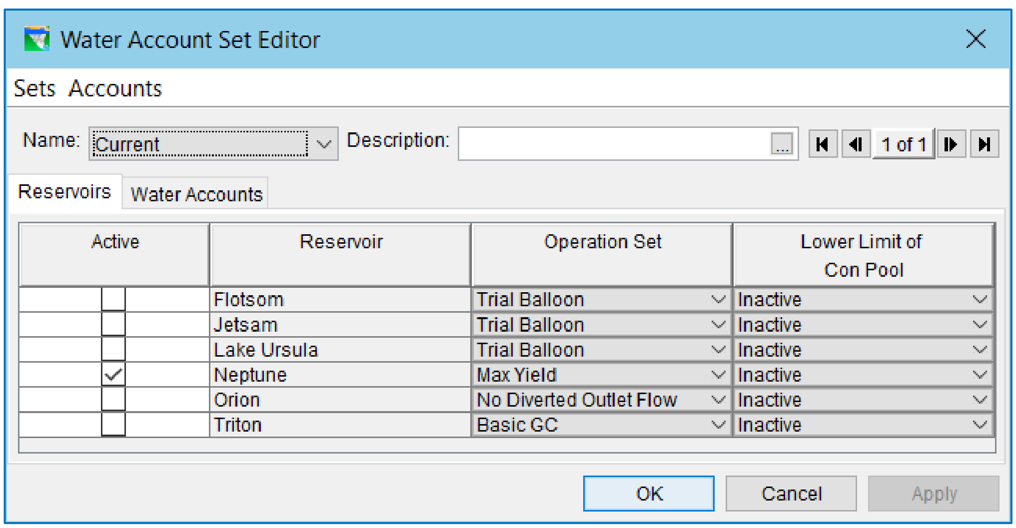


Figure 12.37 Water Account Set Editor—Reservoirs tab

- Create a new Water Account Set by selecting **New** from the **Set** menu. You will be prompted to give your new water account set a name and description.
- Next, from the **Reservoir** tab, select the reservoir(s) that have water accounts by checking their box(es) in the *Active* column of the table. Then select the appropriate Operation Set and the lower limit of the Conservation Storage pool. The designated Guide Curve for the operation set is considered the upper limit of the Conservation Storage pool. Click **Apply**.
- Next create your Water Accounts. Select the **Water Accounts** tab (Figure 12.38).
- From the **Accounts** menu, select **New**. You will be prompted to give your water account a name and description. You may create multiple water accounts. For each account, do the following:

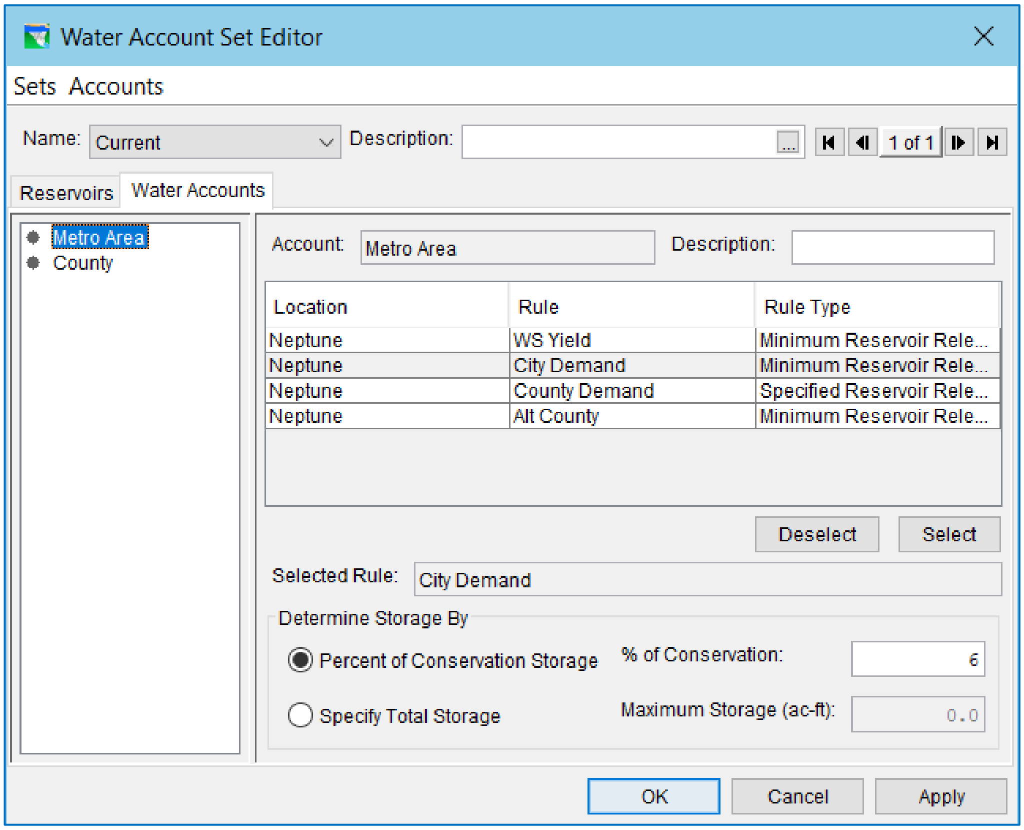


Figure 12.38 Water Account Set Editor—Water Accounts tab

- Select an account in the lefthand pane of the editor to activate its information. ResSim will display a list of all minimum or specified flow and hydropower rules used in the Water Account Set's active reservoir(s). Choose the rule that represents the release allocated to the water account holder and click **Select**.
- The conservation pool storage allocated to that water account holder is either specified as a *percent of the conservation pool* volume, or as a total volume. Select the appropriate radio button and enter either the percent or the maximum volume. Click **OK**.

Note: Water accounting output available in the simulation.dss file includes:

- Max storage of water account
- Computed storage of water account
- Water account inflow
- Water account demand withdrawal

12.5 Capacity Outage Schedules

The Capacity **Outage Schedule** is the first of four features that can be added to an operation set by activating it from the **Operations** menu of the **Reservoir Editor**. By default, this feature is OFF because it is assumed that all outlets are available for use all the time; but for many reservoirs, especially hydropower reservoirs, that assumption is false. Periodic gate and turbine maintenance occurs at most projects and the Outage Schedule feature allows you to specify when an outlet is expected to be unavailable and assess the potential impacts of that downtime on the overall operation of the reservoir.

A scheduled outage is typically used to represent scheduled maintenance of the reservoir's outlets. If an outlet is not available due to an unforeseen circumstance, then you might consider using the **Outlet Capacity Overrides** in the **Simulation** module (Section 15.7).

To *activate* the **Outage Schedule** feature:

- Open the **Reservoir Editor** and select the reservoir that needs an outage schedule.
- Select the **Operations** tab.
- From the **Operations** menu, select **Use Outage Schedule**.

  The **Outages** sub-tab will become active (Figure 12.40) and a check mark will appear in front of the **Use Outage Schedule** option in the **Operations** menu to indicate that the **Outage Schedule** is ON (Figure 12.39).

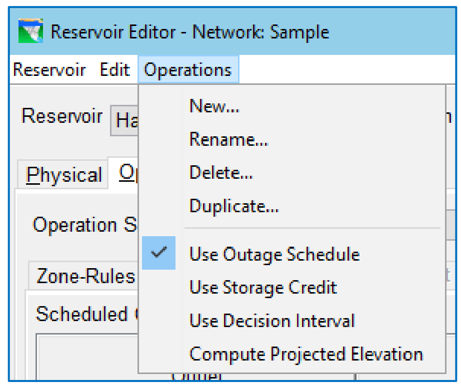


Figure 12.39 Reservoir Editor—Operations Menu—Use Outage Schedule

The **Outages** sub-tab contains the **Outage Schedule** edit panel, which allows you to specify which outlets are scheduled to be out, when each outage will occur, and what fraction of the outlet's capacity will be ***available*** during the outage and whether or not this is a regularly **Repeating** outage (Figure 12.40).

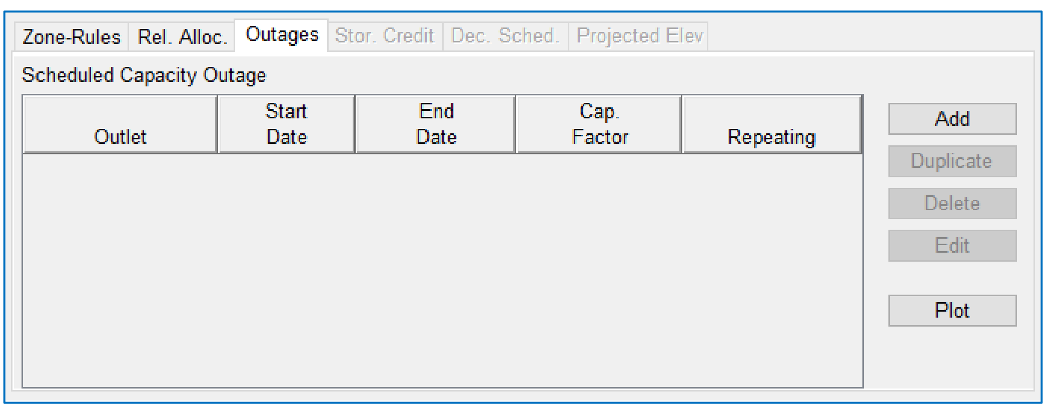


Figure 12.40 Capacity Outage Schedule—Edit Panel

To add an outage to the list of **Scheduled Capacity Outages**:

- *Click* the **Add** button located to the right of the **Scheduled Capacity Outages** table. The **Outlet Capacity Schedule Entry** dialog (a.k.a, **Outage Editor**, Figure 12.41) will open. To define the outage, make a selection for each of the following outage attributes:

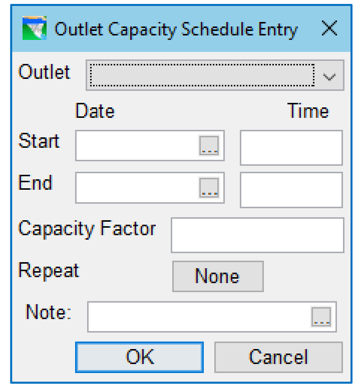


Figure 12.41 Outlet Capacity Schedule Entry Dialog

Outlet—select the outlet that is scheduled for maintenance.

Starting Date & **Time**—enter the date and time when the outage will begin. The ... button in the date field can be used to open a calendar widget to help you specify the date.

Ending Date & **Time—**enter the date and time when the outage will end.

Capacity Factor—enter a value that represents the fractional portion of the outlet's capacity that will ***available*** for release during the outage. For example, if a spillway outlet has 4 equal gates and only one of them will be unavailable, then ¾ of the spillway capacity, or 75%, will be *available* and you would enter 0.75 as the capacity factor.

Repeat—The label on the **Repeat** button shows the basic recurrence interval for the outage. *Click* the button to open the **Repeat** dialog (Figure 12.42) where you will specify the *recurrence interval* for the repeating outage.

A repeating outage uses the **Start** and **End** times to determine the duration of the outage and the first occurrence of the outage will begin on the **Start Date** and **Time**. Each subsequent occurrence will begin one *recurrence interval* after the *start* of the previous occurrence.

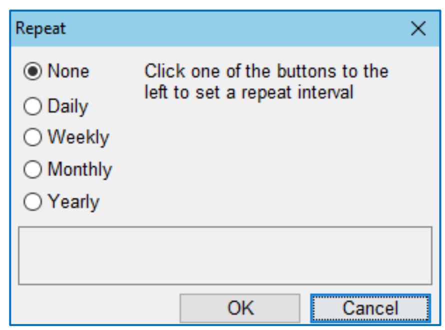


Figure 12.42 Outage Repeat Dialog

Select one of the radio buttons to indicate how often the outage should occur. Options include:

None—the outage will occur only once, beginning on the date and time specified.

Daily—specify the recurrence interval, in number of days (Figure 12.43).

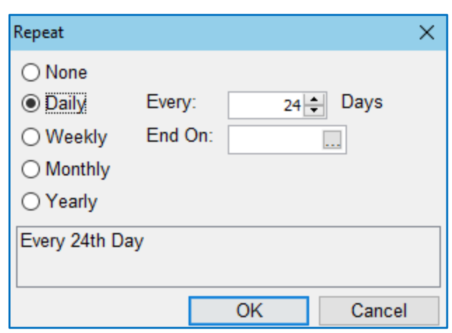


Figure 12.43 Outage Repeat Dialog

Every:—each of the repeat options (except None) allow you to specify how many of the option's time units actually represent the recurrence interval. The default is 1.

End On:—this is a specific date when the repeating outage stops completely. If blank, the outage repeats throughout the whole simulation.

Weekly—specify the recurrence interval, in number of weeks.

Monthly—specify the recurrence interval, in number of months.

Yearly—specify the recurrence interval, in number of years.

Note:—enter a description of the outage. This note can only be seen in the Outage Editor. It will not appear in the **Scheduled Capacity Outage** table.

Once an outage has been specified, it will become an entry in the **Scheduled Capacity Outage** table (Figure 12.44). If you select a row in this table, the **Duplicate**, **Delete**, and **Edit** buttons become available. Use these buttons to manage your scheduled outages.

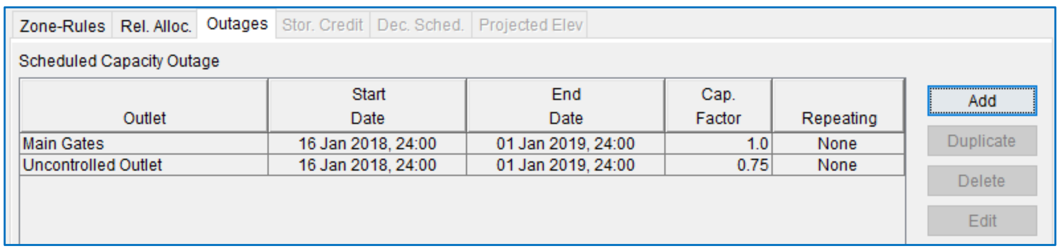


Figure 12.44 Scheduled Capacity Outage Example

12.6 Storage Credit

As discussed in Section 10.4, the **Reservoir Editor** allows you to pick the top of zone curve for any zone in your operation set to act as the *guide curve*. The storage above the guide curve is considered *flood* storage and the storage below is considered *conservation* storage.

The **Storage Credit** feature allows you to vary the *allocated* flood storage (and thus the guide curve) of the current reservoir based on the *available* storage space in one or more other reservoirs in the watershed. The idea is that if *incidental* storage is available in the crediting reservoirs, then the required flood control space in the current reservoir can be reduced by some fraction of the available credit space. Reducing the required flood control space has the net effect of raising the guide curve and thus increasing the reservoir's potential conservation storage. Note—the crediting reservoirs need not have dedicated flood storage space; their credit storage could be a result of conservation releases.

This feature requires that you identify:

- the **reservoirs** that could provide flood storage ***credit*** to the current reservoir,
- the total reservoir storage at "full pool" for each reservoir,
- the **maximum** amount of **credit** each reservoir could provide, and
- whether **negative credit** can be computed for that reservoir. A net negative credit could *increase* the required flood storage at the current reservoir and thus *lower* the guide curve.

To activate the **Storage Credit** option for a reservoir:

- Open the **Reservoir Editor**.
- Select the reservoir to receive the credit.
- Select the **Operations** tab.
- From the **Operations** menu, select **Use Storage Credit**. The **Stor. Credit** sub-tab will become *active* and a check mark will appear in the **Operations** menu in front of the **Use Storage Credit** option to indicate that storage credit will be computed and used (Figure 12.45).

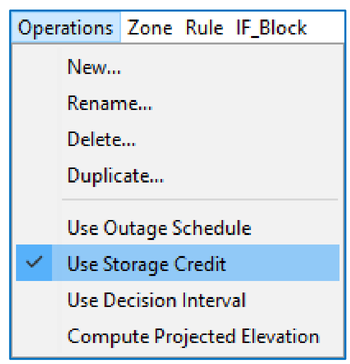


Figure 12.45 Reservoir Editor—Operations Menu

The **Stor. Credit** sub-tab contains the **Storage Credit** edit panel (Figure 12.46). Use this tab/editor to identify the crediting reservoirs and specify their storage credit parameters. The components of the **Storage Credit** edit panel are:

Crediting Reservoir List—The table in the upper left of the editor is the list of reservoirs that can provide storage credit to the current reservoir.

Edit Reservoir Set button—This button is located in the upper right corner of the editor. This button will open a **Reservoirs** Selector dialog; use it to select the crediting reservoirs.

Storage Credit Relationship—The table in the lower left of the editor is the relationship between the available storage credit and the credit that can actually be applied to the required flood storage at the current reservoir.

Seasonal Variation Edit button—This button is located in the bottom right corner of the editor. It will open the **Seasonal Variation** editor which you can use to define a set of seasons across which the Storage Credit relationship varies.

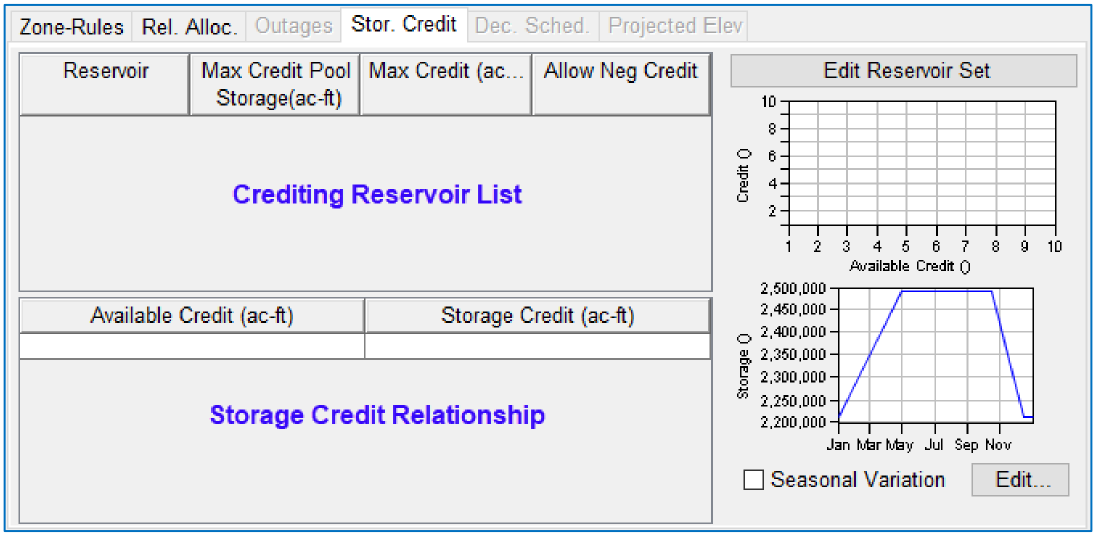


Figure 12.46 Storage Credit Edit Panel

Thumbnail Plots—Both of the two thumbnail plots located on the right side of the edit panel are associated with the Storage Credit Relationship.

The upper plot displays the relationship(s) as defined in the table. If the Storage Credit Relationship varies seasonally, this plot will show a curve for each season's (*column's*) relationship.

The lower plot displays the guide curve of the current reservoir in terms of storage. For each *row* in the lower table, a curve is added to the plot to show how the potential flood storage credit would impact the required flood pool storage and, thus, the guide curve.

As with all thumbnail plots, if you *double-click* on one, a full-size plot will open.

To identify the reservoirs that can provide storage credit to the current reservoir:

- Select the **Edit Reservoir Set** button. The **Reservoirs** Selector dialog will open (Figure 12.47).
- In the **Available** list, *double-click* on each reservoir that can provide storage credit to the current reservoir; this will move the reservoirs from the **Available** list to the **Selected** list. For a detailed description of the **Selector** dialog, refer to Appendix C.

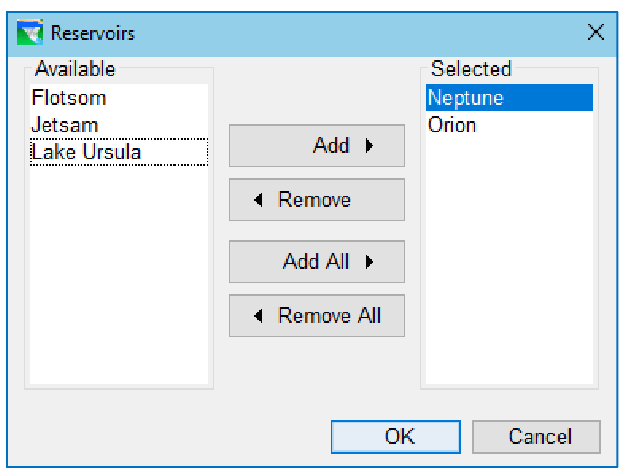


Figure 12.47 Storage Credit—Reservoirs Selector Dialog

- Click **OK** to accept your selections and close the **Reservoir** selector dialog.
- The reservoir(s) you selected will appear in the Crediting Reservoirs List of the **Storage Credit** edit panel. For each reservoir in the list, specify:

  Maximum Credit Pool Storage—<u>this is the total storage of the reservoir below which available storage can be counted as storage credit</u>. If the crediting reservoir has an allocated flood storage pool *and* this space can be used as credit storage, then the **Maximum Credit Pool Storage** would be at the top of the flood pool; if credit storage can only come from available space in the conservation pool, then the **Maximum Credit Pool Storage** would be at the top of the conservation pool. If the crediting reservoir does not have a flood pool, then the **Maximum Credit Pool Storage** would probably be at "full" or "maximum" pool.

  Maximum Credit—this is the amount of available storage at the crediting reservoir that can be used as credit to the current reservoir.

  Allow Neg Credit—specify whether the reservoir can provide "negative" credit when the current pool is above the Maximum Credit Pool storage. Negative credit from a credit reservoir would have the effect of lowering the guide curve at the reservoir where you are adjusting the guide curve. This could occur if the current storage in the crediting reservoir's pool is greater than its maximum credit pool storage.

- Next, in the Storage Credit Relationship table, specify how much flood **Storage Credit** can be applied to the required flood storage at the current reservoir as a function of the total available credit storage in the crediting reservoirs. This can be defined by a single curve or by a seasonally-varying family of curves. See Appendix C for a detailed description of the **Seasonal Variation** Editor that you can use to define the seasons.

  Figure 12.48 shows an example of a completed Storage Credit specification. In this example, two crediting reservoirs have been identified and the Storage Credit relationship has been specified as a seasonally varying relationship where at its maximum, the relationship is not quite 1:1 and at its lowest (in the summer months), the relationship indicates that no credit is allowed.

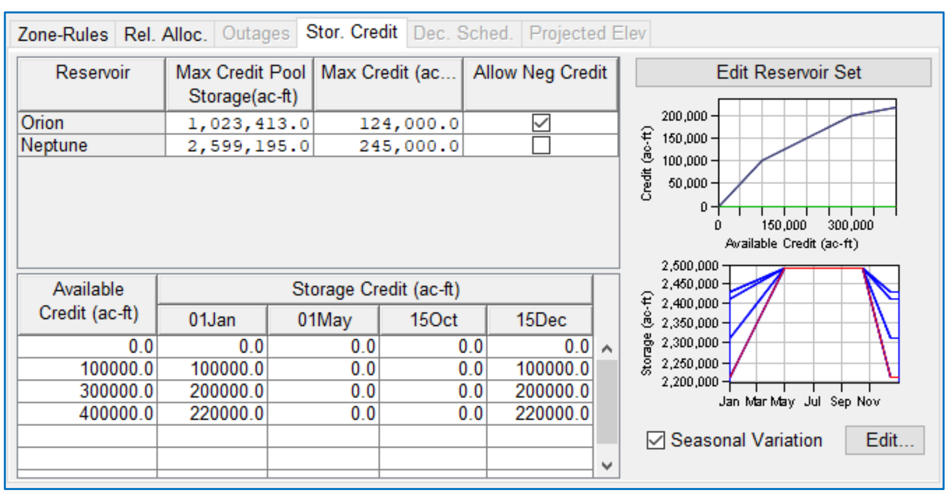


Figure 12.48 Storage Credit Edit Panel—Credit Definition for Selected Reservoir(s)

12.7 Reservoir Decision Schedule

By default, each reservoir in your network makes a release decision for every computation interval (timestep). By activating the **Decision Schedule** feature, you can specify a different decision interval or a decision schedule for each reservoir. By changing a reservoir's decision interval, you are telling the reservoir to evaluate conditions over a different or varying time horizon to determine its releases; within the release capacity of the reservoir, each release decision will be held constant until the next decision. Your selection of the decision interval should be appropriate for the compute interval of the simulation.

To activate the **Decision Interval** feature for a reservoir:

- Open the **Reservoir Editor**.
- Select the reservoir to receive the credit.
- Select the **Operations** tab.
- From the **Operations** menu, select **Use Decision Interval**. The **Dec. Sched** sub-tab will become *active* and a check mark will appear in the **Operations** menu in front of the **Use Decision Interval** option to indicate that the **Decision Schedule** feature is ON (Figure 12.49).

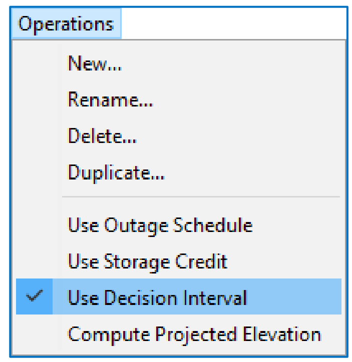


Figure 12.49 Reservoir Editor—Operations Menu—Use Decision Interval

The **Dec. Sched.** sub-tab contains the **Decision Schedule** edit panel. Use this editor to specify the decision interval or schedule.

To specify the reservoir decision interval or schedule:

- Select an **Interval Option** from the list. Your three options are:

  Every Time Step—this is the default and what ResSim does when the **Decision Schedule** feature is OFF.

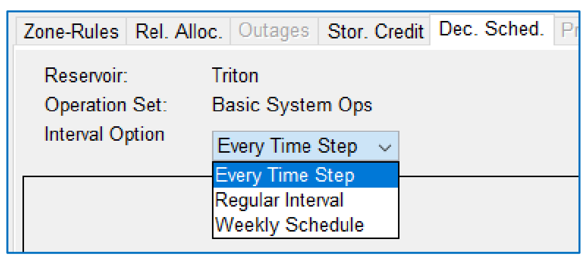


Figure 12.50 Decision Schedule Editor—Decision Interval Options

  Regular Interval—use this option if you want the reservoir to make its release decisions on a larger interval than the current timestep. The lower panel of the editor will fill with four options for defining the new decision interval (Figure 12.51).

  Interval—this is the amount of time between release decisions. You can choose from the following intervals: 1 hr, 2 hr, 3 hr, 4 hr, 6 hr, 8 hr, 12 hr, Daily, Weekly, and Monthly. But, be careful—you should only select a decision interval that is a multiple of the current timestep. For example, if the timestep is 6 hours, then valid decision intervals include 12 hr, Daily, Weekly, and Monthly.

Start Hour—this is the time of the first decision. You can select from 0000 to 2300.

Day of Week—if you chose Weekly Interval, this is the day of the week when each decision will be made.

Day of Month—if you chose Monthly Interval, this is the day of the month when each decision will be made.

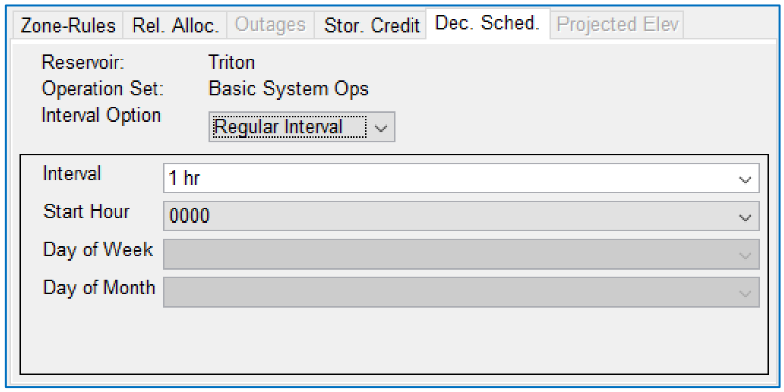


Figure 12.51 Decision Schedule Edit Panel—Regular Interval Option

Weekly Schedule—use this option if you want to specify an irregular schedule for making release decision. The lower panel of the editor will fill with a 7-column table, one column for each day of the week.

In the cells of the table, enter the time(s) of day when the reservoir should make a release decision. The time entries must be entered as values from a 24-hour clock—that means values between 0000 and 2359. And, you should only enter times that coincide with compute intervals.

NOTE: When entering times into the Weekly Schedule, a whole row must be complete. That means that if one day a week you want 2 decisions made, then every day, two decisions must be made; however, the times of each decision on each day need not be the same. Figure 12.52 illustrates an example of an irregular Weekly Schedule with two decisions schedule each day.

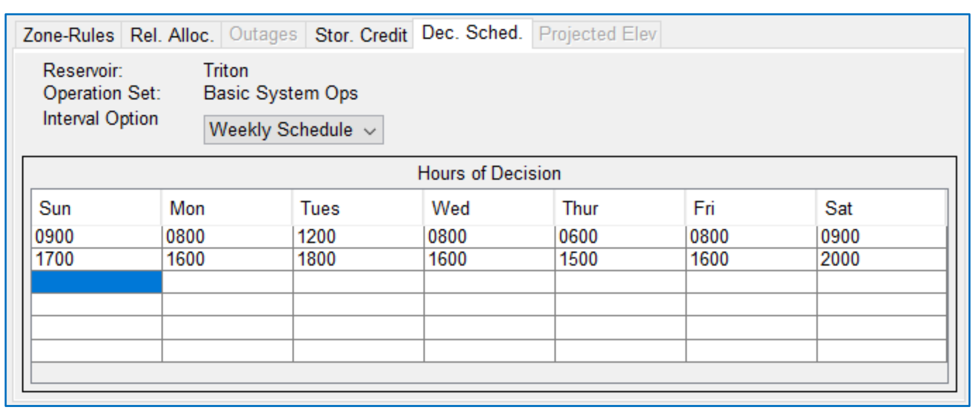


Figure 12.52 Decision Schedule Edit Panel—Weekly Schedule Option

- When you are finished specifying the **Decision Schedule**, *click* **Apply** to accept the **Decision Schedule** and keep the **Reservoir Editor** open. Or, *click* **OK** to accept the revisions and close the **Reservoir Editor**.

12.8 Projected Elevation

Projected Elevation is the last of the advanced operations features that can be added to an operation set by activating it from the **Operations** menu of the **Reservoir Editor**.

This feature acts on the current reservoir and has two parts: 1) it turns on the computation of a projected pool elevation at the end of a specified projection window, and 2) it gives the option to use that projected elevation (instead of the *current* pool elevation) to determine the *active* zone whose rules will be used to influence the reservoir operation (release).

When this option is activated in the **Operations** menu and a **Projection Time Window** has been specified, ResSim will compute a projected reservoir elevation at each timestep of the compute window. You can then use the projected elevation like any other model variable—it can be referenced in a rule, zone definition, IF_Block, diversion, state variable, or scripted rule to influence the operation of the reservoir.

The primary purpose of the Projected Elevation feature is to use the projected elevation to determine the *active* operation zone. This feature was designed as a flood operation to allow less restrictive rules from a higher zone to be used before the pool actually reaches that higher zone. The objective is to prevent or delay the pool from reaching the higher zone.

To *activate* the **Projected Elevation** feature:

- Open the **Operations** tab of the **Reservoir Editor** and select a reservoir.
- From the **Operations** menu, select **Compute Projected Elevation.**

  A check mark will appear in front of the **Compute Projected Elevation** option in the **Operations** menu to indicate that the **Projected Elevation** feature is ON (Figure 12.53), and the **Projected Elev** tab will become active (Figure 12.54).

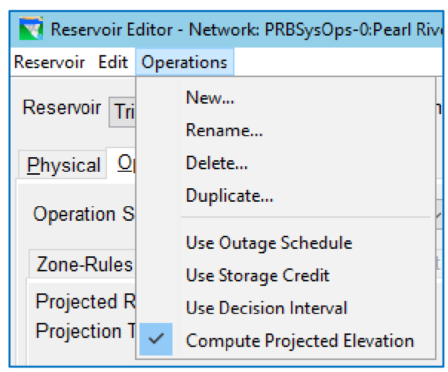


Figure 12.53 Operations Menu—Compute Projected Elevation

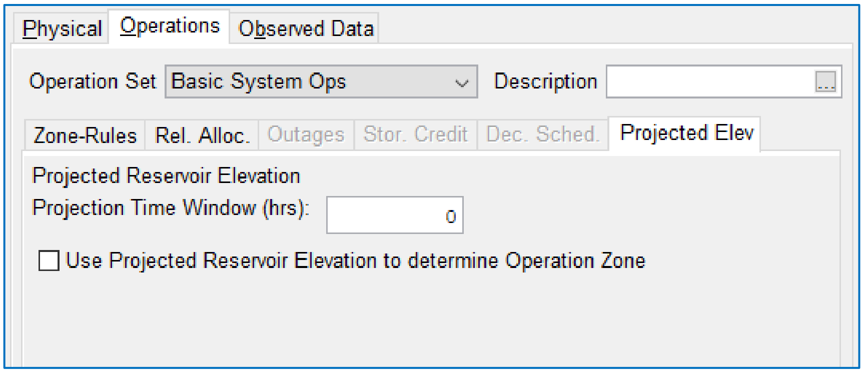


Figure 12.54 Reservoir Editor—Projected Elevation Sub-tab

The **Projected Elev** sub-tab has only two fields for you to set:

Projection Time Window—the number of hours into the future that you want ResSim to estimate the reservoir pool elevation. To perform this computation, ResSim will assume that the release it was making at the end of the previous timestep will be held constant through the Projection Time Window. Note: If you leave 0 in this field, ResSim will simply compute an estimate of the pool elevation at the end of the current timestep assuming that no change was made to the release in this timestep.

Use Projected Reservoir to determine Operation Zone—To use the projected elevation to determine the active zone, place a checkmark in the checkbox. To use the projected elevation in an IF_Block, rule, or script, you do not need to check this box.

12.9 The Reservoir Network Importers

ResSim has two import tools for bringing network data from another model into your current watershed. The first is the **Import Network Wizard**. This tool allows you to copy a network from one watershed into the current watershed. It will step you through the process of identifying the network you want to copy and resolving the associations between each network element and the reciprocal for the current stream alignment or watershed configuration.

The second import tool is the **Import Element Properties Wizard**. This tool is very similar to the Import Network Wizard, but it does not create a new network in your watershed; instead, it copies data from elements in one network into the elements in your current network. The **Import Element Properties Wizard** will step you through the process of selecting the network from which you are going copy the element data and resolving the connectivity to the other elements and their model variables.

PLEASE NOTE: Before you can use the Importers…

Both Import Wizards require that you have one or more Watershed Locations defined for your ResSim installation. A *Watershed Location* is now named Shortcut and it points to a folder on your computer where you have one or more watershed(s) stored. The Importers can only find watersheds that are located in one of these Watershed Locations.

The first section of Appendix A describes how to create and manage Shortcuts.

12.9.1 The Import Network Wizard

The purpose of the **Import Network Wizard** (or Network Importer) is to copy a network from one watershed into another. This is usually needed when an existing watershed is copied and used for a different purpose or study than that of the original model. During its use in the newer study, the model may have evolved and been enhanced in ways that could be useful in the context, or for the purpose, of the original watershed.

However, since the new (copied) watershed "evolved", there may be differences in the underlying framework (the stream alignment and/or configuration) of the network to be imported that must be addressed in order for that network to function as a native part of the watershed it is being imported into.

In addition, there's nothing to prevent you from importing a network from a watershed that did not share its development path with the watershed it is being imported into. The source watershed could be older or lacking in some way (or even broken) and by importing its network into a newer or more complete watershed, the older network can be updated and used as part of the new watershed.

The steps for importing a Reservoir Network into your current watershed are as follows:

- From the Reservoir Network module, select **Import Network...** from the **Network** menu (Figure 12.55).

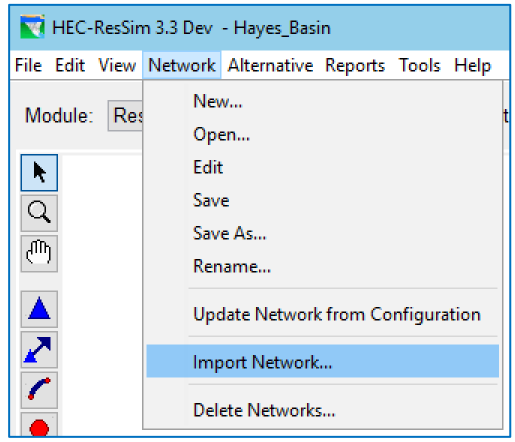


Figure 12.55 Network Menu—Import Network...

- The **Import Network Wizard** will open (Figure 12.56). In Step 1 of 6, select the watershed to *copy from*. The Watershed Locations area shows the list of your Shortcuts to directories on your computer. Highlighting each will show a list of watersheds in the right-hand side. Select the desired watershed to copy *from* and click **Next**.

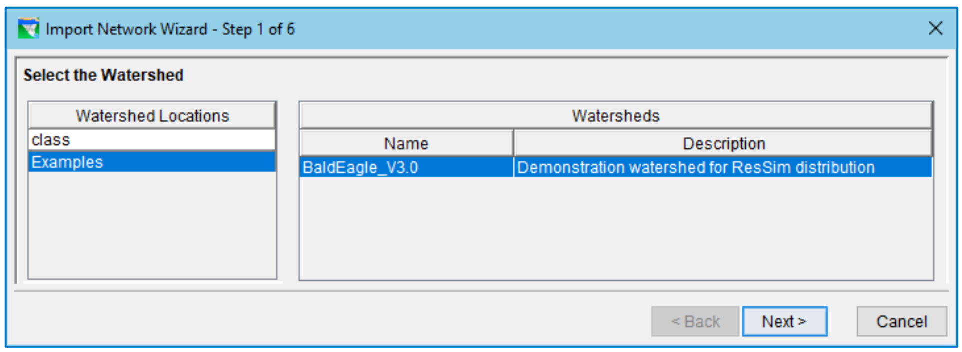


Figure 12.56 Import Network Wizard—Step 1—Select the Watershed

- In the **Import Network Wizard—Step 2 of 6**, a list of available networks in your chosen watershed will be shown (Figure 12.57). Select the network you wish to *copy into* your current watershed and click **Next**.

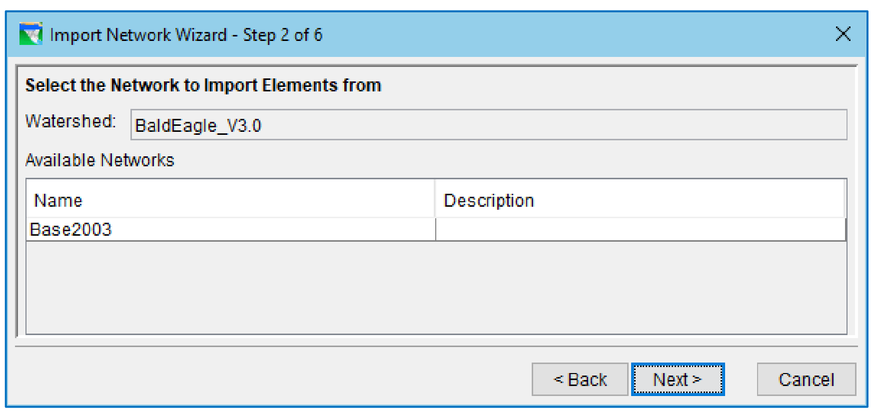


Figure 12.57 Import Network Wizard—Step 2—Select the Network to Import Elements from

- Set the new network name and description in the **Import Network Wizard—Step 3 of 6** (Figure 12.58). All networks in your current watershed are listed under

Existing Reservoir Networks. At the bottom, under New Reservoir Network, create a new name and description for the imported network.

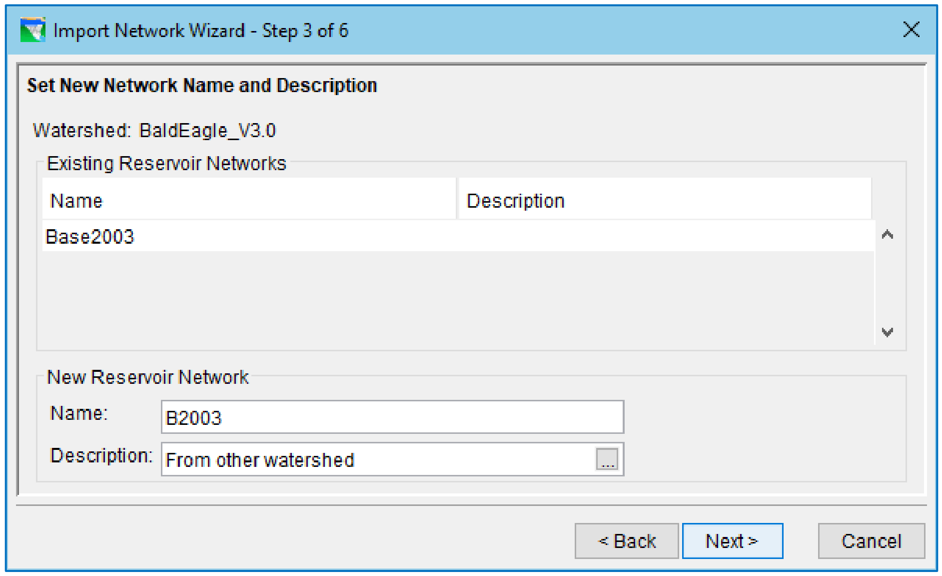


Figure 12.58 Import Network Wizard—Step 3—Set New Network Name and Description

- Click **Next** to open the **Import Network Wizard—Step 4 of 6** (Figure 12.59). The Assign Stream Names dialog shows a table listing each Imported Element, Element Type, and its location (From Stream) in the originating ("import from") watershed. The final column "To Stream" gives the location to which the element will be imported in the current watershed. ResSim will automatically

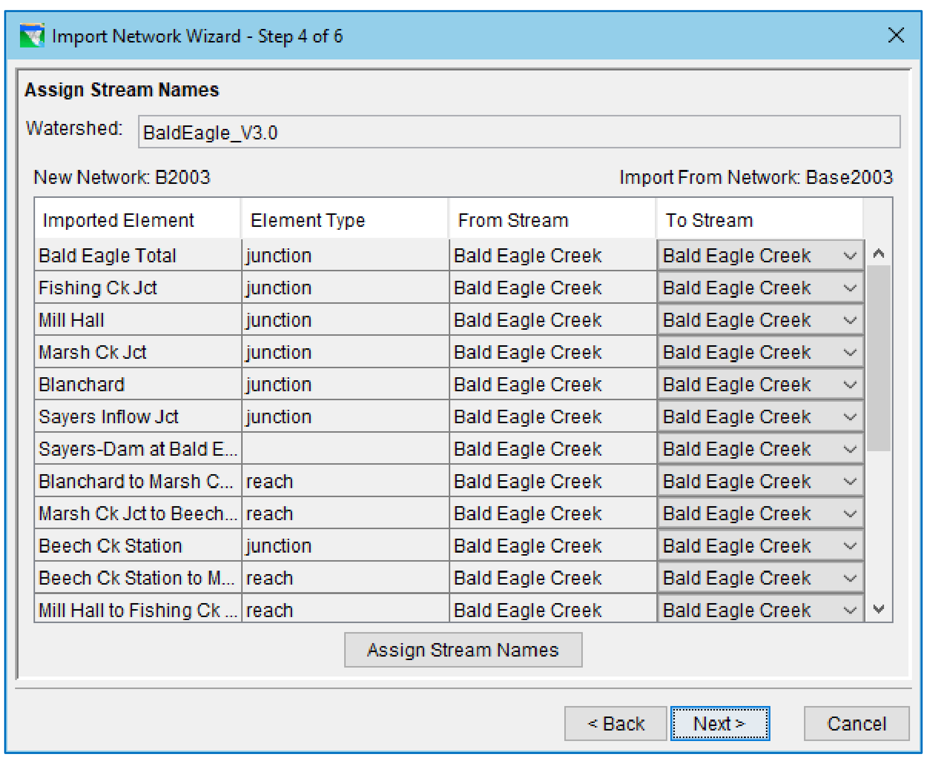


Figure 12.59 Import Network Wizard—Step 4—Assign Stream Names

populate the "To Stream" fields, if it can match a stream name in the "import from" watershed to a stream name in the current "import to" watershed. Update the field using the dropdown menu to select from the list of streams in the current watershed.

- You can also use the **Assign Stream Names** button to associate streams in the "copy from" watershed with streams in the "copy to" watershed. Clicking this button will open the **Select Stream Name** editor (Figure 12.60). Once you have associated the streams, click **OK**, and this will update all the imported element "To Stream" fields accordingly.

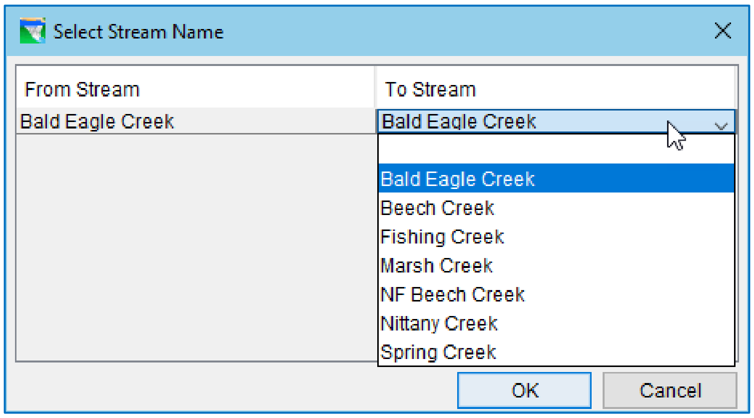


Figure 12.60 Import Network Wizard—Select Stream Name

- Click **Next** to go to the **Import Network Wizard—Step 5 of 6** (Figure 12.61), where you will Resolve the Network Computation Points. The table lists every Imported Element name and type and indicates the associated current Watershed Computation Point. Verify that these are correct, or if they are blank, select them with the dropdown menu. Sometimes an imported element can't be matched to a similar location in the current watershed. When this happens, consider how the two watersheds differ and whether you need to make a change before importing. If the computation point can't be resolved, the associated element will not be imported. Click **Next**, and if there were issues, you will see a warning with a list of any import elements that were not assigned computation points and will therefore not be imported into your current watershed. You can choose the **Back** button if necessary.

- The **Import Network Wizard—Step 6 of 6** shows a summary of all the import information (Figure 12.62). Obvious issues will be in red font. Once you are satisfied with the summary, click **Finish** to complete the import, or select **Back** to make changes in previous steps. If you are sure about the import, confirm by selecting **Yes** in the **Continue with Import** dialog. Choosing **No** will return you to the Wizard's **Step 6 of 6**.

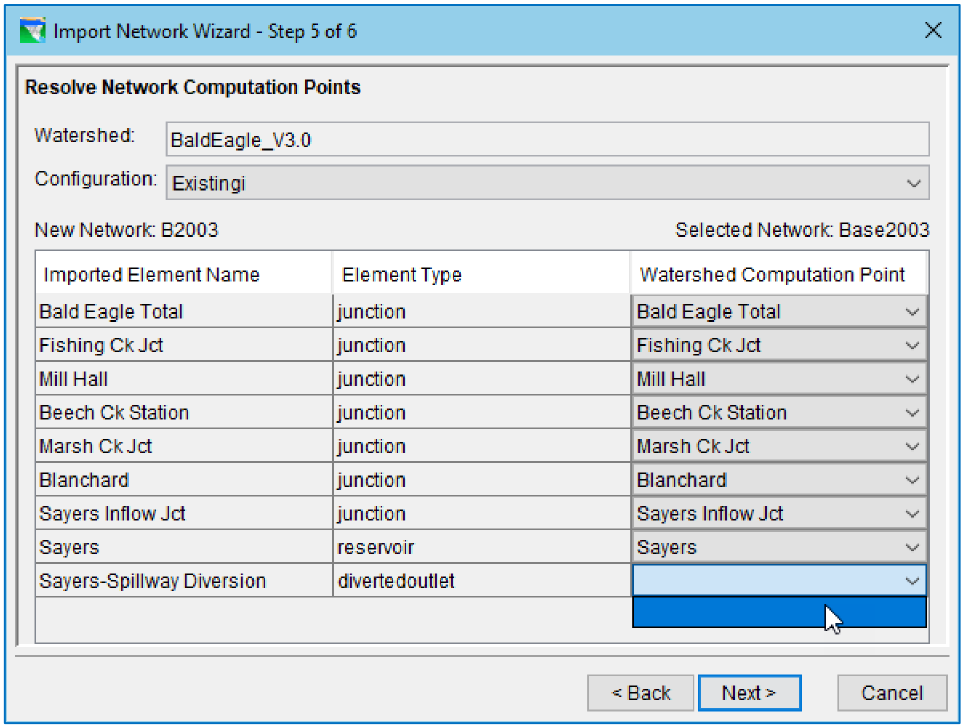


Figure 12.61 Import Network Wizard—Step 5—Resolve Network Computation Points

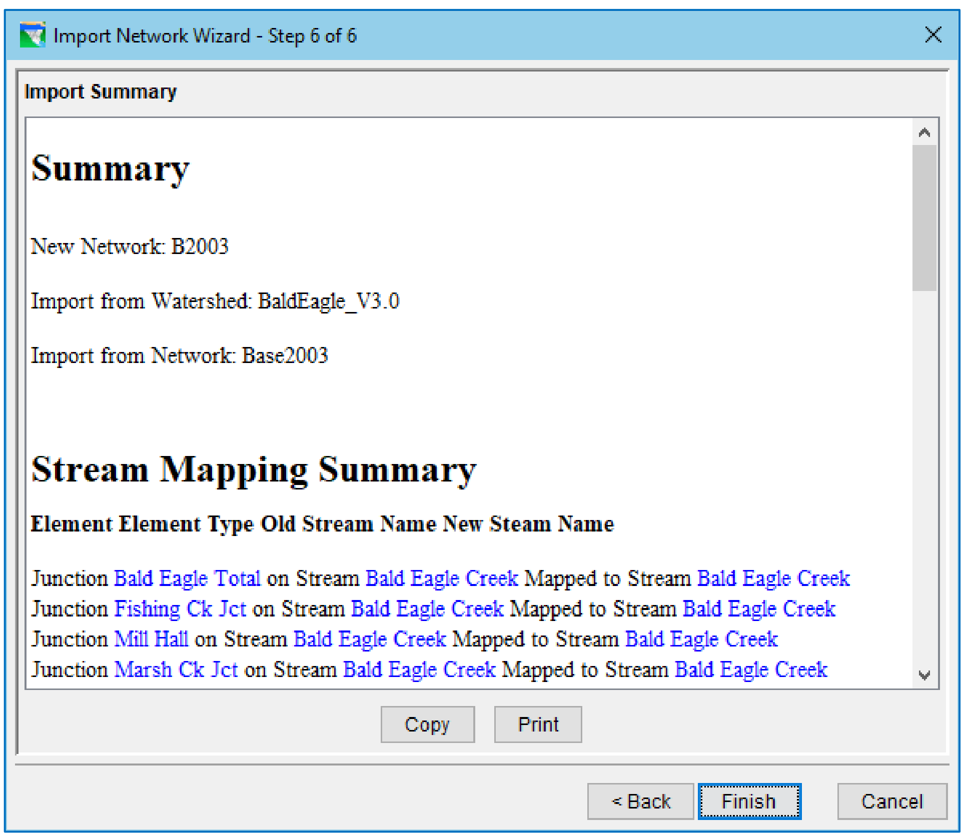


Figure 12.62 Import Network Wizard—Step 6—Import Summary

12.9.2 The Import Element Properties Wizard

After network elements have been drawn and labeled, physical and operational data can be imported from another network using the **Import Element Properties Wizard**. Data that can be imported include:

- reservoir storage and outlet capacity
- reservoir operation sets and rules
- reach routing method and parameters
- junction data, and
- diversion data.

To use the import wizard, open the Network that data will be *imported into*. Then, perform the following six steps:

- From the **Edit** menu in the Reservoir Network module, select **Import Element Properties….** **The Import Element Properties Wizard—Step 1 of 6** will appear (Figure 12.63).

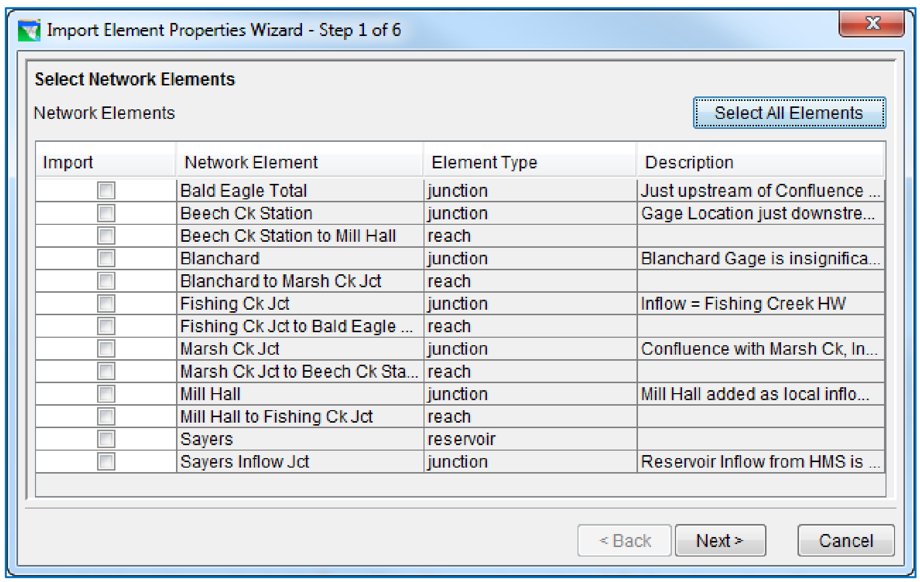


Figure 12.63 Import Element Properties Wizard—Step 1—Select Network Elements

- Select the network elements that you would like to import physical and operational data *into* from an existing network by checking the **Import** boxes to the left of the **Network Element** names or by clicking **Select All Elements** located in the upper right region of the editor.

- Select **Next** to continue and the **Import Element Properties Wizard—Step 2 of 6** will appear (Figure 12.64). Select the watershed that contains the network *from which* you would like to import 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. When the desired watershed is highlighted, click **Next**.

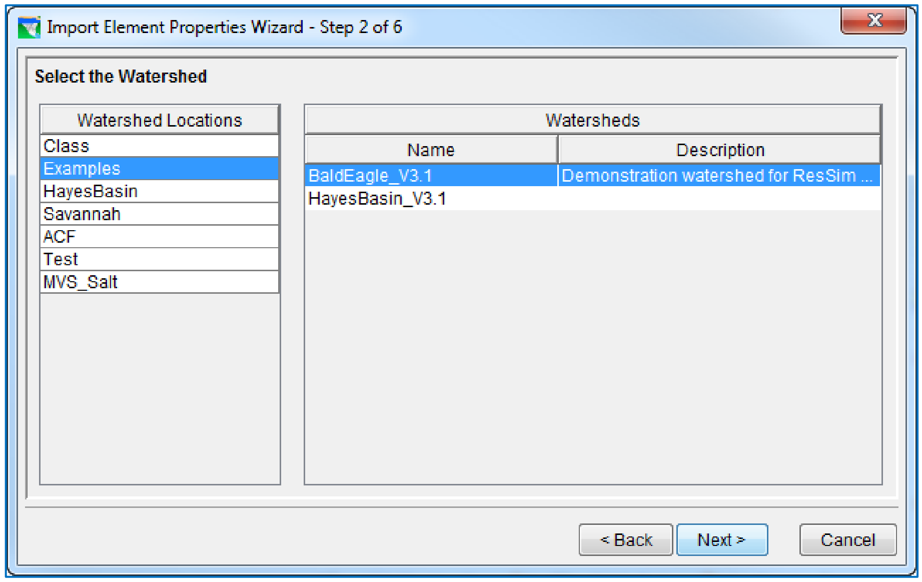


Figure 12.64 Import Element Properties Wizard—Step 2—Select Watershed to Import Data from

- Select **Next** to continue and the **Import Element Properties Wizard—Step 3 of 6** will appear (Figure 12.65). Select the specific network *from which* you would like to import physical and operational data.

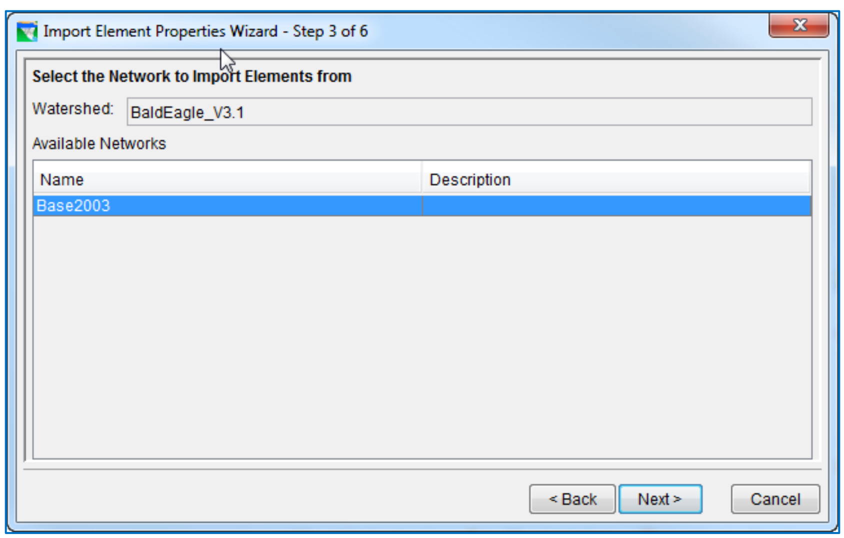


Figure 12.65 Import Element Properties Wizard—Step 3—Select Network to Import Elements from

- Select **Next** to continue and the **Import Network Elements Wizard—Step 4 of 6** will appear (Figure 12.66). Assign the element properties to be imported by matching the appropriate elements in the **Import From** column with their corresponding elements in the **Import Into** column.

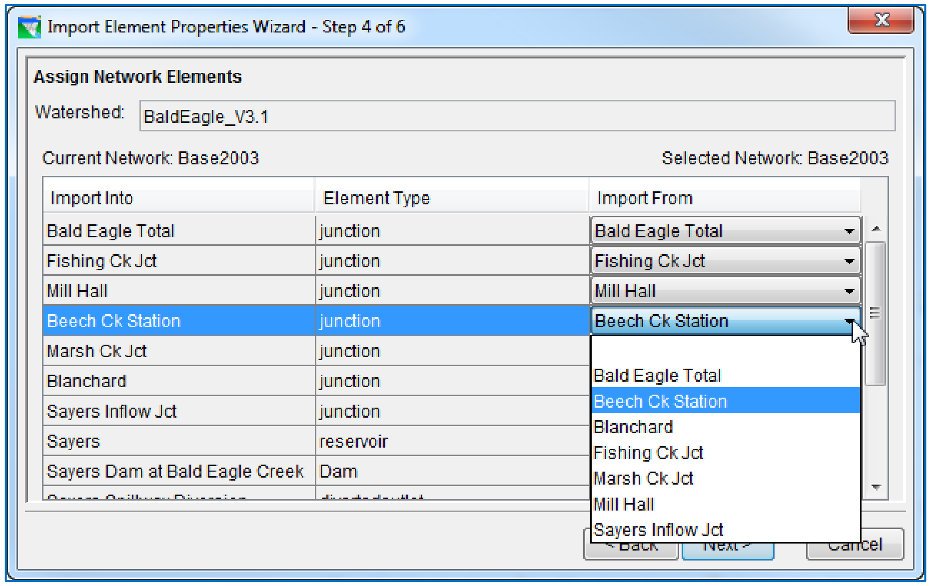


Figure 12.66 Import Element Properties Wizard—Step 4—Assign Network Elements

- Select **Next** to continue and the **Import Element Properties Wizard—Step 5 of 6** will appear (Figure 12.67). Resolve network connectivity by choosing the element in the **Select Network Element** column that you would like the corresponding rule to be applied to. In most cases, what is chosen in the **Select Network Element** column will match the network element in the **References Element** column.

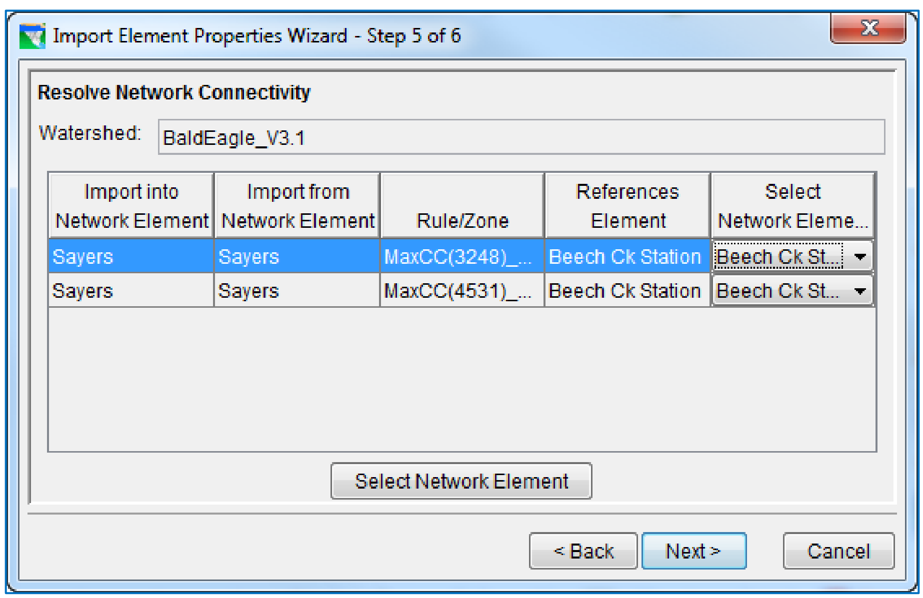


Figure 12.67 Import Element Properties Wizard—Step 5—Resolve Network Connectivity

- Select **Next** to continue to the **Import Element Properties Wizard—Step 6 of 6** (Figure 12.68). Resolve network connectivity by choosing the element in the **Select Network Element** column that you would like the corresponding rule to be applied to. In most cases, what is chosen in the **Select Network Element** column will match the network element in the **References Element** column.

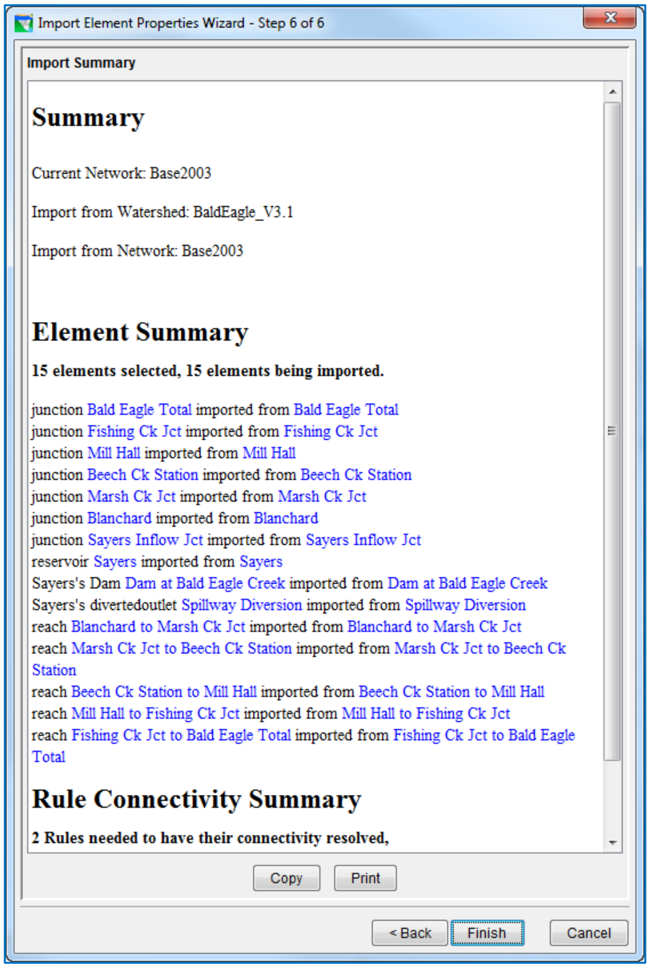


Figure 12.68 Import Element Properties Wizard—Step 6—Import Summary

- Select **Finish** to carry out the import, or select **Back** to make changes in the previous steps. If you are sure about the import, confirm by selecting **Yes** in the **Continue with Import** dialog (Figure 12.69). Choosing **No** will return you to the Wizard's **Step 6 of 6**.

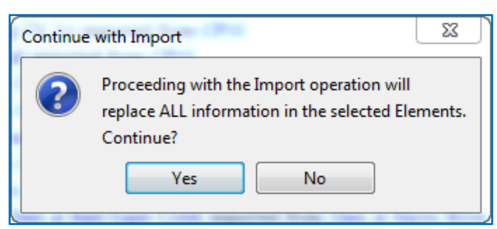


Figure 12.69 Continue with Import

- After finishing with the Element Properties Import process, a dialog will appear summarizing the results of the import carried out. Check to ensure that all physical and operational data was successfully imported by reviewing the **Import Results** summary (Figure 12.70) and by viewing the various network element editors in the Reservoir Network module.

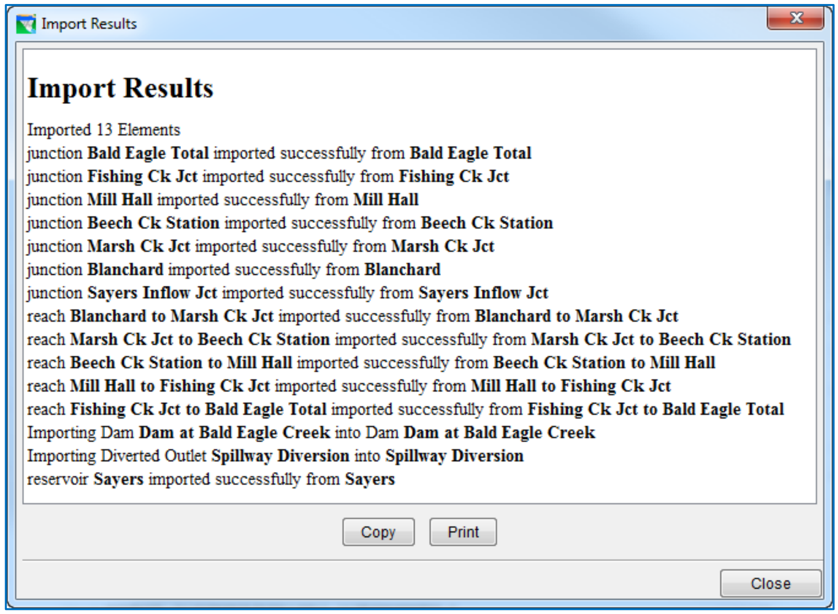


Figure 12.70 Import Results

- Select **Close** to complete the process of Importing Element Properties from one Watershed/Network into another.

Chapter 13 Reservoir Systems

A **Reservoir System** in ResSim is defined as two or more reservoirs that operate together for common goals. Reservoir systems are created by defining system operation rules for two or more reservoirs. ResSim provides for tandem operation to manage the storage distribution between upstream and downstream reservoirs on the same stream. Tandem operations are created by applying a tandem rule at an upstream reservoir operating for a downstream reservoir (i.e., two reservoirs in series). In addition, ResSim supports parallel operation of reservoirs, where two or more reservoirs on different streams control for common downstream requirements, through the use of common downstream control (for flow or stage limit) rules. For each individual reservoir, system operation rules are prioritized among other rules in the operation set. (Refer to Section 11.1.3 for a more detailed description of rule prioritization.) This chapter will present the concept of system operation, specifically implicit and explicit methods for determining the system balance, and provide guidance for using the Reservoir System Editor to set up explicit system storage balances.

13.1 Concept of Reservoir Systems

When a tandem or parallel reservoir system is defined, the model determines the priority and the amount of release to make from each reservoir in order to operate towards a storage balance among the system reservoirs. For every decision interval, an end-of-period storage is first estimated for each reservoir based on the sum of beginning-of-period storage and period average inflow volume, minus all potential outflow volumes. The estimated end-of-period storage for each reservoir is compared to a desired storage that is determined by using a system storage balance scheme. The priority for release is then given to the reservoir that is *furthest above* the desired storage. When a final release decision is made, the end-of-period storages are recomputed. Depending on other constraints or higher priority rules (Chapter 12), system operation strives for a storage balance such that the reservoirs have either reached their Guide Curves or they are operating at the **desired storage** (percent of the active storage zone).

The storage balance across reservoir systems is based on the definition of **System Storage Zones**. These zones divide each system reservoir into comparable zones, across which to balance. There are two methods by which the desired storage balance is determined: <u>implicit</u> (default) and <u>explicit</u> (user-defined). The implicit method delineates the *default* storage balance scheme for the reservoir system. The explicit method is optional and allows a *user defined* storage balance scheme for the reservoir system. Detailed descriptions and examples are presented in Sections 13.1.1 and 13.1.2 and demonstrating the functionality of implicit and explicit storage balance methods.

13.1.1 Implicit System Storage Balance Method

The *default* method in ResSim for determining the desired storage balance in a reservoir system is referred to as the *implicit* method. This method applies to both tandem and parallel system operations. The implicit method is automatically used when a reservoir system is established – either by using a common **Downstream Control** rule in two or more parallel reservoirs, or adding a **Tandem Operation** rule to an upstream reservoir operating for a downstream reservoir.

For example, consider a two-reservoir tandem system, as shown in Figure 13.1. Reservoir 1 is the upstream reservoir where a **Tandem Operation** rule has been applied in its operation set, as shown in Figure 13.2. (See Section 11.9 for details about adding the Tandem Operation rule.) This establishes an implicit system operation with the downstream reservoir, Reservoir 2. Assume that each reservoir has the same amount of storage capacity (100,000 ac-ft). For each of the reservoirs, the Guide Curve has been set to be the top of Conservation zone. (See Section 10.4 for instructions on setting the Guide Curve.) The conservation storage in Reservoir 1 is 75,000 ac-ft, whereas the conservation storage in Reservoir 2 is 30,000 ac-ft.

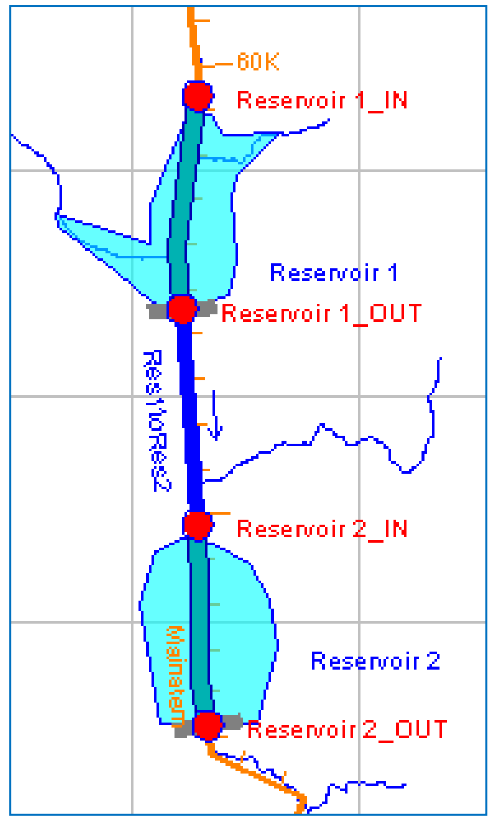


Figure 13.1 Example of a Two-Reservoir Tandem System

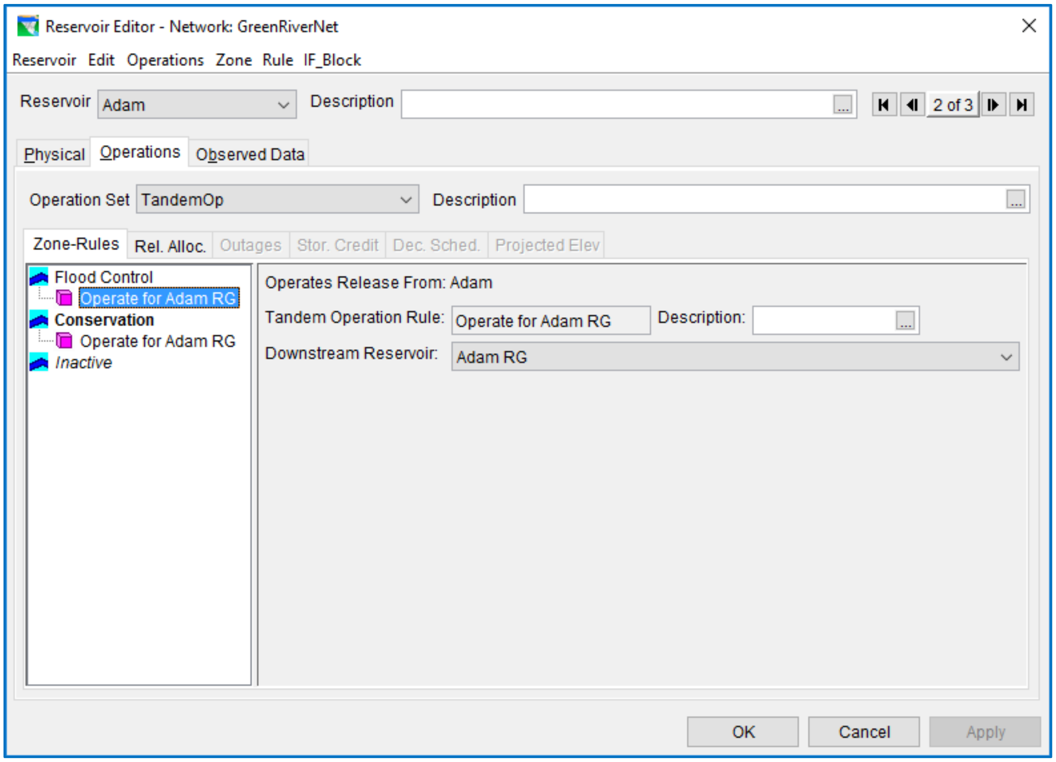


Figure 13.2 Tandem Operation Rule Included in Upstream Reservoir

The implicit system storage balance scheme (illustrated in Figure 13.3) takes into account the **System Storage** (the total storage from the reservoirs in the system). In this example, the system storage ranges from empty (0 ac-ft) to full (200,000 ac-ft). Additionally, this default scheme considers only one **System Storage Zone**, the System Guide Curve (Sys G.C.) storage, which amounts to the sum of both reservoirs' conservation storages (105,000 ac-ft).

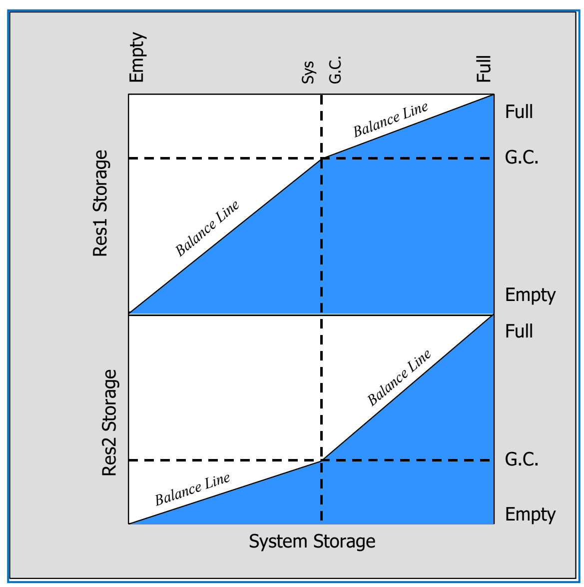


Figure 13.3 Implicit System Storage Balance

The **desired storage** for each reservoir is determined through an implicit "balance line". The balance line is simply a linear relationship between storage at each reservoir and the system storage. For each reservoir, the balance line hinges on the intersection of the reservoir's Guide Curve (G.C.) storage and the System Guide Curve (Sys G.C.) storage. For system storage *less than* the System Guide Curve storage, the balance line has a lower limit that corresponds to empty storage at the reservoir versus empty system storage, and the upper limit corresponds to Guide Curve storage at the reservoir (75,000 ac-ft at Res. 1 and

30,000 ac-ft at Res. 2) versus System Guide Curve storage (105,000 ac-ft). For system storage *greater than* the System Guide Curve storage, the lower limit of the balance line corresponds to Guide Curve storage at the reservoir (75,000 ac-ft at Res. 1 and 30,000 ac-ft at Res. 2) versus System Guide Curve storage (105,000 ac-ft), and the upper limit corresponds to full storage at the reservoir versus full system storage.

At the end of each decision interval (i.e., end-of-period), the **desired storage** for a reservoir corresponds to a point on the balance line that coincides with the sum of the estimated storages for both reservoirs. When the total estimated storage from both reservoirs is *less than* the System Guide Curve storage, the corresponding desired storages represent an equal percentage of the storage *below* the Guide Curve at each reservoir. When the total estimated storage from both reservoirs is *greater than* the System Guide Curve storage, the corresponding desired storages represent an equal percentage of the storage *above* the Guide Curve at each reservoir.

For instance, as shown in Figure 13.4, assume that preliminary end-of-period storage estimates are 25,000 ac-ft for Reservoir 1 and 45,000 ac-ft for Reservoir 2. The resultant total system storage of 70,000 ac-ft coincides with each reservoir's desired storage (50,000 ac-ft for Reservoir 1 and 20,000 ac-ft for Reservoir 2) found along the balance line from empty system storage to System Guide Curve storage. These desired storage values signify a desired balance because they amount to an equal percent (66.7%, in this case) of the Guide Curve storage at each reservoir: 50,000 of 75,000 ac-ft at Reservoir 1, and 20,000 of 30,000 ac-ft at Reservoir 2.

With 25,000 ac-ft estimated as its end-of-period storage, Reservoir 1 would be below its desired storage of 50,000 ac-ft. On the other hand, at an estimated storage of 45,000 ac-ft, Reservoir 2 would be above its desired storage of 20,000 ac-ft.

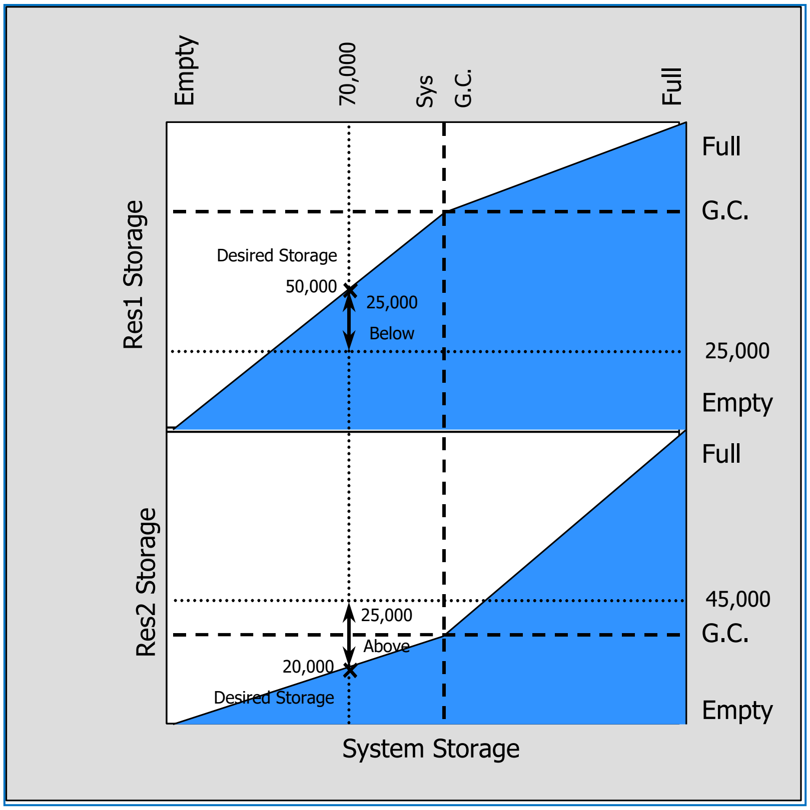


Figure 13.4 Example of Desired Storages using the Implicit System Storage Balance Method

Since Reservoir 2 is above its desired storage, it receives the priority to release for this period in order to drop its storage down, as close as possible, to the desired storage. Unless other constraints (such as maximum physical outlet capacity, maximum flow limit rules, or flow rate of change limit rules) restrict releases and have higher priority than the system operation rule, Reservoir 2 would increase its releases in order to drop its pool to the desired storage of 20,000 ac-ft. As for Reservoir 1, it is forced to cut back its releases so that its storage can rise, as close as possible, to its desired storage of 50,000 ac-ft. If there are no restrictions (such as minimum flow limit rules or flow rate of change limit rules) that could require a different release due to having a higher priority than the system operation rule, Reservoir 1 would stop releasing from its outlet(s).

In the implicit system operation, a release decision made for a particular time period may not necessarily achieve the desired balance. The reservoirs in the system are considered "in balance" when both reservoirs have reached their Guide Curves, or they are operating at equivalent storage levels in terms of percentage of their counterpart system storage zones.

13.1.2 Explicit System Storage Balance Method

The *user-defined* method in ResSim for determining the desired storage balance in a reservoir system is referred to as the *explicit* method. This method can be used for an established reservoir system, whether tandem or parallel. For example, consider two parallel reservoirs (Reservoir 1 and Reservoir 2) operating for a common downstream location (MyTown), as shown in Figure 13.5. In this case, both reservoirs are operating for a common downstream location, and each has the same downstream control rule applied in its operation set (establishing an implicit system operation).

As described in Section 12.1.1, the implicit scheme by default develops balance lines, using a single system storage zone (System Guide Curve), to define linear relationships between storage at each reservoir and the total system storage. The user can further modify these balance lines explicitly to characterize the desired storage distributions using one or more system zones and placing inflection points along the balance line.

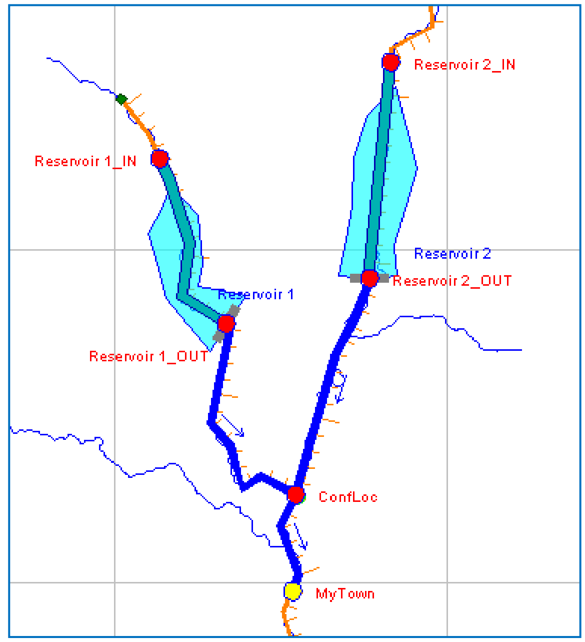


Figure 13.5 Example of a Two-Reservoir Parallel System

For this example, both reservoirs have the same storage characteristics, maximum storage capacities and conservation storage as described in the tandem example in Section 13.1.1. Additionally, for Reservoir 1, the top of the Flood Control zone is at a storage of 85,000 ac-ft. For Reservoir 2, the top of Flood Control is at a storage of 65,000 ac-ft.

Figure 13.6 shows an explicit scheme defined such that Reservoir 1 fills its conservation zone more rapidly than Reservoir 2, and Reservoir 2 fills its flood control zone at an initially faster rate than Reservoir 1. This is accomplished by first identifying two system zones. For instance, System Conservation would represent one system zone that is the aggregate of the conservation storages from the two reservoirs. The other system zone would be the System Flood Control zone, the total of both reservoirs' flood control storages. As shown in Figure 13.6 and summarized in Table 13.1, a customized desired storage balance can be made by introducing inflection points to the balance lines within each system zone. Inflection points would transform the implicit balance line into an explicit curve. The inflection points allow the slope of the line, or the relationship between individual reservoir storage and system storage, to vary. An unlimited number of balance line inflection points could be added within each system zone to further refine and shape the desired balance distribution.

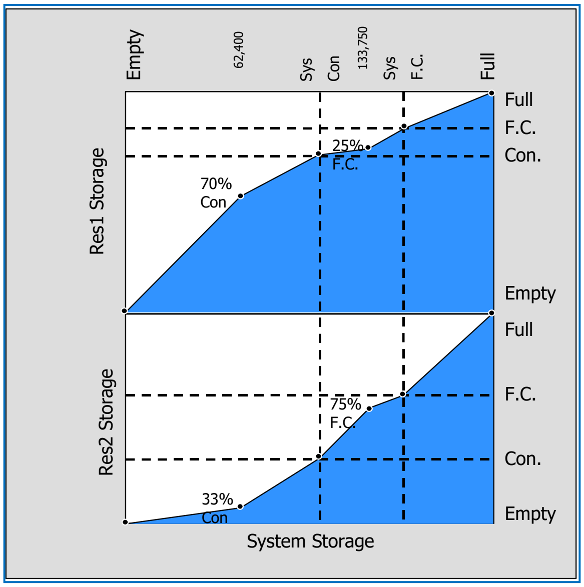


Figure 13.6 Explicit System Storage Balance

Table 13.1 Explicit System Storage Balance

Storage	Reservoir 1	Reservoir 2	System Storage
Full	100,000 ac-ft	100,000 ac-ft	200,000 ac-ft
F.C.	85,000 ac-ft	65,000 ac-ft	150,000 ac-ft
%F.C.	(25% F.C.) 77,500 ac-ft	(75% F.C.) 56,250 ac-ft	133,750 ac-ft
Con	75,000 ac-ft	30,000 ac-ft	105,000 ac-ft
%Con	(70% Con) 52,500 ac-ft	(33% Con) 9,900 ac-ft	62,400 ac-ft
Empty	0 ac-ft	0 ac-ft	0 ac-ft

In this example, within the System Conservation zone, balance line inflection points are set at 70 percent of the conservation storage (52,500 ac-ft) for Reservoir 1 and 33 percent of the conservation storage (9,900 ac-ft) for Reservoir 2. As a result, these inflection points coincide with system storage of 62,400 ac-ft, and reshape their respective balance line curves according to the general criterion that Reservoir 1 fills up its conservation zone to 70 percent in the time Reservoir 2 fills to only 33 percent. Similarly within the Flood Control System zone, balance line inflection points set at 25 percent of the flood control storage (77,500 ac-ft) for Reservoir 1 and at 75 percent of the flood control storage (56,250 ac-ft) for Reservoir 2 coincide with system storage of 133,750 ac-ft. This would satisfy the requirement that Reservoir 2 fills up its flood control zone faster than Reservoir 1.

As demonstrated in Figure 13.7, for estimated end-of-period storages of 25,000 ac-ft at Reservoir 1 and 45,000 ac-ft at Reservoir 2, the resultant system storage of 70,000 ac-ft coincides with desired storages found along the explicitly defined balance line curves within the System Guide Curve storage zone. The desired storage levels are 56,500 ac-ft for Reservoir 1 and 13,500 ac-ft for Reservoir 2.

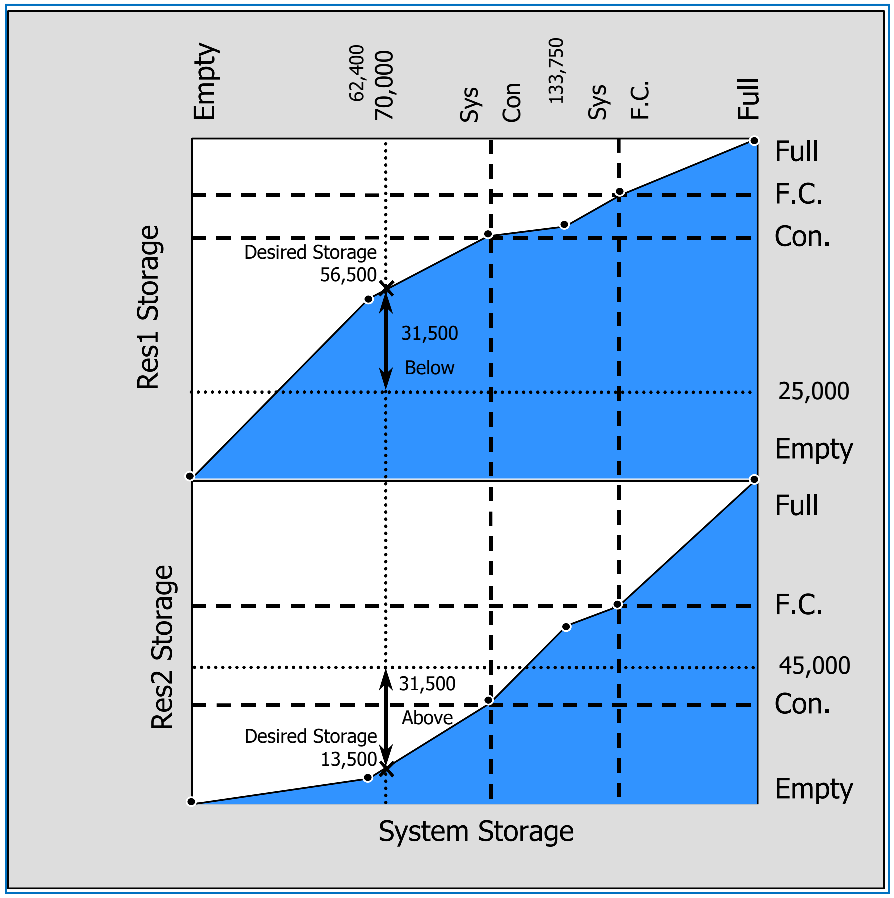


Figure 13.7 Example of Desired Storages using the Explicit System Storage Balance Method

At 45,000 ac-ft, Reservoir 2 would be above its desired storage of 13,500 ac-ft. As such, Reservoir 2 receives the priority to release for this period. Unless other constraints restrict releases and have higher priority than the system operation rule, Reservoir 2 would increase its releases in order to reduce its storage, as close as possible, to the desired storage. On the other hand, with only 25,000 ac-ft of estimated storage, Reservoir 1 would be below its desired storage of 56,500 ac-ft. Then Reservoir 1 is forced to cut back its releases for this particular time period so that its storage can rise, as close as possible, to the desired level. If there are no higher priority rules that require a release, Reservoir 1 would not make a release from its outlet(s).

Similar to the implicit system operation, the explicit system operation is carried out each time period when system rules are in effect. The process of determining desired storages is repeated every decision interval in order to assign the priority for release to the reservoir that is farthest above the desired storage. A release decision made for a particular time period may not necessarily achieve the desired balance. The reservoirs are considered "in balance" when both reservoirs have reached their Guide Curves or are operating at the desired storages levels along their balance line curves as prescribed in the explicit storage balance scheme.

The user interface process of creating an explicit system storage balance is described in subsequent sections.

13.2 Overview of the Reservoir System Editor

The **Reservoir System Editor** is used to create explicit system storage balances for selected reservoir systems. The editor is very similar to the **Operations** tab of the **Reservoir Editor** (Section 10.1). An example of the **Reservoir System Editor** is shown in Figure 13.8 and reflects the example data for the explicit storage balance method previously discussed in Section 13.1.2.

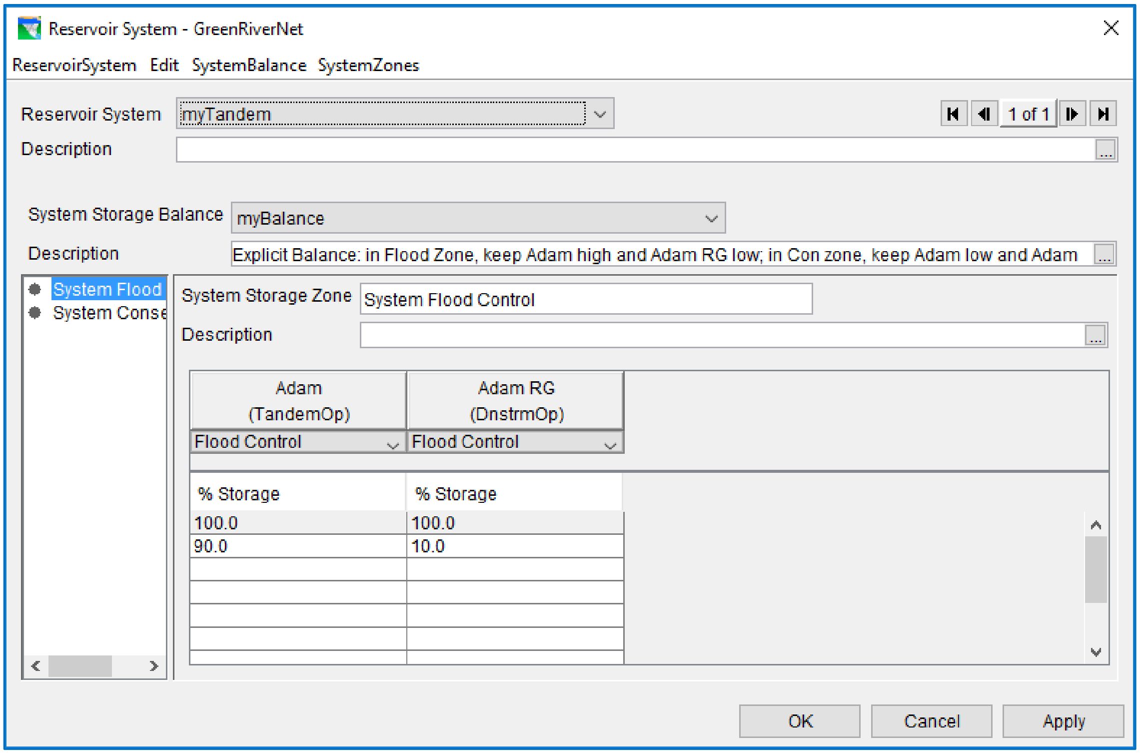


Figure 13.8 Reservoir System Editor—New Reservoir System

As previously discussed, a reservoir system is implicitly created when two or more reservoirs are operating in tandem or when parallel reservoirs are operating together for a common downstream location. For system operations, you can either accept the implicit default system storage balance or you can create and define one or more explicit **System Storage Balance** schemes. For each system storage balance scheme you develop, you must define the **System Storage Zones** and the distribution of storage across the individual reservoirs (similar to the concept of **Reservoir Operation** sets discussed in Section 10.2).

The remainder of this chapter will discuss the **Reservoir System Editor** in detail and will provide instructions for specifying explicit system storage balance data.

13.3 Accessing the Reservoir System Editor

To specify the explicit reservoir system balance scheme, you will use the **Reservoir System Editor** (previously shown in Figure 13.8). This editor is available in the **Reservoir Network Module** and is accessed from the **Edit** menu (in the menu bar), as shown in Figure 13.9.

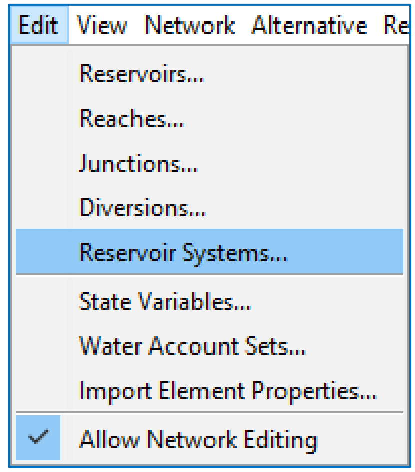


Figure 13.9 Reservoir Network Module—Edit Menu—Reservoir Systems

13.4 Reservoir System Editor Menu Items

At the top of the **Reservoir System Editor**, the **Menu Bar** includes four menus unique to this editor. The menus are **ReservoirSystem**, **Edit**, **SystemBalance**, and **SystemZones**. These menus provide the following options: creating, renaming and deleting reservoir systems; editing reservoir sets; creating, renaming and deleting system operation sets; and creating and deleting system zones. These options are presented in the following paragraphs and described in subsequent sections of this chapter.

- The **ReservoirSystem** menu (Figure 13.10) allows you to:
    - create a **New** system,
    - **Rename** a system,
    - **Delete** a system, and
    - **Close** the editor.

  Refer to Section 13.5 for additional information.

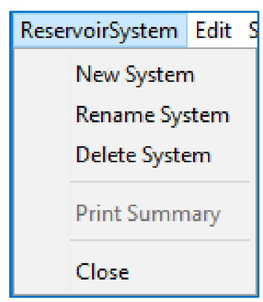


Figure 13.10 Reservoir System Editor—ReservoirSystem Menu

- From the **Edit** menu (Figure 13.11):
    - select **Edit Reservoir Set** to specify which reservoirs are to be included in the reservoir system.

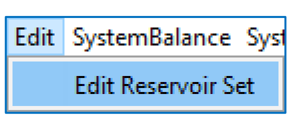


Figure 13.11 Reservoir System Editor—Edit Menu

- The **SystemBalance** menu (Figure 13.12) allows you to:
  - create a **New** system storage balance,
  - **Rename** a system storage balance, or
  - **Delete** a system storage balance.

  Before you can create a new system balance, you must first define which reservoirs are included in the system (see previous paragraph for instructions to edit the reservoir set).

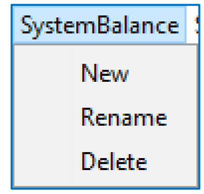


Figure 13.12 Reservoir System Editor—SystemBalance Menu

- The SystemZones menu (Figure 13.13) allows you to:
  - create **New** system zones, or
  - **Delete** system zones and is active only after a system balance has been created.

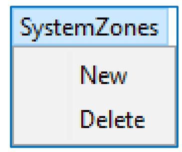


Figure 13.13 Reservoir System Editor—System Zones Menu

13.5 Defining a New Reservoir System

The **Reservoir System Editor** is used for creating a Reservoir System with explicit system storage balances. The process of setting up a new reservoir system includes defining a new reservoir system, creating a system operation set (Section 13.7), defining system zone values (Section 13.8), and specifying the storage balance (Section 13.9) for each reservoir in your system.

The initial steps in creating a reservoir system that uses explicit system storage balancing are:

- From the **ReservoirSystem** menu, select **New System**. The **New Reservoir System** dialog will open (Figure 13.14).

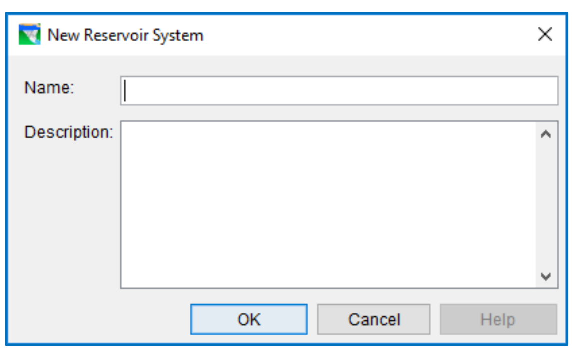


Figure 13.14 New Reservoir System

- Enter a **Name** and **Description** for the new reservoir system.
- Click **OK**. The **New Reservoir System** dialog will close.

The name and description of the new **Reservoir System** will now appear in the **Reservoir System Editor**, with all other fields remaining blank (as shown in Figure 13.15).

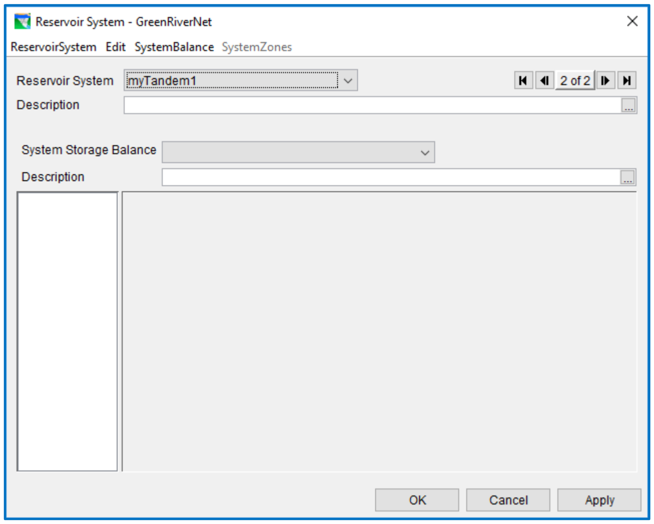


Figure 13.15 Reservoir System Editor—New Reservoir System

13.6 Selecting Reservoirs for the System

Once you have named your reservoir system, you will need to select all of the reservoirs to be included in the system storage balancing. *Only those reservoirs that have not been included in another reservoir system are available to be selected.*

To select the reservoirs to be included in the system storage balancing:

- Choose **Edit Reservoir Set** from the **Edit** menu of the Reservoir System Editor. The **Reservoir Selection Editor** will open (Figure 13.16). The available reservoirs in your network (that have not been included in another reservoir system) appear in the **Available** pane on the left side of the **Reservoir Selection Editor**. To add a reservoir to your new reservoir system, click on the reservoir's name and click **Add**. To select all of the available reservoirs, click **Add All**.
- The reservoirs you select will move from the **Available** pane to the **Selected** pane on the right. To remove a reservoir from the selected list, click on its name and click **Remove**. To remove all reservoirs from the **Selected** list, click **Remove All**.

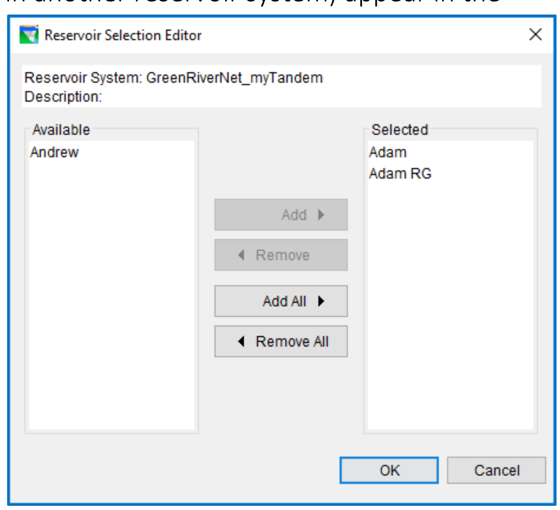


Figure 13.16 Reservoir Selection Editor

- Click **OK** to approve your choices and close the **Reservoir Selection Editor**.

Although you have selected reservoirs for your reservoir system, they will not yet appear in the **Reservoir System Editor**. You must first specify the system storage balance and define reservoir system storage zones.

13.7 Defining a System Storage Balance

To define the **System Storage Balance** for the reservoirs in your reservoir system:

- Select **New** from the **SystemBalance** menu of the **Reservoir System Editor**. The **New System Balance** dialog will open (Figure 13.17).

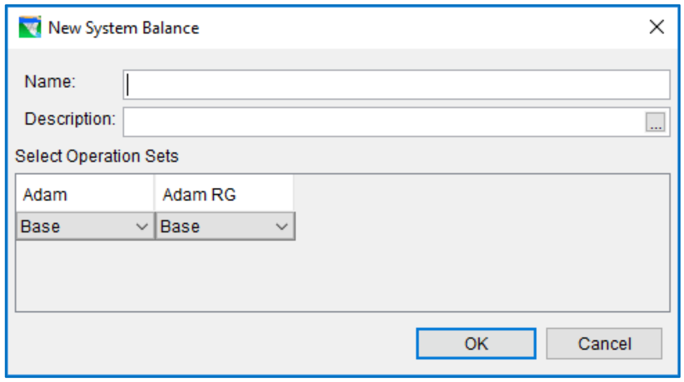


Figure 13.17 New System Storage Balance for Reservoir System

- Enter a Name and Description for the System Storage Balance.
- Below the **Description** area, you will see the names of the reservoirs you have selected for your system. Below the name of each reservoir is a dropdown list of the **Operation Sets** available for that reservoir. Click on the arrow to access the list for each reservoir and then select an operation set that *contains the system operation rule(s)*.
- Click **OK** to close the New System Balance dialog.
- You must now define at least one **Reservoir System Zone**.

13.8 Defining Reservoir System Zones

To define the Reservoir System Zone(s):

- Select **New** from the **Zones** menu of the **Reservoir System Editor**. The **New Storage Zone** dialog will open (Figure 13.18)
- Enter a **Name** and **Description** for the new storage zone.
- Click **OK** to close the **New Storage Zone** dialog.

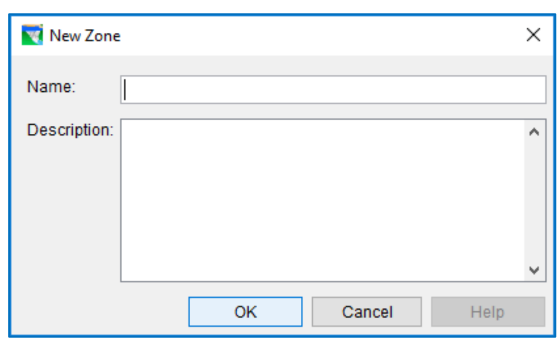


Figure 13.18 New Storage Zone

When you have defined at least one system storage zone for your reservoir system, its name will appear in the white pane at the bottom left of the **Reservoir System Editor**, and a new set of fields will appear in the large gray area, as shown in Figure 13.19. These new fields allow you to configure the system storage balance for your reservoir system.

13.9 Configuring System Storage Balance

Once you have successfully created a new reservoir system (by following the steps presented in Sections 13.5 through 13.8), several new fields become available that allow you to configure the storage balance across reservoirs. We will use Figure 13.19 to illustrate how to do this.

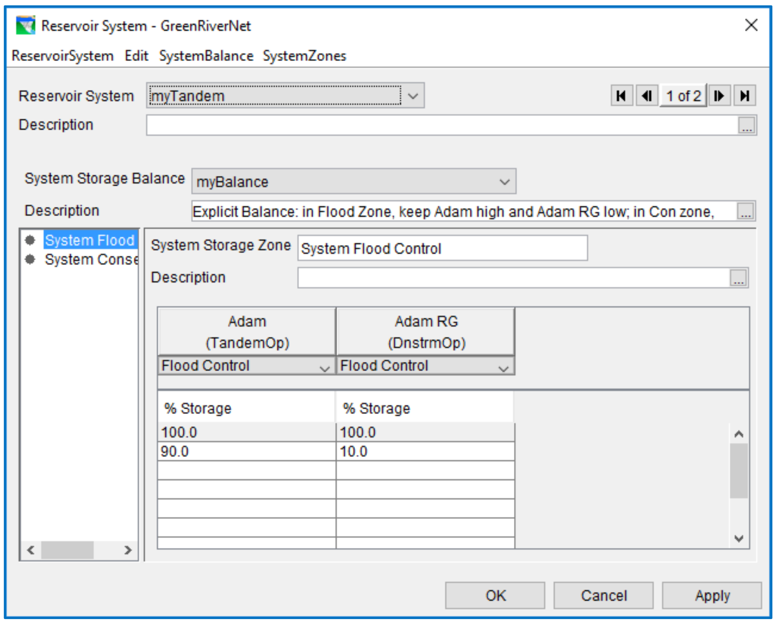
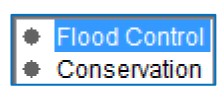
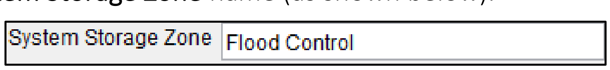


Figure 13.19 Configuring System Storage Balance

- Choose a **System Storage Balance** option from the drop-down list. Its description will appear in the editable **Description** field, and the available **System Storage Zones** will appear in the white pane on the left side of the editor window.



- In Figure 13.19, the **System Storage Balance** field indicates we are using a System Storage Balance called "Parallel SysOp_MyTown," and there are two **System Storage Zones**, "Flood Control" and "Conservation."
- Click on a **System Storage Zone** in the white pane on the left side of the editor window to select it. Its name and description will appear to the right in the editable **System Storage Zone** name (as shown below).



In Figure 13.19, "Flood Control" is selected, so its name and description appear in the **System Storage Zone** and **Description** fields. For the selected zone ("Flood Control" in this example), in the list beneath each reservoir name, select a reservoir zone for each reservoir in the system. In Figure 13.19, there are two reservoirs in the system, and we have chosen the "Flood Control" zone for each reservoir.

- In the table below the reservoir operation sets, enter percentages of system storage zones for each reservoir, as shown in Figure 13.20. These storage percentages represent *inflection points* along the balance line curve, which delineates how the reservoirs will balance when system operations are performed.

% Storage	% Storage
100.0	100.0
90.0	10.0

Figure 13.20 Percent Storage for each Reservoir in a Two-Reservoir System

- Click **Apply** to save your changes.
- Repeat the process for each **System Storage Zone** you have created. For example, in Figure 13.19, once you have configured the "Flood Control" System Storage Zone, you would want to configure the reservoir system balance for the "Conservation" System Storage Zone.
- If you wish to configure additional storage balance options, click **Apply** to save your changes for the current storage balance option, then select another storage balance option from the **System Storage Balance** list.
- When you have finished configuring your system storage balance, click **OK** to close the **Reservoir System** Editor.

13.10 General System Operation Notes

As previously discussed, you can allow ResSim to use its implicit storage balance, or you can create an explicit system storage balance. In addition, the following notes related to system operations are provided:

- When not identified in an explicit storage balance definition, maximum storage (100% full) is considered to be the maximum storage value in the elevation-storage table defining the pool of each reservoir.
- Minimum storage (0% full) is considered to be the top of the Inactive zone unless the user has deleted the Inactive zone from the operation set, in which case, minimum storage of zero is used.
- The Guide Curve does not have to be the top of the Conservation zone. The implicit balance scheme will recognize the specified guide curve at each reservoir when delineating the default balance lines.
- For parallel reservoirs to operate as a system for a common downstream control point, the same *common* downstream control rule must exist at all of the parallel system reservoirs. The rule must be created at only one of the reservoirs. Once created, that rule will automatically be available in the list of existing rules for the other reservoirs. Then, to establish system operation, the downstream control rule

can be added to the operation set at the other reservoirs via the "Use Existing" option in the Operations tab of the Reservoir Editor (Section 11.1.2).

If the downstream control rules are created separately at each reservoir, even if the data entered into the rules is identical, system operation will not be invoked.

- For tandem reservoirs, there can be an intermediate control point(s) for which the upstream reservoir operates.
- System operation rules can be prioritized along with other rules in the operation set (i.e., depending on prioritization, they may be overridden).
- Implicit and explicit storage balance schemes can be established among two or more reservoirs. When defining an explicit system balance, all reservoirs identified for parallel or tandem operation must be included in a single reservoir system (refer to Section 13.6). For example, if you have three parallel reservoirs operating for a common downstream location, and only two of them are included in an explicit system storage balance, the explicit system storage balance will be ignored and the default implicit storage balance scheme will be used instead.

When one or more reservoirs are involved in *both* tandem and parallel system operation and you want to define an explicit storage balance between the parallel reservoirs, you should put *all* the participating reservoirs (both parallel and tandem) into a single reservoir system and the balance definition should address all the reservoirs in the system. For example, if you have two reservoirs operating in parallel for a common downstream location and one of the two reservoirs is part of a tandem system with an upstream reservoir, it is best to include all three reservoirs in the explicit reservoir system and associated system balance. Otherwise, if only the parallel reservoirs are included in the explicit system, and the tandem reservoirs are left to operate using the implicit system balance, then undesirable operations will result due to incompatibility between the implicit and explicit balance schemes.

- The explicit system storage balance is defined using the Reservoir System Editor, as presented in the preceding sections of this chapter. To simulate the explicit system operation, the explicit system storage balance must be selected in the Alternative Editor's operations tab (Sections 14.6) along with the reservoir operation sets that contain the system operation rules.

Chapter 14
Defining Alternatives

Each alternative represents the combination of a reservoir network, a selection of one operation set per reservoir, the specification of the initial conditions of each reservoir, and the selection of inflows and other input time-series datasets needed by your model. An Alternative consists of a Reservoir Network (previously created from a Configuration), Run Control specifications, an Operation Set for each Reservoir in the network, a Storage Balance Operation Set for each Reservoir System in the network (if applicable), a definition of initial (Lookback) conditions, and a mapping of all Time-Series records to identified local inflows. To develop an Alternative, you use the **Alternative Editor** to name the Alternative and give it a description, define the Time Step and Flow Computation method, select Reservoir Operation Sets, select System Operation Sets (if applicable), select a Lookback Type, associate time-series data with locations, associate observed data with locations, and save the Alternative you have created. This chapter will guide you through these steps.

14.1 Preparing to Develop Alternatives

Before you can develop an alternative, you need to define the operational reservoir data using the Reservoir Editor. Chapter 11 describes this procedure. If your network contains reservoir systems, you need to define the system storage balance (as described in Section 13.7).

14.2 Accessing the Alternative Editor

To access the **Alternative Editor** (Figure 14.1), choose **Edit** from the **Alternative** menu in the Reservoir Network Module.

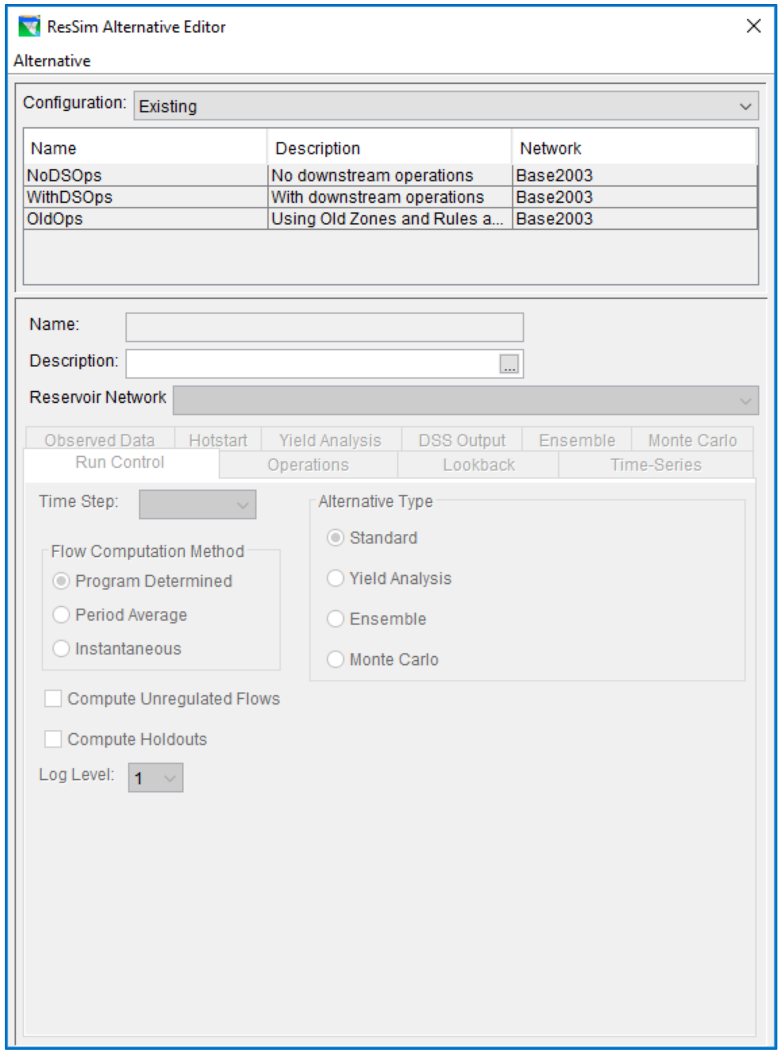


Figure 14.1 Alternative Editor

14.3 Creating a New Alternative

The first step in creating an Alternative is to give it a Name and Description, and then select the reservoir Network you want to use.

To create a New Alternative:

- From the **Alternative** menu of the Alternative Editor, select **New**. The **New Alternative** dialog will open (Figure 14.2).
- Enter a **Name** and **Description**. Use the ... button to open a larger editing window for the description.

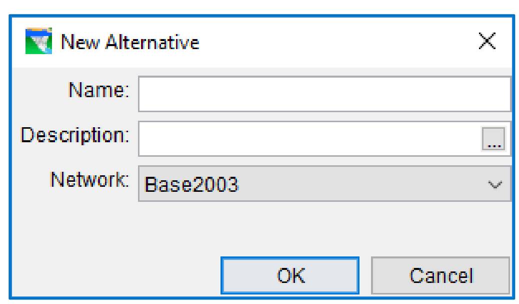


Figure 14.2 New Alternative

- Select a **Network** by choosing from the list of available networks.
- Click **OK** to close the New Alternative dialog. The name and description you entered will now appear in the **Name** and **Description** fields of the Alternative Editor (Figure 14.3).

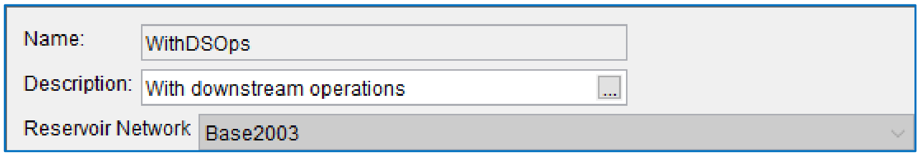


Figure 14.3 Alternative Editor—Name and Description Fields

14.4 Selecting a Time Step and Flow Computation Method

Once you have given your alternative a name and description and chosen the reservoir network, you will need to select the timestep and flow computation method for the alternative.

- From the **Run Control** tab, select the timestep from the drop-down menu (Figure 14.4):

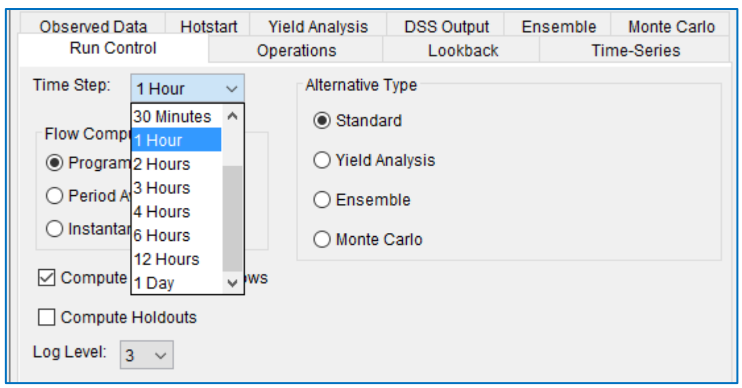


Figure 14.4 Alternative Editor—Run Control Tab—Time Step

- Select the Flow Computation Method (Program Determined, Period Average, or Instantaneous) (Figure 14.5).

  Flow Computation Method: choose one of the following methods to indicate the appropriate technique to be used in the computations:

  - **Program Determined**—determined by ResSim from reviewing time intervals of the input time series data.
  - **Period Average**—typically used when the time interval of the input time series data is daily or longer.
  - **Instantaneous**—typically used for short-interval data (e.g., hourly) that is less than a daily time interval.

- Select Compute Unregulated Flows, Compute Holdouts, and Choose the Log Level (Figure 14.5).
  - Check **Compute Unregulated Flows** if you want ResSim to perform additional calculations to determine the unregulated conditions in the watershed (i.e., without the regulation of reservoir and diversion projects). This process occurs after the computations have been completed; therefore, if the unregulated flows are not of interest, you can un-check this option to save compute time.
  - Check **Compute Holdouts** when you want ResSim to calculate the amount of water the reservoir stores (as opposed to just releasing inflow). Holdouts indicate the effects of reservoir regulation and are most applicable when you will be performing a subsequent Flood Impact or Flood Damage analysis.
  - Select the **Log Level** (1-10) to control the amount of detailed messages that will be displayed during the computations. A log level of 10 will provide the highest level of detailed messages.

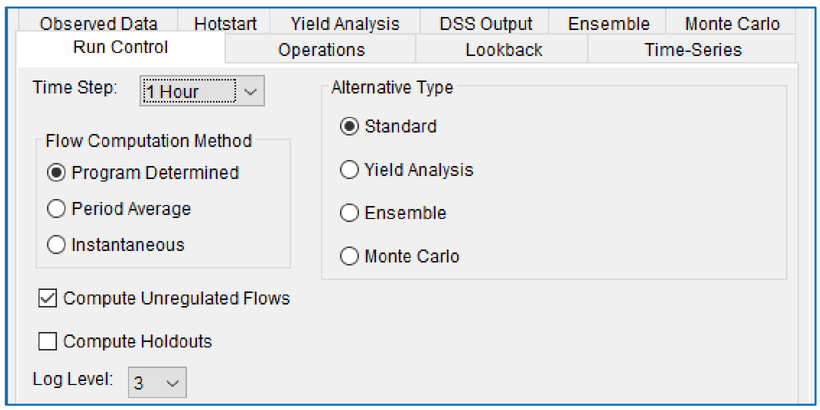


Figure 14.5 Alternative Editor—Run Control Tab—Flow Computation Method

- Select the Alternative Type (Figure 14.5).

  Alternative Type: The default alternative type is **Standard**, and almost all models use this default, but occasionally there is a need for advanced techniques offered with other alternative types. You can read about these other alternative types in the Appendices. Select one of the following:

  - **Standard**—default alternative type for doing classic ResSim simulations.
  - **Yield Analysis**—used to run iterative simulations that can solve for reservoir or storage account yield (Appendix F).
  - **Ensemble**—designed for use with ensemble streamflow forecasts, this alternative type allows you to run multiple correlated inflow sets for one alternative and produce multiple correlated output datasets. (Appendix G).
  - **Monte Carlo**—designed to incorporate risk and uncertainty into results. A simulation is run numerous times with specified input variables sampled from given distributions (Appendix H).

14.5 Selecting a Reservoir Operation Set

Now that you have selected the alternative's timestep and flow computation method, you will need to select the **Operation Set** you want each reservoir to follow for the alternative. You will have created the operation set(s) using the Reservoir Editor (as described in Chapter 10).

To select a reservoir operation set for an alternative:

- In the Alternative Editor, select the Operations tab (Figure 14.6).

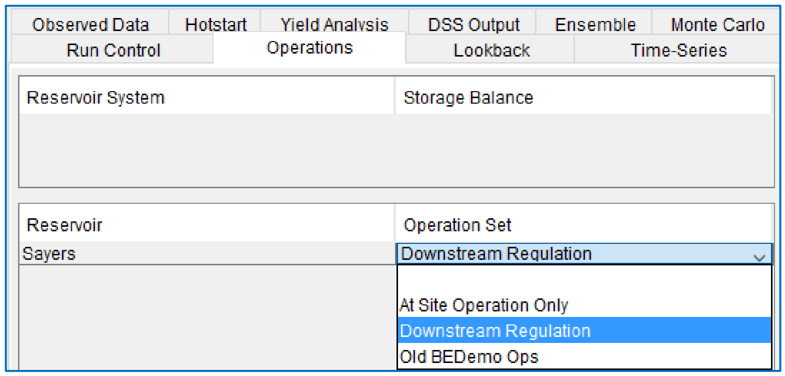


Figure 14.6 Alternative Editor—Operations Tab—Reservoir Operation Set

- Select an Operation Set (for each reservoir in the network) by choosing from the **Operation Set** dropdown list.

14.6 Selecting a Reservoir System Storage Balance

When you create a reservoir system and define an explicit storage balance scheme for a system operation (described in Chapter 12), an entry will appear in the Alternative Editor's Operations tab that identifies the Reservoir System along with a field in which you are required to select a system storage balance operation for the reservoirs to follow. Note that the individual operation sets for each reservoir containing the Downstream Control and/or Tandem Operation rules have to also be selected in the alternative in order to simulate system operation.

To select a system operation scheme for an alternative:

- In the **Alternative Editor**, select the **Operations** tab (Figure 14.7).
- In addition to selecting the operation set for each reservoir (as previously described in Section 14.5), select a **Storage Balance** for each **Reservoir System** you have configured by choosing from the **Storage Balance** dropdown list. The selection options will be either the operation that contains the explicit storage balance scheme you want to apply, or "NONE" if you wish to have the alternative use the implicit (default) storage balance scheme instead of the explicit storage balance you previously set up.

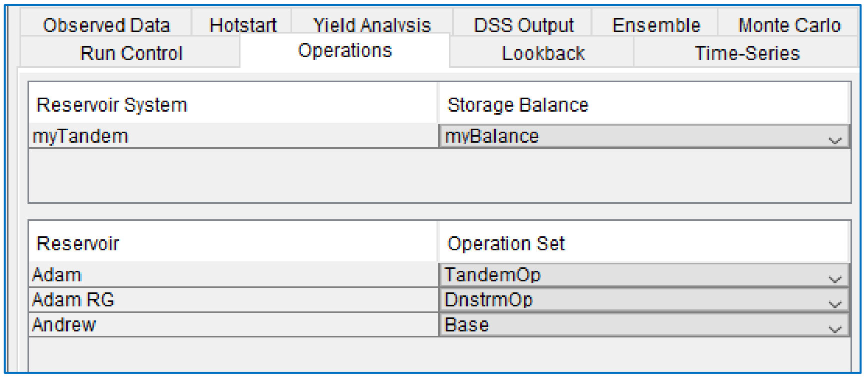


Figure 14.7 Alternative Editor—Operations Tab—Reservoir System Storage Balance

14.7 Selecting a Water Account Set

If you are using the ResSim water accounting feature in your alternative, you will need to specify the account set to use in the Operations tab of the Alternative Editor. See Appendix F for information on ResSim water accounting.

To select a Water Account Set for an alternative:

- In the **Alternative Editor**, select the **Operations** tab (Figure 14.8).

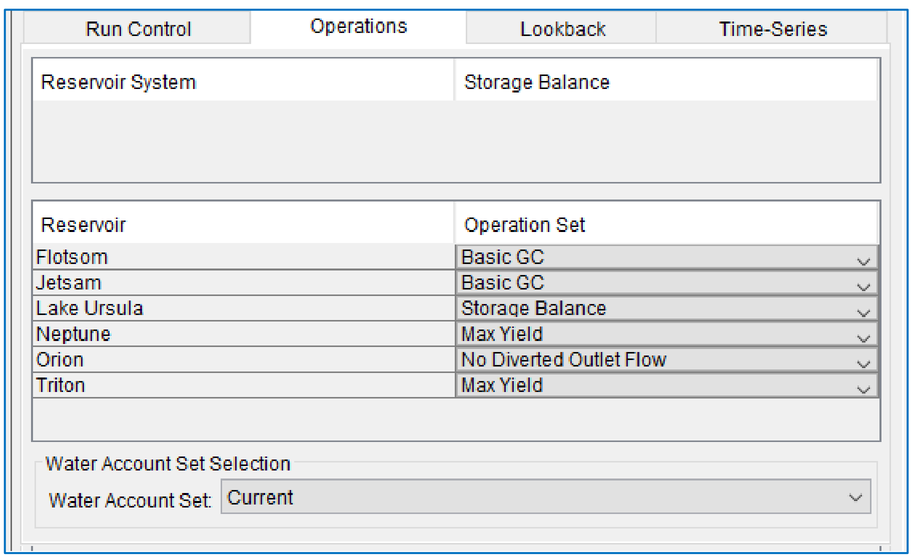


Figure 14.8 Alternative Editor—Operations Tab—Water Account Set Selection

- At the very bottom of the Operations tab is the **Water Account Set Selection** pane. Select the correct **Water Account Set** using the dropdown menu. If NONE is selected, water accounting will not be performed for the alternative.

14.8 Selecting Lookback Type

After specifying the operation set for each of your reservoirs, you will need to specify the initial (or starting) conditions for the alternative. This is referred to in ResSim as the

Lookback (or warmup) period. You will need to specify whether Constant or "mapped" Time-Series data will define each element and parameter during the Lookback period.

To set the Lookback **Type** for locations in your reservoir network:

- In the Alternative Editor, select the Lookback tab.
- For each location or parameter, in the **Type** column double-click on the appropriate arrows to select either **Constant** or **Time-Series** from the dropdown list (as shown in Figure 14.9).

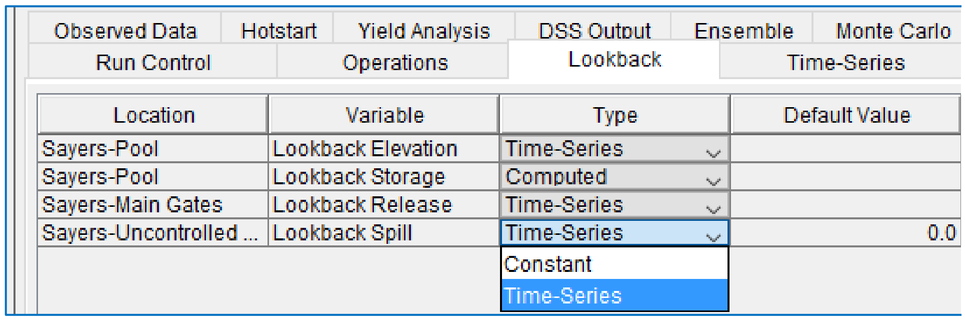


Figure 14.9 Alternative Editor—Lookback Tab

- For locations or parameters where you have selected **Constant** as the lookback type, enter the value for the constant in the **Default Value** field.

14.9 Associating Time-Series Data with a Location

Next, you will need to associate (by "mapping") a time-series record to each location or parameter for which Time-Series Data are needed. You can also use inflow multipliers to increase or decrease the inflow values.

To associate Time-Series Data with a Location:

- In the **Alternative Editor**, select the **Time-Series** tab (Figure 14.10).

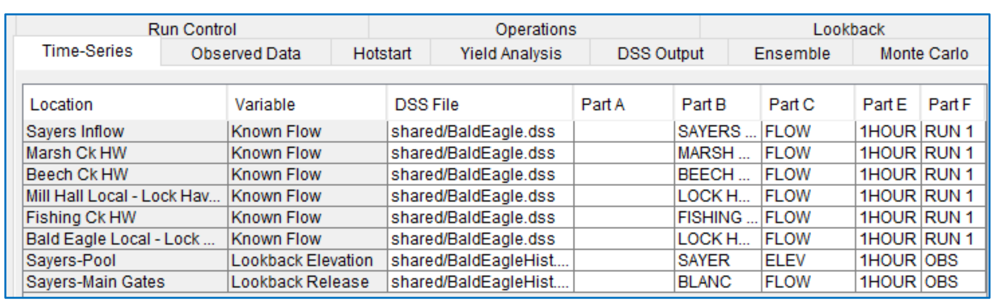


Figure 14.10 Alternative Editor—Time-Series Tab

- Each row in the Time-Series table is a local flow location (specified in the Local Flow tab within the Junction editor in the Reservoir Network module), an element/parameter you have defined on the Lookback tab as Time Series, or a Time Series that is referenced in the Operation rules. Select each row, one at a time, and click **Select DSS Path** to access the **Select Pathname** dialog (Figure 14.11).

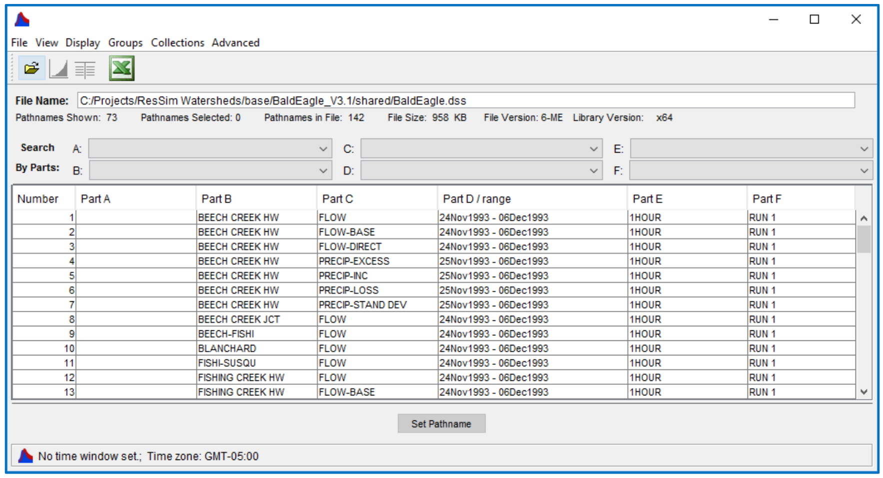


Figure 14.11 Select Pathname

The **Select Pathname** dialog allows you to select records from a list of pathnames (or *catalog*) in the database. From the **View** menu, you can choose to display a straight list of pathnames or a list of pathnames separated into parts, and you can refine the list by searching for either a string in the pathnames or for specific pathname parts. For detailed information about working with DSS files, refer to the discussion of **HEC-DSSVue** in Appendix D.

- To select time-series records for a location or element/parameter in your alternative, open the DSS file you wish to browse. If you know the name of the file, you can type the file name (including the path) directly into the **File Name** box to open the DSS file. Otherwise, choose **Open** from the **File** menu or click the folder icon to select the DSS database file you want.
- Once you have selected a file, the **Select Pathname** dialog displays the filename, the number of pathnames shown in the list, the number of pathnames selected, the total number of pathnames in the database file, and the size of the database file. The individual pathnames display in a table beneath the search area.
- Use the **Search** feature to "filter" and locate individual records in the DSS file or scroll through the list of pathnames with the vertical scrollbar.
- Click on a pathname in the list (to select it).
- If you would like to view the DSS data for the selected pathname, select **Plot** or **Tabulate** from the **Display** menu, or you can click on **Plot** or **Tabulate** .
- When you are satisfied with the DSS record you have chosen for a location or element/parameter, click on **Set Pathname** and the selected pathname will appear in the Time-Series tab of the Alternative editor.
- Plot and Tabulate are available at the bottom of the Alternative Editor for use in reviewing the time series data associated with a location.
- You can increase or decrease the magnitude of the flow data values by clicking on **Inflow Multipliers**. Inflow Multipliers...

- If you are using the flow multipliers, a check will appear in the checkbox to indicate inflow multipliers are being used.
- The **Inflow Multipliers** editor (Figure 14.12) appears when you click on Inflow Multipliers.

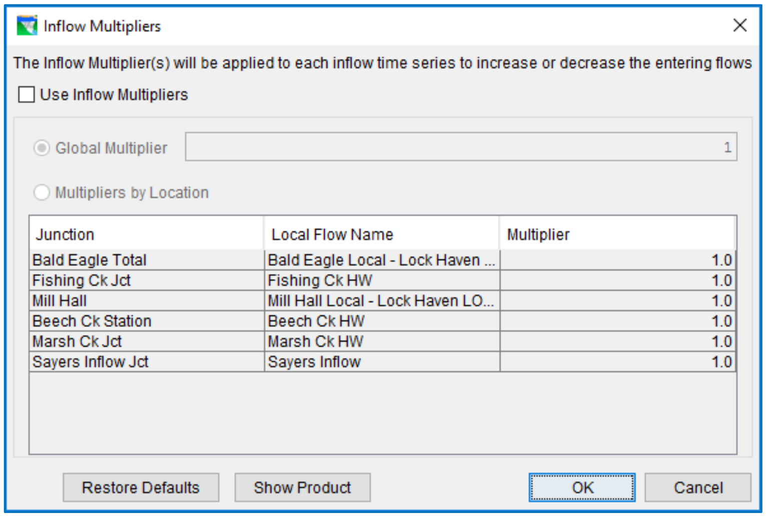


Figure 14.12 Inflow Multiplier Editor

Click in the **Use Inflow Multipliers** checkbox to activate the editor (as shown in Figure 14.13).

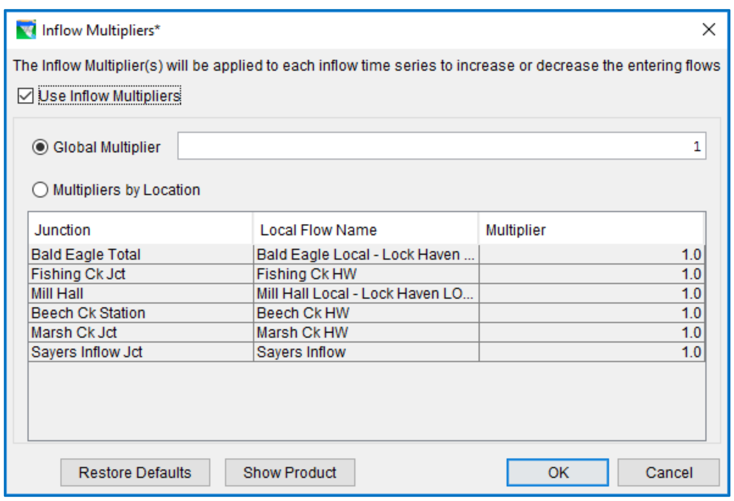


Figure 14.13 "Activated" Inflow Multiplier Editor

The following options are available for entering the Inflow Multipliers:

- The **Global Multiplier** option is used when you want the same multiplier to be applied to all locations, as shown in Figure 14.14.

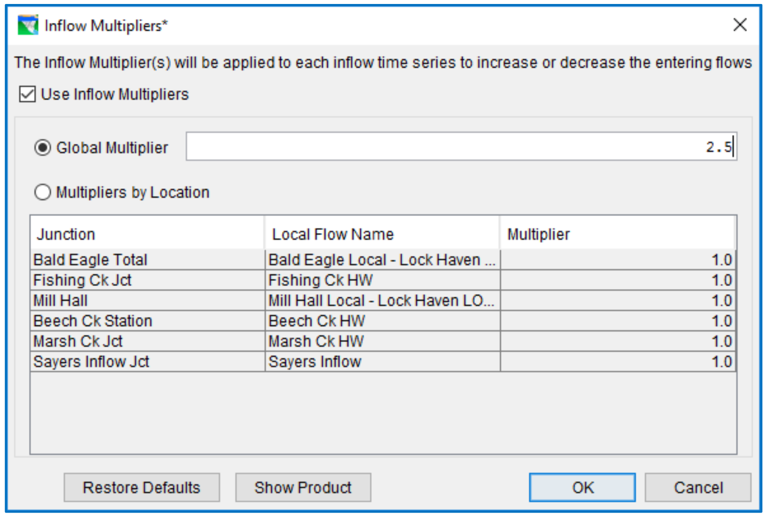


Figure 14.14 Inflow Multiplier Editor—Global Multiplier

- The **Multipliers by Location** option is used when you want a different multiplier to apply to each location, as shown in Figure 14.15.

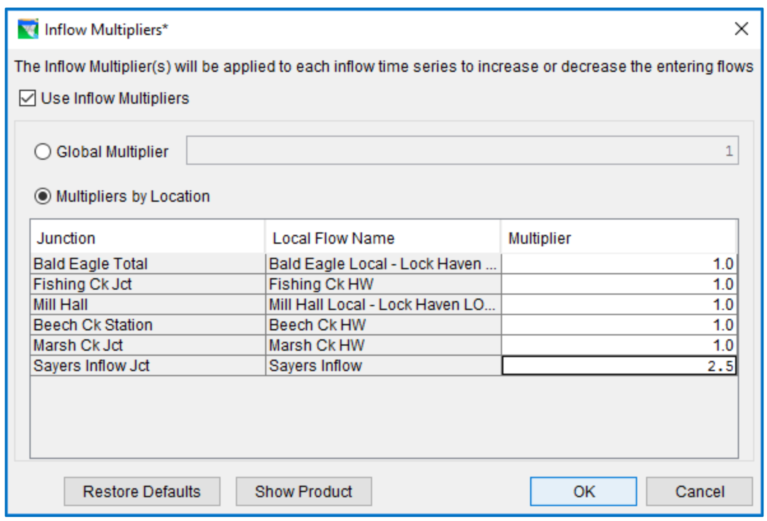


Figure 14.15 Inflow Multiplier Editor—Multipliers by Location

- If you uncheck the **Use Inflow Multipliers** checkbox, then you de-activate the multipliers, but the values remain in the editor for future use, if desired.

14.10 Defining Observed Data

While defining your reservoir network, if you have specified (by checking the appropriate box or boxes in the Observed tab within each element's editor in the Network module) that observed data is available for individual reservoirs, junctions, reaches, diversions, and diverted outlets, then the **Observed Data** tab of the Alternative Editor will list these locations in a table along with the DSS path information associated with them, as shown in Figure 14.16.

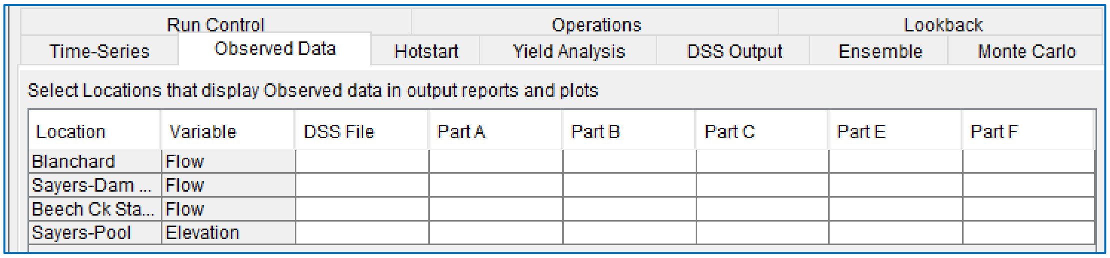


Figure 14.16 Alternative Editor—Observed Data Tab

14.11 Hotstart Options

Typically, the Lookback period is used for *warming-up* the state of the watershed. 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 Hotstart options. In other words, you can use the Hotstart as a way of assigning a previously simulated watershed state to another simulation.

To use the Hotstart option, you must first run a simulation that saves system state data to "Hotstart" files. You may create one or multiple Hotstart files from a single simulation, differentiated by the dates saved in the hotstart files. Each of those files can be used to inform an alternative of starting state data, but only one Hotstart file can be *used* by an alternative. The Hotstart file used is selected by setting the using alternative's simulation start date to correspond to the Hotstart date. The alternative then uses a Hotstart file, rather than Lookback data to determine initial conditions.

To define the Hotstart Options:

- In the **Alternative Editor**, select the **Hotstart** tab (Figure 14.17). The Hotstart editor is divided into two panels:
- Create Hotstart File
- Read Hotstart File

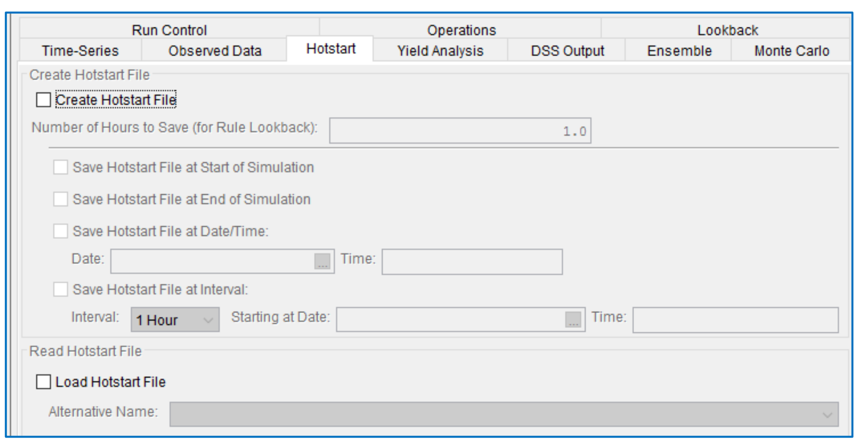


Figure 14.17 Alternative Editor—Hotstart Tab

- ☑ Create Hotstart File Check the box labeled "Create Hotstart File".

- Number of Hours to Save (for Rule Lookback) Enter an appropriate value for the "Number of Hours to Save (for Rule Lookback)". Because some rules or other operations may rely on data from previous timesteps, the Hotstart feature will save the appropriate number of timestep values (based on the number of hours you enter into this text field) for all its computed time series. This puts the responsibility on you, the modeler, to determine how many hours should be saved.
- Check one or more of the Hotstart options for saving the Hotstart files. You can choose to write the Hotstart files at the "Start of Simulation", at the "End of Simulation", at a specific "Date/Time", or you can choose the "Save Hotstart File at Interval" option and then select the Interval from the available list and specify the Date and Time to start writing the *recurring interval* Hotstart files. Caution should be used when selecting the Interval option since many Hotstart files can be written, depending on the interval you select and the length of the simulation used when saving the Hotstart files.

The "Save Hotstart File at Interval" option can write many, many Hotstart files. Therefore, use caution when selecting the recurring Interval with Starting Date and Time option.

Figure 14.18 shows an example portion of the Alternative editor where Create Hotstart File is being invoked. Hotstart options for an alternative named "HotStart" are being used to save Hotstart files at a specific **Date/Time** (27Nov1993 at 1400 hours). Since this example indicates that the number of hours to include in the Hotstart files (prior to the saved Hotstart Date/Time) is 1 hour, the Hotstart files will contain system state data from 27Nov1993 at 1300 hours through 20Oct1999 at 1400 hours.

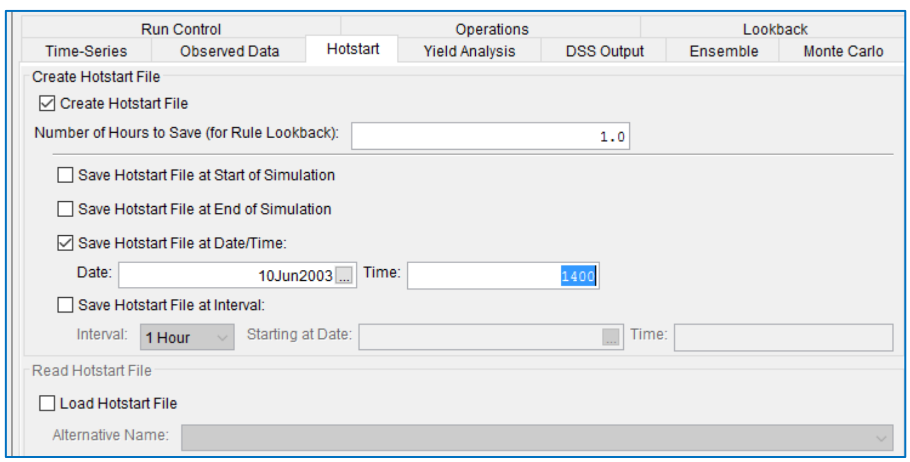
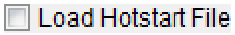


Figure 14.18 Alternative Editor—Create Hotstart File

After simulating an alternative (e.g., "HotStart") that creates Hotstart files, a **hotstarts** folder is created in the rss folder of the watershed and contains two types of files corresponding to the selected Hotstart options: These files have extensions of "**dss**" and "**nhs**" and are named based on the selected Hotstart option(s). Figure 14.19 shows the watershed folder's file system after a folder named **hotstarts** was created that contains two files whose names are based on the options shown in Figure 14.18 (for this example, the Hotstart files reflect a Date and Time of 27Nov1997 at 1400 hours).

- 

For the alternative that *uses* (or *loads*) Hotstart files, check the box labeled "Load Hotstart File" (to use the system state from a previously simulated alternative). The simulation starting date of an alternative that *uses* a Hotstart file must correspond to the save date for one of the Hotstart files saved in the alternative that *created* the Hotstart files. For the above example, a Hotstart file has been created for 27Nov1993 at 1400 hours, so this must be the starting date and time for any simulation using the alternative that Loads that Hotstart file.

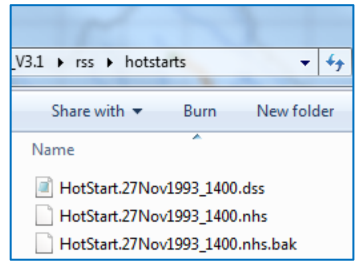


Figure 14.19 Hotstart Files

- Once you have checked the Load Hotstart File checkbox, then you need to select the **Alternative Name** to indicate which alternative was used in creating the Hotstart files (Figure 14.20).

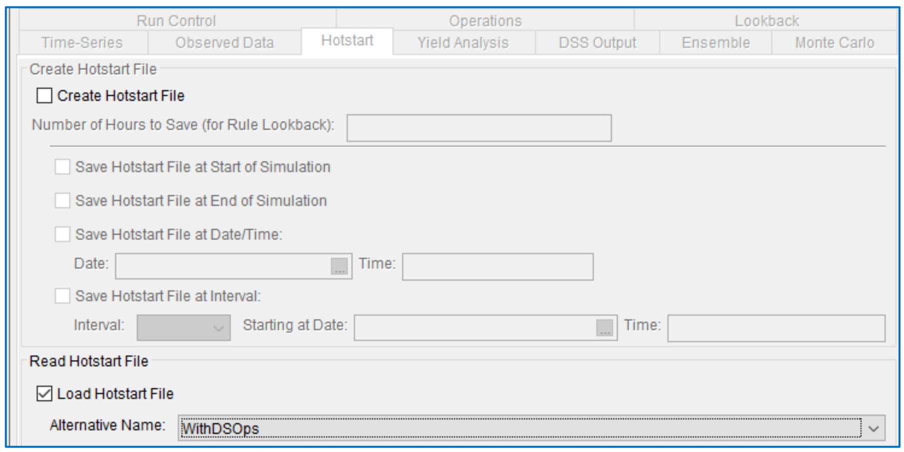


Figure 14.20 Alternative Editor—Load Hotstart File

14.12 DSS Output

The **DSS Output** tab of the **Alternative Editor** allows the user to optimize write time to the simulation.dss output file by specifying which elements and variables should be written to the file during a compute. The **DSS Output** tab contains a tab for each DSS output element, i.e. **Reservoirs**, **Reaches**, **Diversions**, **Junctions**, and **State Variables** (Figure 14.21).

Initially, the **Output All DSS Records** check box is selected and all element tabs are disabled. All DSS output will be written to the simulation.dss file. If the **Output All DSS Records** check box is unselected, the element tabs will become available and the user can check DSS data to not write.

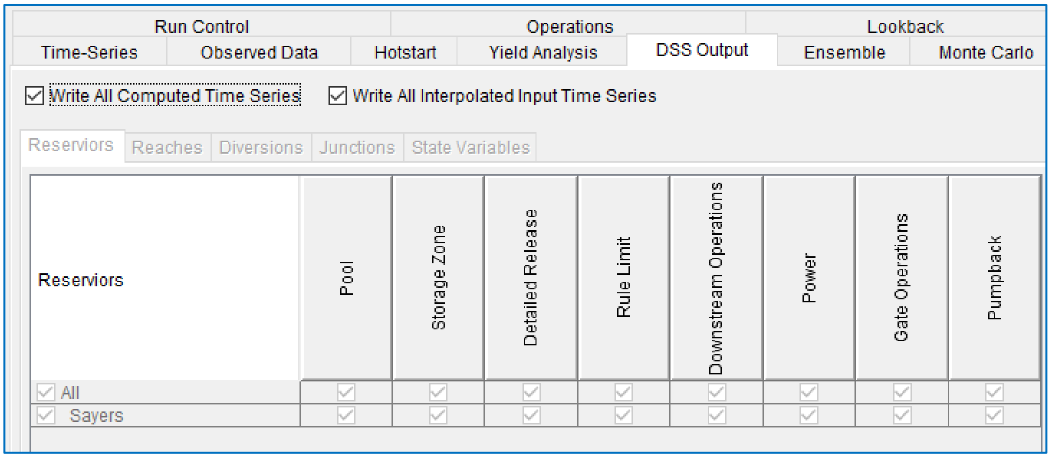


Figure 14.21 Alternative Editor—DSS Output Tab Default

To select DSS data to write to the simulation.dss file at the end of a ResSim Alternative compute, the user would check the row (specific element) and column (variable) for a particular output variable. The top row of checkboxes represents all elements and variables in the table. The user can select or unselect the entire column using the all checkboxes. Individual elements and their single variables can be selected/unselected by checking/uncheck the checkbox in the (specific element) and column (variable) for a particular element. The top checkbox displays a third state, which indicates various rows are selected for that column.

14.13 Saving an Alternative

When you have finished defining an alternative, save it by selecting **Save** from the **Alternative** menu of the **Alternative Editor**, as shown in Figure 14.22. You may then close the Alternative Editor. It is a good idea to save your **Network** and **Watershed** after creating a new Alternative. To do this, select **Save** from the **Network** menu (Figure 14.22), and then select **Save Watershed** from the **File** menu (Figure 14.23).

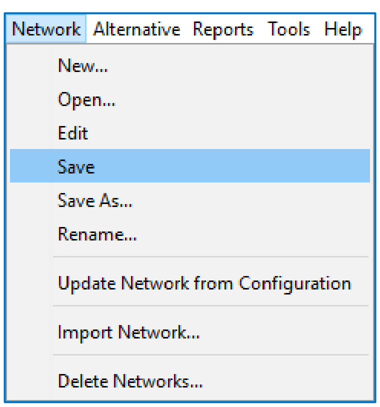


Figure 14.22 Reservoir Network Module—Network Menu—Save Network

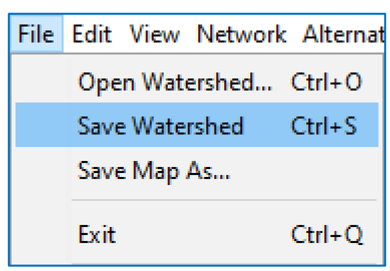


Figure 14.23 File Menu—Save Watershed

Chapter 15 Running Simulations and Analyzing Results

When you have entered all required data and have created Alternatives, you are ready to perform a Simulation. A Simulation is where you specify the time window and time interval parameters for either a single Alternative or a group of Alternatives. After a successful Simulation, you can analyze the results, make revisions, and perform additional Simulations to better evaluate the reservoir operations in your watershed.

When you create a simulation, you must specify a simulation time window and select the alternatives to be analyzed. ResSim then creates a folder with the name of the simulation in the rss folder of the watershed; this folder represents the *simulation*. Within this simulation folder will be a partial copy of the watershed, including only those files needed by the selected alternatives. Also created in the simulation folder is a DSS file named simulation.dss, which will contain all the DSS records that represent the input and output for the selected alternatives.

15.1 Recognizing Simulation Screen Components

The Simulation Module (Figure 15.1) provides the tools you will need to create and run Simulations. A discussion of the components and features of the Simulation Module follows and begins on the next page so that the descriptions can be viewed with the figure.

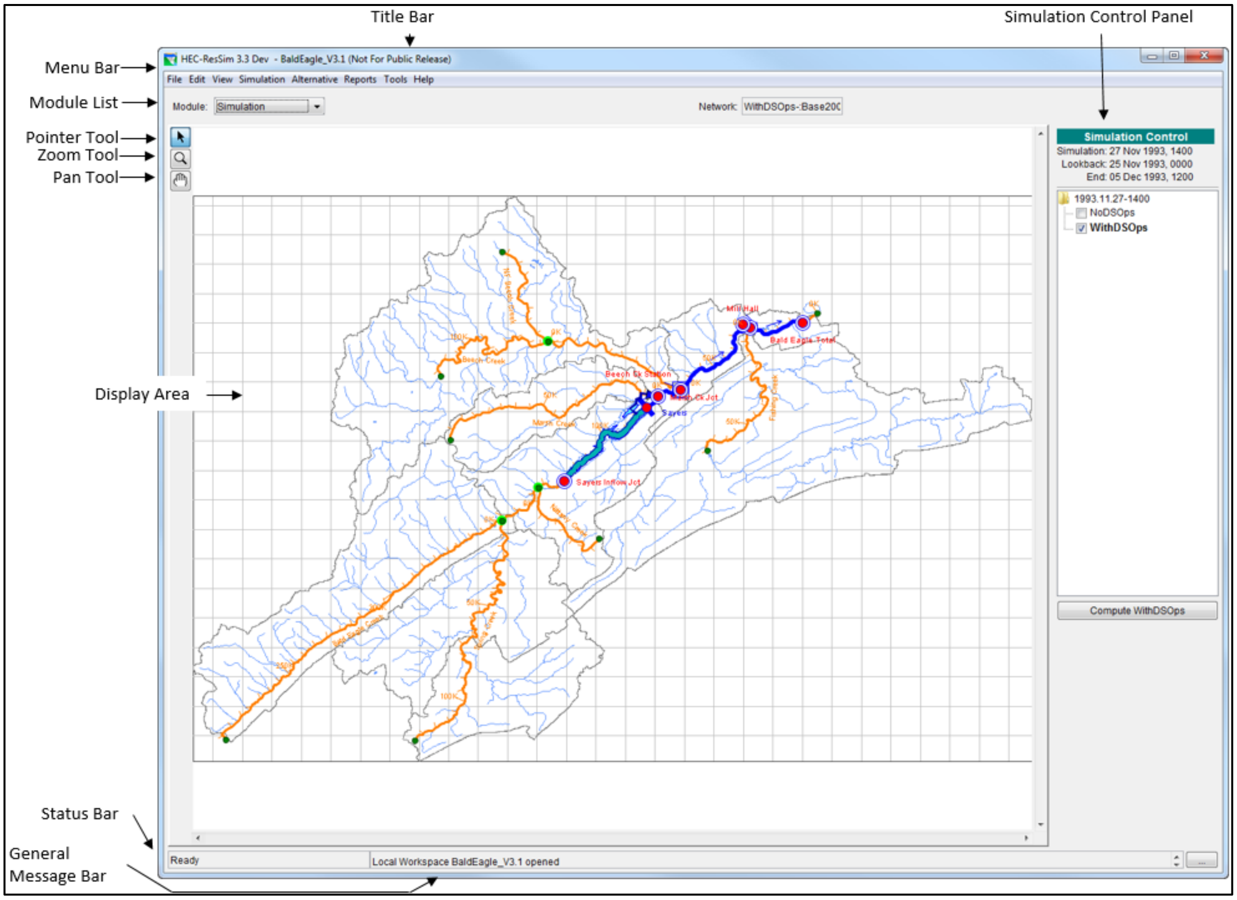


Figure 15.1 Simulation Module—Main Window

The **Title Bar** displays the name of opened watershed (displayed to the right of the **HEC-ResSim** name).

The **Simulation Control Panel** shows the time window for the currently opened simulation and provides controls to manipulate the simulation and alternatives. For more detail, see Section 15.1.3.

The **Menu Bar** contains menus of commands that you can use in ResSim. The items on the menus change as you switch between the various Modules, offering Module-specific commands. You can select a menu bar item by clicking on the name of the menu (such as **File**), then pointing and clicking on the item you wish to select. The Menu Bar is described in more detail in the next section.

The **Module List** contains all the available Modules of ResSim. Use this list to move between the Watershed Setup, Reservoir Network, and Simulation Modules. By default, the Module List opens to the Module most recently used.

15.1.1 Menu Bar

The following is an overview of the **Simulation Module**'s Menu Bar tools, which allow you to create and edit Simulations. The tools specific to this module will be described in more detail in the context of particular tasks later in this chapter.

The **File** menu (Figure 15.2) allows you to **Open** an existing watershed, **Save** a watershed, **Save Map** (saves the display area) and **Exit** ResSim. Your most-recently-used watersheds are listed at the bottom of the File menu.

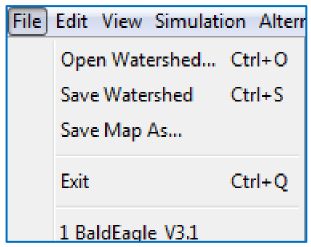


Figure 15.2 File Menu

The **Edit** menu (Figure 15.3) provides access to the **Script List**, allows you to **Set the Active Alternative,** and allows access to editors for **Reservoirs**, **Reaches**, **Junctions**, **Diversions**, **Reservoir Systems**, and **State Variables**. Additionally, you can select **Run...** to open the ResSim Editor for the active alternative. Note that any revisions you make to these elements applies only to the active Alternative. If you want the revisions to apply to subsequent Simulations, you must save the changes to your base directory (Section 15.8.1).

Figure 15.3 Edit Menu

In the **View** menu (Figure 15.4) select **Zoom to All** to restore your watershed map view to full size. **Layers...** opens the Layers Selector dialog. **Unit System** allows you to customize the display (view) settings for your watershed. If a dialog or editor window is open but inactive**, Restore Windows** brings the dialog or editor window to the front as the active window. The **Grid Lines** option allows for turning on or off the grid lines in the map region.

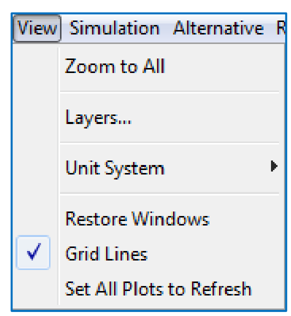


Figure 15.4 View Menu

The **Simulation** menu (Figure 15.5) is unique to the Simulation Module. It allows you to create a **New** Simulation, and **Open**, **Re-Open**, **Close**, **Simulation List…**, **Replace from Base…**, **Save**, or **Delete** an existing Simulation. The **Edit** command opens the Simulation Period dialog, while **Info** allows you to view the name, directory path, and user information for the current Simulation. The **Run Manager** provides the user with the capability of computing multiple alternatives without having to manually compute each one separately. You can also access **Overrides** for alternatives within the Simulation.

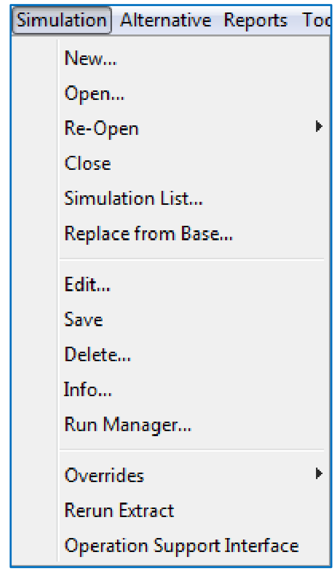


Figure 15.5 Simulation Menu

The **Alternative** menu (Figure 15.6), also available in the Reservoir Network Module, provides access to the Alternative Editor.

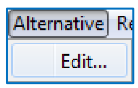


Figure 15.6 Alternative Menu

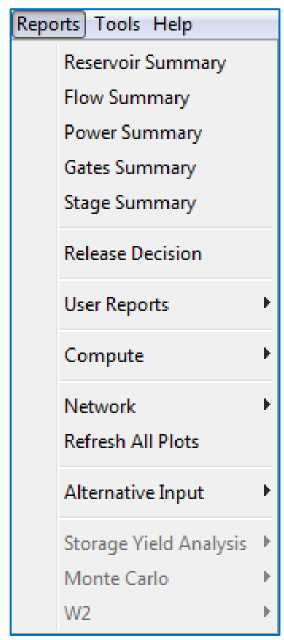


Figure 15.7 Reports Menu

The **Reports** menu (Figure 15.7) provides access to the following reports: **Reservoir Summary**, **Flow Summary**, **Power Summary**, **Gates Summary**, **Stage Summary**, **Release Decision**, and **User Reports**. You can also access **Compute** logs and **Network** reports (including the Reservoir List, Reach List, Junction List, Diversion List, and the "Advanced" Network Connectivity summary report). The **Refresh All Plots** option will cause all open plots to reflect any changes that have been made in the event of a simulation being recalculated after changes have been made to an alternative.

As in the other Modules, the **Tools** menu (Figure 15.8) provides access to **HEC-DSSVue**, **Scripts** and the **Script Editor**, the **Console Log** (which displays the information written to the ResSim.log file), **HEC-DSS Output** (a temporary log file that contains messages produced when DSS files are accessed), **Options**, and **Information** (which provides details about client, user, and watershed settings as well as server and system properties).

The **Options** Editor has tabs allowing access to the following items which are the same as those in the Network Module:

Model Directories (watershed locations for storing your watersheds); *Compute Display* settings (set the colors for compute messages and the format of log files); *Debug Levels* (set the level of debug messages); *General* (choose whether or not you want a confirmation message to appear when you exit ResSim and choose whether you want the last watershed reloaded at startup of the program); and, *Fonts* (for various window components). Specific to the Simulation Module, are two additional tabs: *Simulation* and *ResSim Compute*. These two options are discussed further in Section 15.4.2.

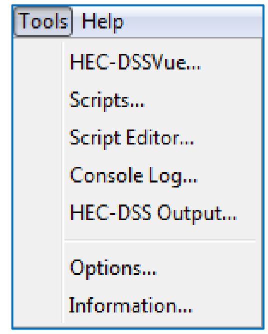


Figure 15.8 Tools Menu

In the **Help** menu (Figure 15.9), the **About HEC-ResSim** command displays information about the version of ResSim. Also, from this menu you can access user documentation and **Install Example Watersheds** (after you define a watershed location as described in Section 3.2.1)

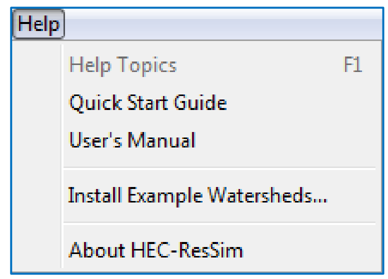


Figure 15.9 Help Menu

15.1.2 Map (Mouse) Tools

The **Map (Mouse) Tools**, which appear in a toolbar on the left side of the ResSim screen, allow you to interact with objects in the map display. The Simulation Module has three Map Tools and all are available in the other ResSim Modules:

Pointer Tool—in the Simulation Module, right-click model schematic elements in the map display with the Pointer Tool to access editors, default and user-defined plots, and release decision reports (for reservoirs).

Zoom Tool—the Zoom Tool allows you to zoom in and out of the display area in all Modules. To zoom in, hold the left mouse button down and outline the area you want to enlarge. To zoom out, click the right mouse button. Zooming out using the right-click button zooms out by a factor of two, positioning the clicked location at the center of the screen.

Pan Tool—after you have zoomed in with the Zoom Tool, you can use the Pan Tool to view watershed areas that fall outside of the ResSim window borders.

15.1.3 Simulation Control Panel

The **Simulation Control Panel** (Figure 15.10) displays details about the current Simulation and allows you to interact with Alternatives.

Displayed at the top of the panel are the **Simulation** time, **Lookback** time, and **End** time.

Below these details is the **Simulation Tree**, which displays the name of the current Simulation and its associated Alternatives. The currently **active Alternative** displays as **bold**, and a check mark in the box next to an Alternative indicates that it will be included when displaying results. Right-click on a Simulation or Alternative to access their context menus.

When you select an Alternative, **Compute** becomes available, allowing you to execute a run.

At the bottom of the Simulation Control Panel, the **Scripts** area displays buttons that launch user-created scripts.

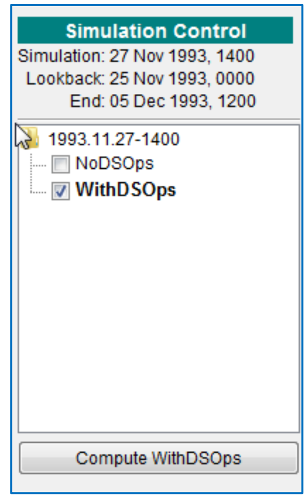


Figure 15.10 Simulation Menu

15.1.4 Display Area

The **Display Area** displays model schematic objects and map layers representing the **Active Alternative**.

Model elements of a Reservoir Network will not appear in the display area until you have created or opened a Simulation and activated an Alternative (Section 15.4.1).

15.2 Creating a Simulation

Once you have created a reservoir network, entered element data, and developed alternatives in the Reservoir Network Module, you can configure the model for a Simulation in the Simulation Module.

- From the **Simulation** menu, select **New**. Or, in the **Simulation Control Panel**, right-click on the currently-active **Simulation** or ***No Simulation*** folder to access the context menu. Select **New** (Figure 15.11). The **Simulation Period** dialog will open (Figure 15.12).
  - The **Name** field contains a default name for the Simulation based on the current date and time. You may either accept the default or enter a name that is more meaningful to you.
  - Enter a **Start date** and **Time** specifying when you want the Simulation to begin. It must occur after the Lookback Date.

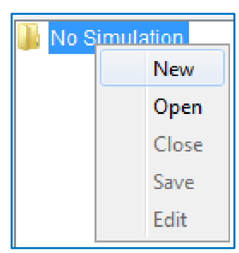


Figure 15.11 Simulation Control Panel Context Menu—New Simulation

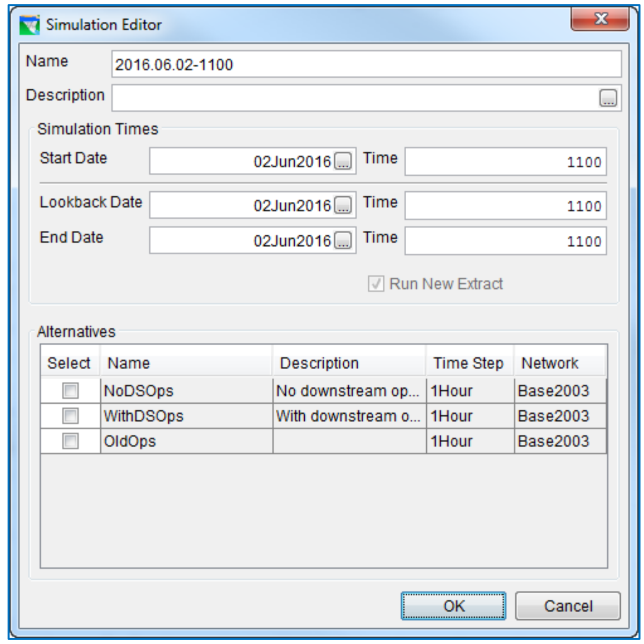


Figure 15.12 Simulation Period

- Enter a **Lookback Date** and **Time.** The Lookback is the "warm-up" period before the Simulation begins.
- Enter an **End Date** and **Time** specifying when you want the Simulation to conclude.
- Choose a **Time Step** from the list. The Time Step (or timestep) is the computation interval and can be 15 Minutes, 30 Minutes, 1 Hour, 3 Hours, 6 Hours, 12 Hours, or 1 Day.
- The **Alternatives** table includes all of the Alternatives you have defined in the Reservoir Network Module. Select one or more applicable Alternatives for the Simulation by checking the boxes next to them in the **Select** column. Ensure the Alternatives you select include time-series date for the entire Simulation period.

- Click **OK** to close the Simulation Period dialog. The **Creating Simulation** window (Figure 15.13) will inform you of the status as ResSim creates the Simulation you have defined.

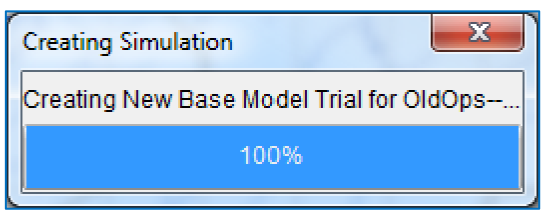


Figure 15.13 Creating Simulation Window

The Simulation you have created will now appear in the Simulation Control Panel.

> ***Tip*** The Reservoir Network will not appear in the map region until you have set an Alternative to Active (Section 15.4.1).

15.3 Working with Existing Simulations

If you have previously created a ResSim Simulation, you may want to make revisions to the Lookback date and time or End date and time (the Start date and time cannot be changed), or you may want to revise data in an Alternative (operation rules, reservoir operation zones, etc.). To accomplish this, you will open an existing Simulation.

15.3.1 Opening an Existing Simulation

To open an existing Simulation:

- From the **Simulation** menu, select **Open**. Or, in the **Control Panel**, right-click on the currently-active Simulation or *No Simulation* folder to access the context menu (Figure 15.14). Select **Open**.

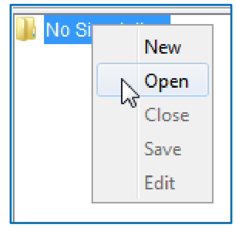


Figure 15.14 Simulation Control Panel Context Menu—Open Simulation

- The **Open Simulation** dialog will open (Figure 15.15).

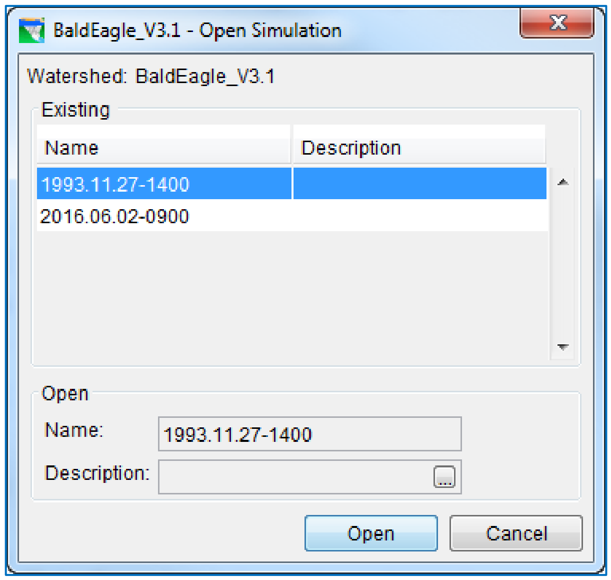


Figure 15.15 Open Simulation

- Click on the Simulation you want and select **Open**.
- The time period details of your selected Simulation will now appear in the Simulation Tree (Figure 15.16), and the map display will now update to show the model schematic for the **Active Alternative** (see Section 15.4.1 for information about setting the alternative to active).

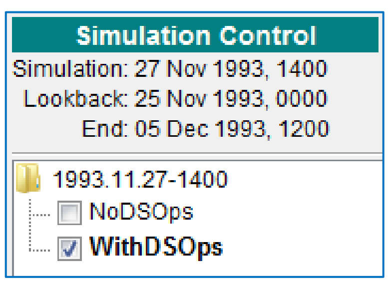


Figure 15.16 Simulation Tree

15.3.2 Editing a Simulation

Once you have created a Simulation, you can use the **Simulation Period** dialog (Figure 15.12) to edit the Lookback Date and Time, the End Date and Time, Time Step, and selected Alternatives. *You cannot change the Start Date or Time*.

You can access the **Simulation Period** dialog two ways:

- Select **Edit** from the **Simulation** menu.
- Right-click on the **Simulation folder** at the top of the simulation tree in the Simulation Control Panel, then select **Edit** from the context menu.
- When editing a simulation, it may be necessary to recopy the time series data to the simulation.dss file used by ResSim for its computations. There are two methods available for doing this:
- From the Simulation Period dialog, check the Run New Extract box and click OK.
- In the Menu Bar, click Simulation→Rerun Extract.

15.4 Computing a Simulation

After you have created a new simulation or have opened an existing simulation, you are ready to make one of the alternatives active (if not already active) and have ResSim perform the computations.

15.4.1 Setting the Active Component

Once you have opened a Simulation, the simulation tree will show all of the selected Alternatives for the Simulation you have chosen.

The **Active** alternative of a simulation will be identified in the Simulation tree with **bold** text and its model schematic will be displayed in the map region. It will be computed if you press **Compute** in the Simulation Control Panel.

Most importantly, the Active alternative is *editable*. Only the Active alternative can be edited in the Simulation module. If you need to edit more than one alternative in your simulation, you must do so one at a time, making each alternative "active" in turn to make the desired changes. Section 15.7 describes the ways you can edit the various elements of the active alternative.

To set the Active Alternative:

- In the simulation tree, right-click on the Alternative you want to be active.
- Select Set as Active from the context menu (Figure 15.17).

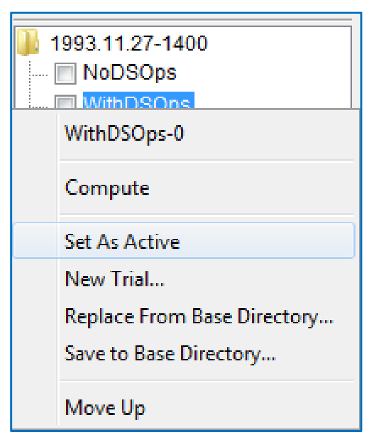


Figure 15.17 Simulation Control Panel Context Menu—Set Alternative As Active

15.4.2 Computing the Simulation

After inputting all data and parameters as desired, you can compute a Simulation.

In the Simulation Control Panel of the main window of the Simulation Module, the simulation tree displays the current Simulation as a folder, beneath which is a list of the Alternatives associated with the

Simulation. Also shown in the Simulation Control Panel is the time information associated with the Simulation.

To execute a Simulation, you must first set an Alternative as Active. Right-click on an Alternative in the Simulation Control Panel and, from the context menu, select **Set as Active** (Figure 15.17).

The name of the active Alternative appears in bold in the simulation tree and **Compute** becomes available. Also, the model schematic for the active Alternative will appear in the display area.

To compute a Simulation, either click **Compute** in the Simulation Control Panel or, in the simulation tree, right-click on the Alternative and select **Compute** from the context menu (Figure 15.18).

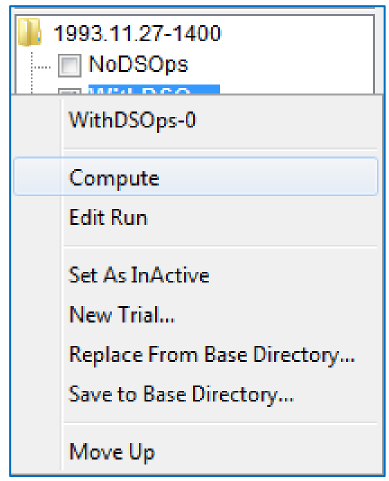


Figure 15.18 Simulation Control Panel Context Menu—Compute

When you compute a Simulation, a **Compute** window opens, as shown in Figure 15.19. The **Compute** window provides **Message Output** that contains information regarding the status of each step of the computation process. The **Progress Bar** indicates the percentage of completion for each step. When the computation is finished, the **Progress Bar** is completely filled in and reads "100%" along with the message "Compute Complete" in the **Message Bar** of the **Compute** window.

If there are errors or any problems during the execution process, you can review the Message Output Text area of the compute window. The **Compute Log** can also provide information regarding any errors (Section 14.5.1).

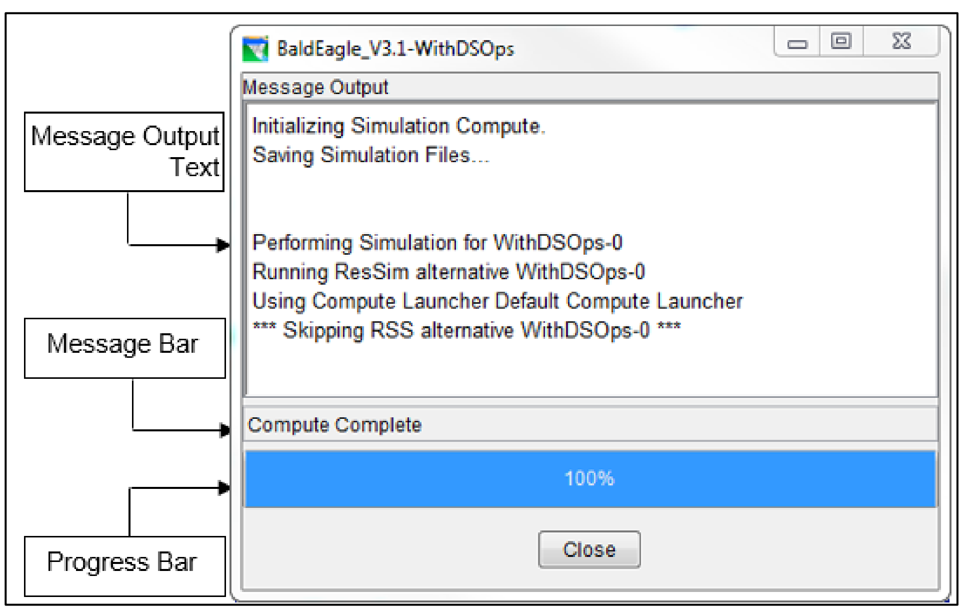


Figure 15.19 Compute Window

Click **Close** to close the Compute window.

As previously mentioned in Section 15.1.1, there are two tabs available from the **Tools Options** menu that are specific to the Simulation Module. These two options are *Simulation* and *ResSim Compute* and are described below:

- *Simulation* options—as shown in Figure 15.20, you can choose to Reload Last Simulation on Startup, Restore Simulation Tree State, and/or indicate that the **Compute Button Forces a Recompute**.

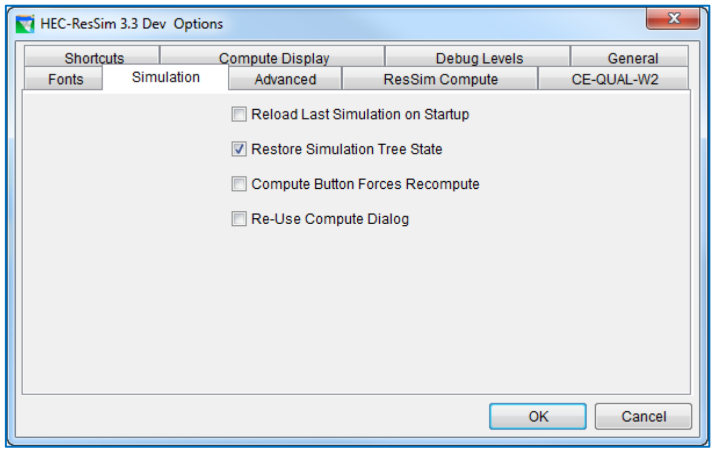


Figure 15.20 Simulation Module—Tools Menu—Options

- *ResSim Compute* options—as indicated in Figure 15.21, there are default compute options that you can revise if needed. These options include: **Minimum Number of Compute Passes** (default = 2); **Error Tolerance Factor for Storage Calculation** (default = 0.00001); **Maximum number of sub-steps for Storage Calculation** (default = 200); a checkbox to indicate whether or not you want to **Save Release to Guide Curve to DSS**; and, a button to **Edit Global ROC Options**. If you change from these values and want to go back to the defaults, then you can click **Reset Defaults**.

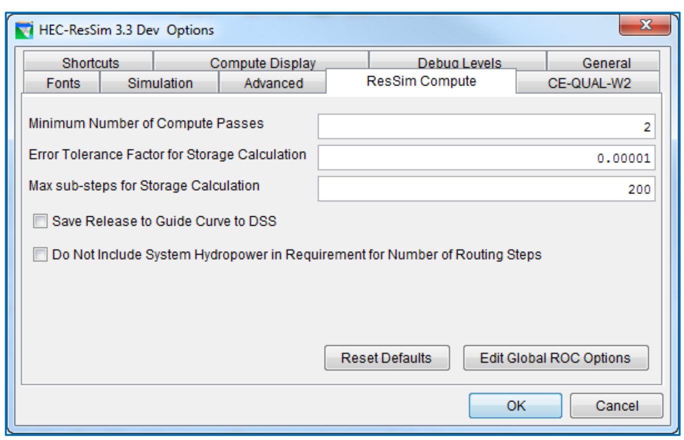


Figure 15.21 Simulation Module—Tools Menu—Compute

A **Compute Pass** refers to one ResSim program solution search moving from the upstream elements of the network to the downstream elements. The default is two passes. Certain system operation rules, such as system hydropower, automatically force four passes. A user may want to set the minimum number of passes to 3 or more if there are diversion elements in the network that are not performing correctly or that seem to have been ignored by upstream reservoirs.

The **Storage Calculation** refers to a ResSim storage integration approximation calculation performed across a computation step to assure the continuity principle is enforced over the outlet capacity range experienced during the timestep. If there are severe nonlinear shapes or discontinuities in the range of outlet capacities experienced during the timestep, then ResSim subdivides the time to better represent the average outflow during each time subdivision and therefore to better represent the storage at the end of the timestep. Storage calculation allows for a better simulation of small reservoirs subjected to large inflows and therefore large variations in outlet capacities during a timestep. If a small reservoir in a simulation has its storage oscillate unrealistically, increasing the **error tolerance** factor and/or the maximum number of **sub steps** may improve the behavior of the small reservoir.

For each time period, releases are determined that represent the amount of water that needs to be released in order to reach the reservoir's guide curve. By default, these releases are not stored to DSS. During analysis of the results, you may find it useful to know what these values were computed to be by selecting the **Save Release to Guide Curve to DSS** option.

If the operation set contains reservoir *rate of change* rule(s) and *downstream operation* rule(s), you can indicate to include complex logic to consider the rate of change limits when determining the release needed for downstream operation by revising the options in the **Global ROC Editor** (Figure 15.22).

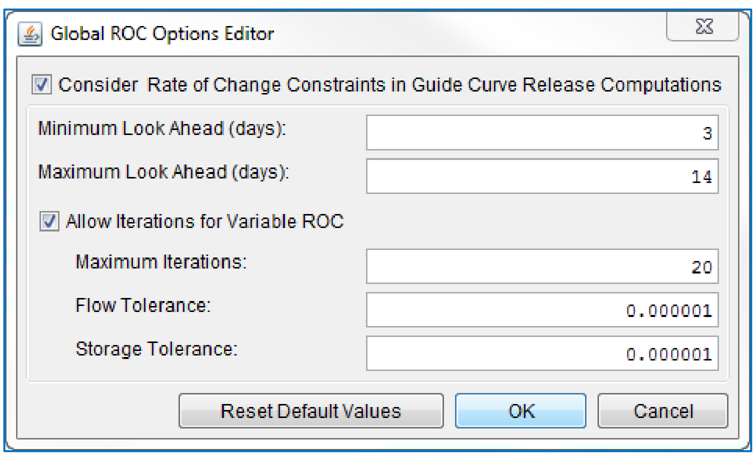


Figure 15.22 Global ROC Editor

15.5 Trials

After one or more alternatives have been added to a simulation, you can create a **Trial** of the alternative. A Trial is a copy of an existing alternative and can be used to quickly test "what if" scenarios involving various changes to the alternative (e.g. flows, rule priority, overrides, etc.). Unlike Alternatives, Trials are created in the Simulation Module. To create a Trial:

- In the **Simulation Control Panel**, right-click on the Alternative that you want to make a Trial for and select **New Trial…** (Figure 15.23).
- The **Create Trial Run** dialog will open.
- Enter a name for the Trial, preferably something descriptive (e.g. "2xFlows", "MaxFlowRulePriority") and then click **OK** to finish creating the Trial (Figure 15.24).
- The newly created Trial will appear in the **Simulation Control Panel**, nested beneath its parent alternative (Figure 15.25).

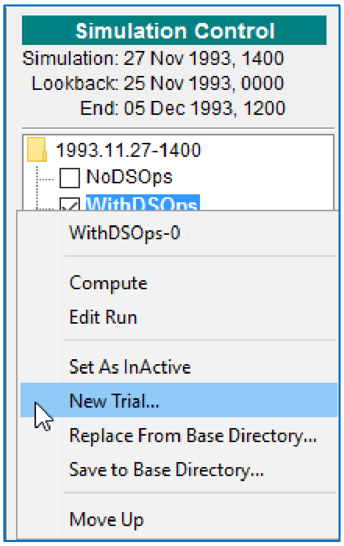


Figure 15.23 Creating a New Trial

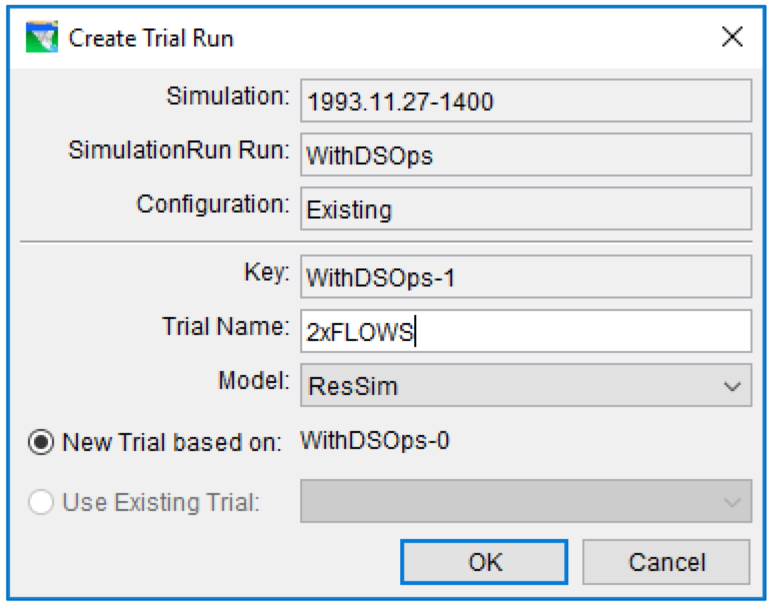


Figure 15.24 Create Trial Run Dialog

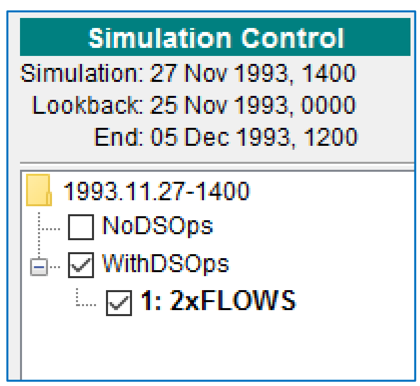


Figure 15.25 Trial Nested Under Parent Alternative

- After creating a trial, it should automatically be made active, but if not, right-click on the trial in the **Simulation Control Panel** and select **Set As Active**. The active trial or alternative will appear in bold text.
- Changes can be made to the trial either by:
  - right-clicking in the **Simulation Control Panel**, selecting **Edit Run**, and then selecting the portion of the trial that you wish to edit,

- right-clicking on a network component in the map window, or
- selecting **Alternative→Edit...** in the Menu bar of the Simulation Module

Trials can be reset to the original conditions of their parent alternative by right clicking on the Trial in the **Simulation Control Panel** and selecting **Restore from Original Alternative** (Figure 15.26). Alternatively, a trial can replace its parent alternative by selecting **Accept ResSim trial**.

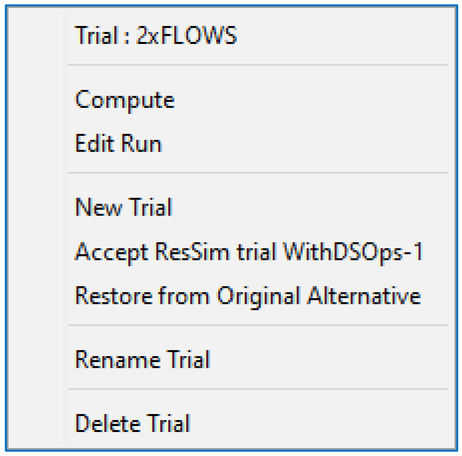


Figure 15.26 Simulation Control Panel Context Menu—Trials

15.6 Reviewing Simulation Results

After computing a Simulation, you can review results in many different forms. **Compute Messages** provide information about each step of the computation process. **Plots** and **Tables** in the Simulation Module offer detailed views of data and model results. **Reports** provide details about individual components of the Reservoir Network. These options for viewing your results are described in the following sections.

15.6.1 Selecting Alternatives for Plotting and Review

To view the results from an alternative, check the box next to the alternative(s) you want to plot or review.

15.6.2 Viewing Compute Logs

If there are errors or any problems during the computation process, the **Compute Log** (Figure 15.27) can provide information regarding the type of problem that exists. To view **Compute Logs**, select **Compute** from the **Reports** menu, then select the appropriate Alternative.

Tip Only Alternatives that have a checkmark to the left of the alternative name in the simulation tree will appear in the list for viewing Compute Reports. These checkmarks indicate the alternatives that you are interested in viewing results for.

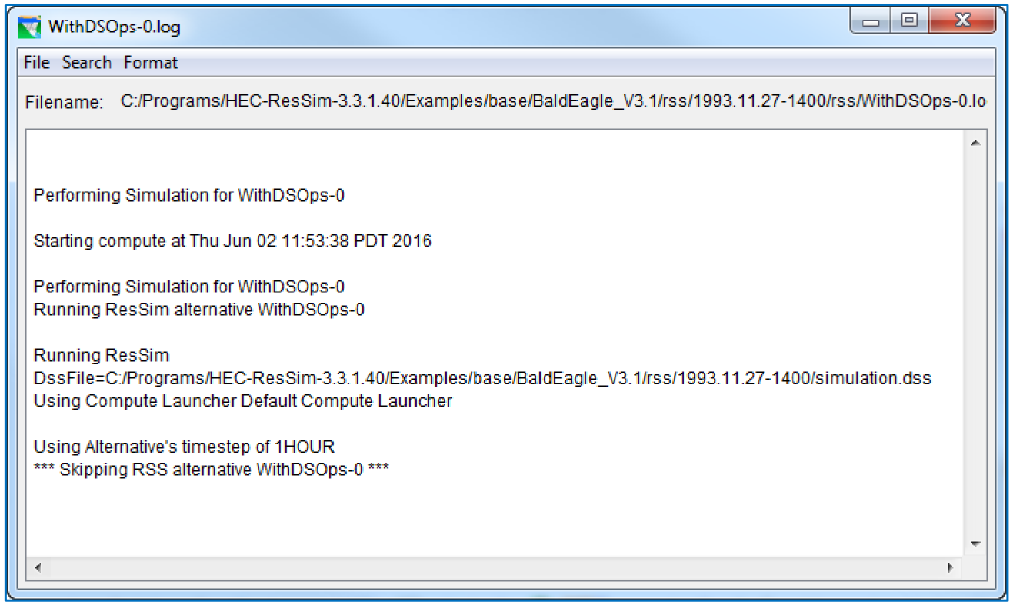


Figure 15.27 Compute Log

You can use the **Find** and **Find Next** commands in the **Search** menu to locate specific text in a log.

You can customize the appearance of the Compute Log with the **Colors** and **Font** commands in the **Format** menu.

The **Colors** command allows changes to the **Foreground** and **Background** colors. Selecting either one opens the **Select Color** dialog. See Appendix D for instructions on how to use the **Color Chooser** tools.

The **Font** command opens the **Select Font** dialog (Figure 15.28), which allows you to choose the font **Type** and **Size**. Also, you can choose whether or not to set the appearance to **Bold** and/or **Italic** characters.

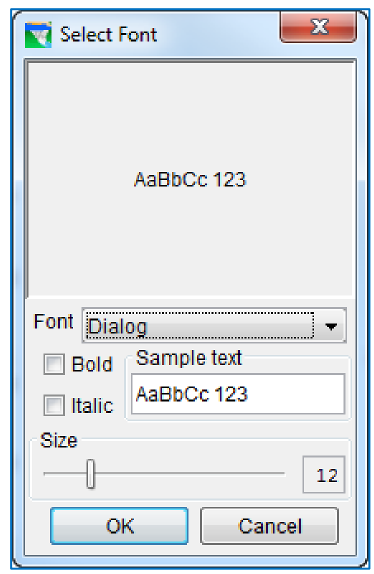


Figure 15.28 Compute Log—Format Menu—Select Font

15.6.3 Viewing the Alternative Input Report

The **Alternative Input** report provides a complete set of the data used by an alternative to compute results. To access the **Alternative Input** report, select **Reports→Alternative Input** and then select the desired alternative (Figure 15.29).

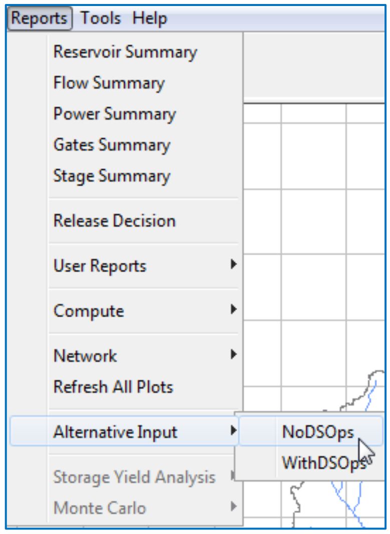


Figure 15.29 Selecting an Alternative Input Report

The Alternative Input Report Editor will open (Figure 15.30). From the editor, the user can specify the Output Type as either XML or HTML and choose which elements of the alternative should be included in the report.

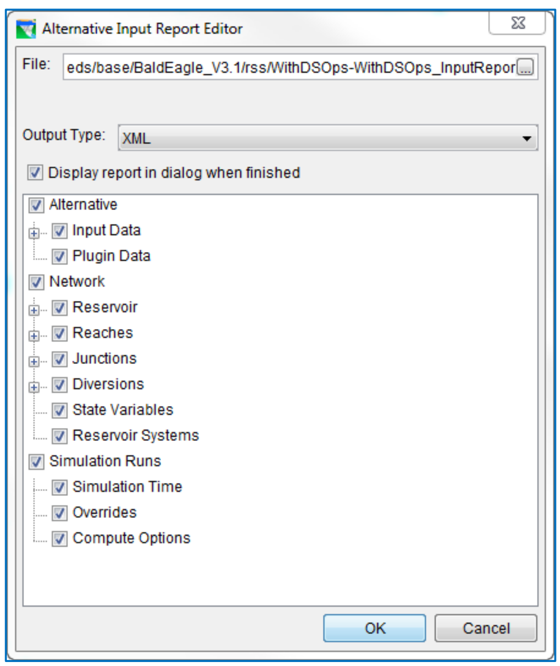


Figure 15.30 Alternative Input Report Editor

15.7 Calibrating the Model and Editing Data

You may need to make adjustments as you test and calibrate your reservoir Simulation model. In the Simulation Module of ResSim, you can access editors that allow you to edit all components of your Reservoir Network, modify Alternatives, and fine-tune Override controls.

15.7.1 Using the ResSim Editor Interface

In the Simulation Module, the **ResSim Editor Interface** (Figure 15.31) provides access to editors for Reservoirs, Junctions, Reaches, and Diversions, as well as State Variables, Systems Operations, the Alternative Editor and the Overrides Editor. You can also use the ResSim Editor Interface to quickly set Compute Options.

To access the ResSim Editor Interface, right-click on the ***Active Alternative*** in the Simulation tree, then select **Edit Run** (Figure 15.31).

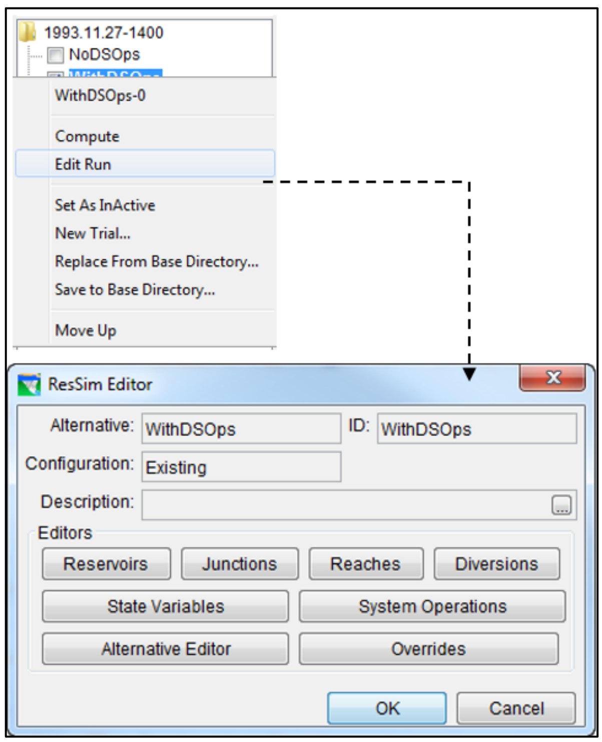


Figure 15.31 ResSim Editor Interface in Simulation Module

15.7.2 Editing Alternative Lookback, Time Series, Observed, and System Operations Data

To edit *references* to Time Series and Observed data, you will need to access the individual editors for **Reservoirs**, **Junctions**, **Reaches**, and **Diversions**. Also, you may want to revise the **System Operations** storage balance. You can access each of these editors either from the **ResSim Editor** Interface (as previously shown in Figure 15.31) or from the **Edit** menu of the Simulation Module or Reservoir Network Module.

To select different Alternative **Operations** or to adjust **Lookback** data or revise the "mapping" of your **Time Series** and/or **Observed** data, you will need to access the **Alternative Editor** (Chapter 13). The Alternative Editor is available from the **ResSim Editor** Interface (as previously shown in Figure 15.31) or from the **Alternative** menu of the Simulation Module. For detailed descriptions of the element editors available in ResSim, refer to Chapters 9, 10, 11 and 12.

15.7.3 Editing Override Values

Once you have computed a Simulation, you may want to disregard (i.e., override) some of the decisions that were made. The **Overrides Editor** (Figure 15.32) allows you to adjust the computed results for each timestep of the Simulation. You can access the Overrides Editor using **Overrides** in the ResSim Editor Interface (as previously shown in Figure 15.31); or, you can select **Overrides** (for the appropriate Alternative) from the **Simulation** Menu.

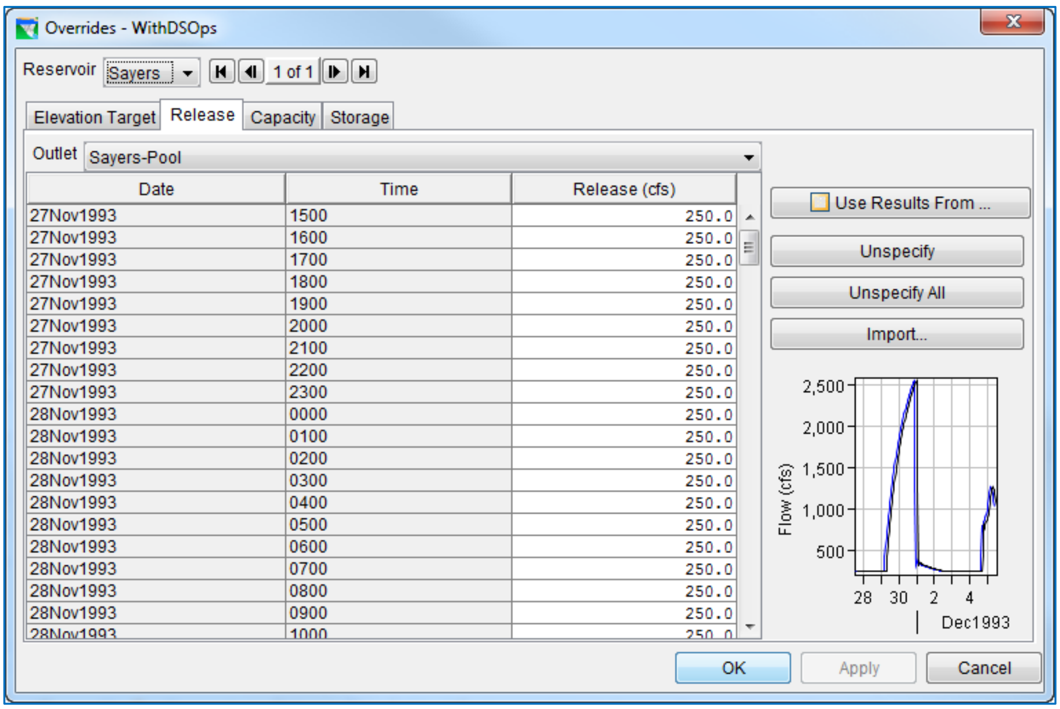


Figure 15.32 Overrides Editor

ResSim will use the values you specify for **Elevation Target** Overrides, **Release** Overrides, Outlet **Capacity** factors within the physical limits of the reservoir, and **Storage** (or Elevation) Overrides. The physical rate-of-change and the amount of water available in the reservoir may preclude ResSim from using your override values.

For the Reservoir shown in the list at the top of the Overrides editor, there are four tabs available for overriding the simulation results: Each tab contains a **Date**, **Time** and either "Elevation Target" or "Release" or "Factor" or "Storage." You can use any one of these override capabilities to adjust the simulation results.

Specifying Elevation Target Overrides:

Initially, the Target Elevation is based on the elevation values you specified for the reservoir's "guide curve" (Reservoir Editor, Operations tab). This is typically the top of the Conservation pool and is commonly referred to as the "target" or "guide curve". In addition to meeting all of the other rules that you specify for a reservoir, ResSim will try to keep the reservoir pool at the guide curve elevation. Therefore, by specifying Elevation Target values within this editor, you are inherently overriding the reservoir's release decisions.

Specifying Release Overrides:

During a Simulation, ResSim determines the reservoir release values based on the rules you specified for the Alternative. However, there may be situations where you do not want the results to reflect the rules for specific timesteps. Therefore, you can enter release values for the pool, the dam, an outlet group, or an individual outlet using the Release overrides tab.

Specifying Capacity Factor Overrides:

The maximum release capacity of an outlet can be increased or reduced by using a factor for the time periods desired. For example, if you want to indicate that an outlet is "out-of-service" for a specific time window within the simulation, you can enter a value of 0 to indicate no release capacity for that outlet.

Specifying Storage (or Elevation) Overrides:

A warning is made to the user when selecting this override option because the mass balance computations can be disrupted by forcing the storage or elevation to be specific values.

Editing Override Values

You can specify Overrides by entering single cell values or by revising multiple adjacent values using a fill function.

To revise a *single* value, double-click in the cell you want to revise and enter the new value.

To revise *multiple* adjacent values:

- Point and click on the first cell, then drag your mouse or Shift+click (hold down the shift key while clicking in other cells) to highlight the cells to be revised.
- Right-click on the highlighted cells, and select **Fill** from the context menu (Figure 15.33).
- The **Table Fill Options** dialog will open (Figure 15.34).

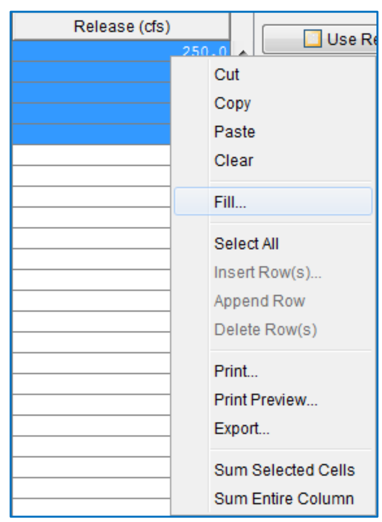


Figure 15.33 Release Overrides Editor Context Menu—Fill Data Values

- Select the appropriate fill option in the **Table Fill Options** dialog (Figure 15.34), then click **OK**.

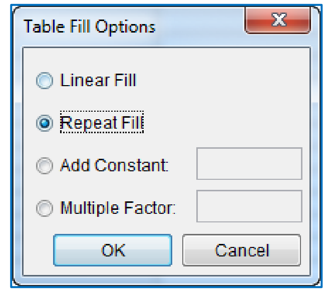


Figure 15.34 Table Fill Options

The revised values will reflect the fill option you selected. In the example shown in Figure 15.34, the **Repeat Fill** option is selected. Therefore, as the table in Figure 15.35 illustrates, all of the highlighted cells equal the value of the first cell selected; also, the color of the revised values changes from black to blue and the thumbnail plot shows the revised values as a red line. The thumbnail plot can be viewed in full size when you double-click on it.

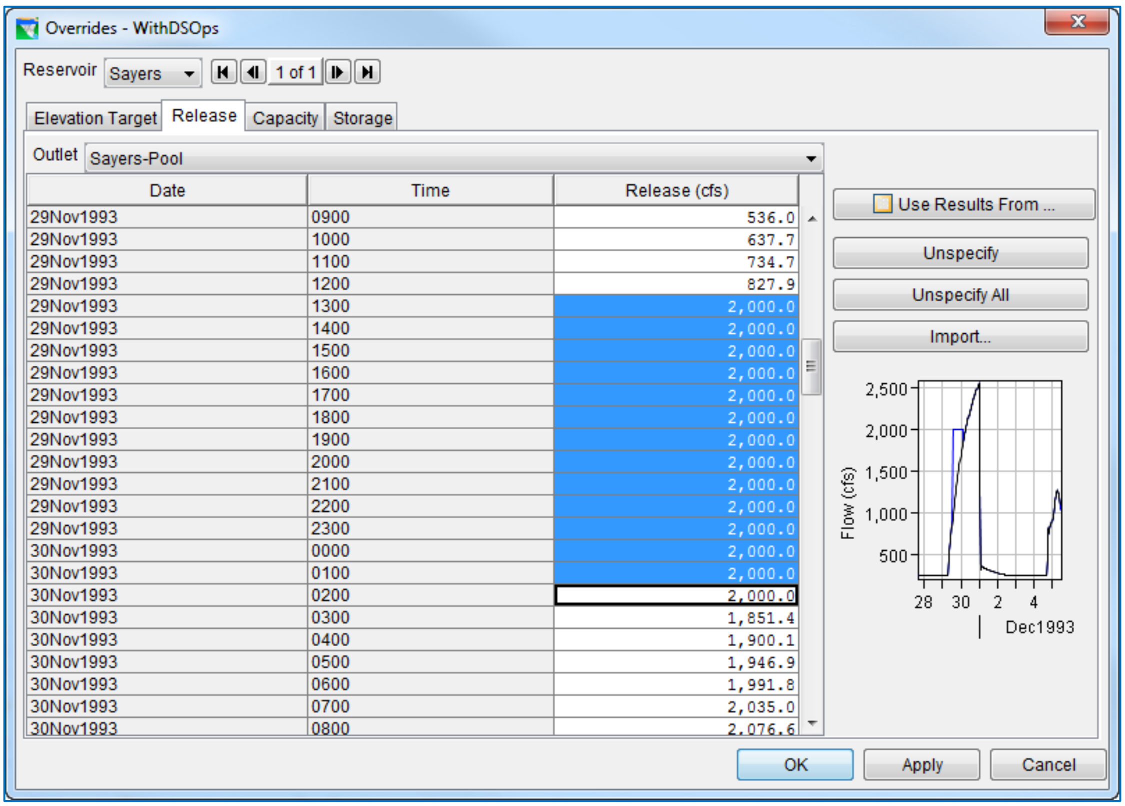


Figure 15.35 Release Overrides Editor Table—Revised Data Values using Repeat Fill Option

For those timesteps 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. In that case, you can use **Unspecify** and **Unspecify All** to indicate that you want ResSim to determine the release values.

Use **Unspecify** when you want release values to be based on the reservoir rules for some of the override values you have specified (override values are in green text after a simulation is computed). Highlight the cells where you have specified override values, then press **Unspecify**.

Use **Unspecify All** when you want all of the release values to be based on the reservoir rules and not have any overrides specified.

Use **Import** when you want to use a time series of override values. The **Import Overrides Time Series** dialog (Figure 15.36) will open, which functions exactly like the **Select Time Series Path** dialog used to select time-series records for an Alternative. Refer to Section 14.9 for more information.

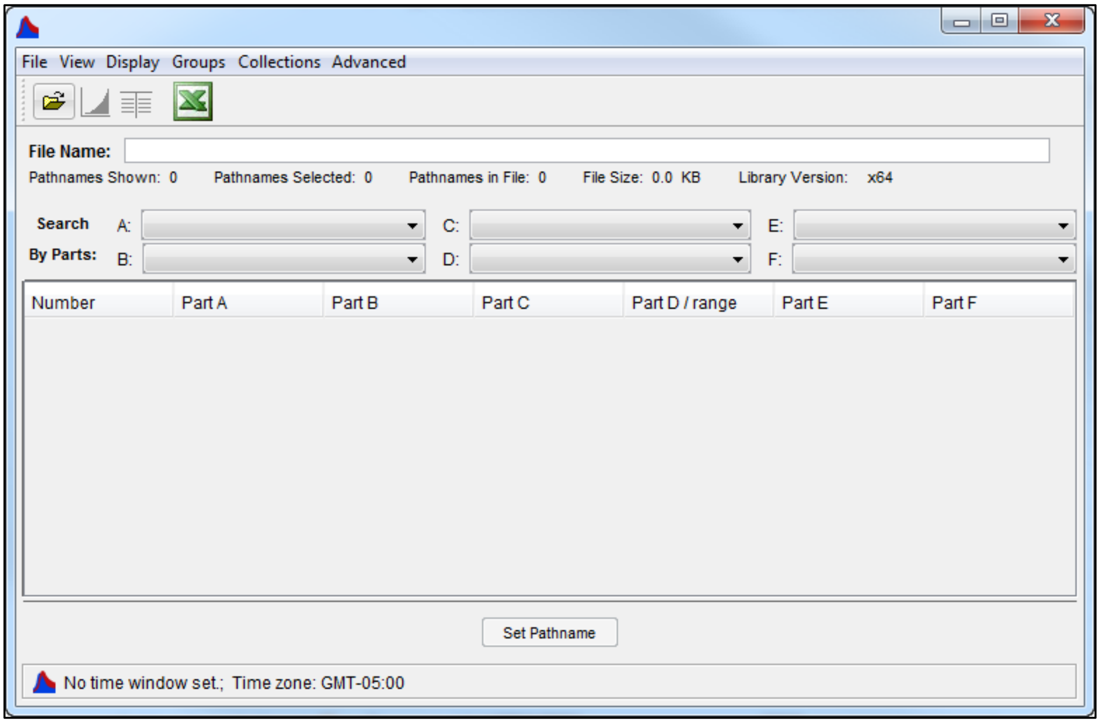


Figure 15.36 Import Overrides Time Series

15.8 Managing Simulation Data

ResSim facilitates archiving and sharing of Simulation data. There are two operations involved in managing Simulation data. 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. Figure 15.37 shows the relationship between the **Reservoir Network** module and the **Simulation** module for saving and replacing data.

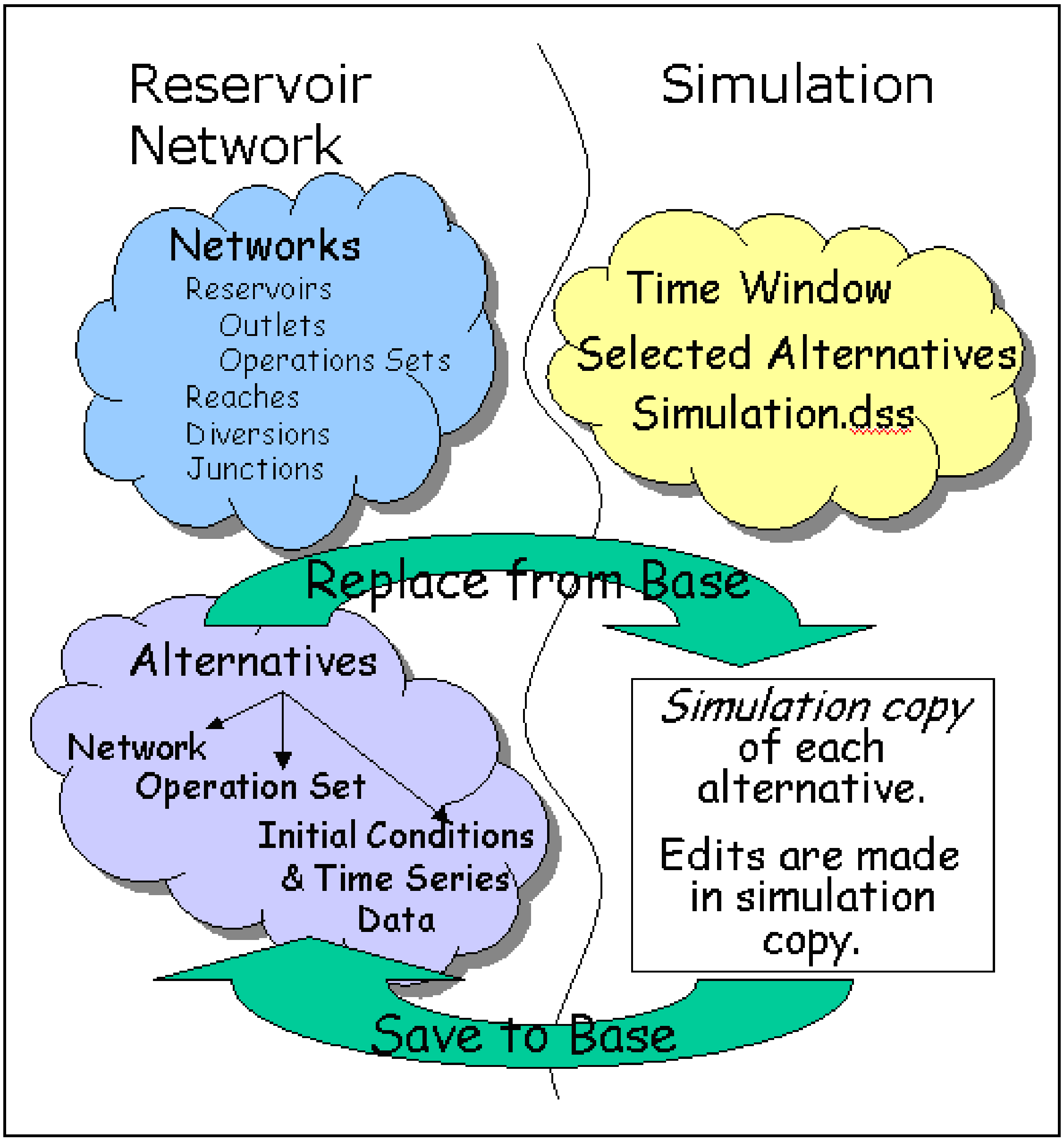


Figure 15.37 Data Relationship between Reservoir Network and Simulation Modules

15.8.1 *Saving* Data *to* the Base Directory

When you edit model data from the Simulation Module, your changes apply only to an individual Alternative and are saved in your Simulation directory (see Appendix A for an overview of the ResSim directory structure).

Important: If you want your changes to be available for use in other Simulations, you will need to Save the data back to the Base directory.

To ***save*** data ***to*** the Base directory:

- In the Simulation Control Panel, right-click on the Alternative to access the context menu, then choose **Save to Base Directory**... (Figure 15.38). The Save Simulation Run Model Parameters to Base Directory dialog will appear (Figure 15.39).

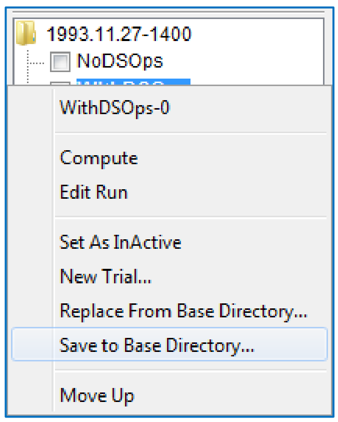


Figure 15.38 Alternative Context menu—Save to Base Directory

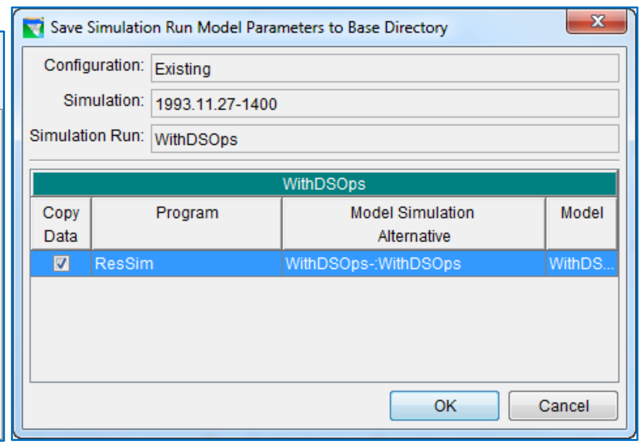


Figure 15.39 Save Simulation Run Model Parameters to Base Directory

- Select the appropriate boxes in the **Copy Data** column beside the items you wish to copy from the Simulation directory to the Base directory (i.e., save back to the Reservoir Network).
- Click **OK** to save the Simulation data to the Base directory.

15.8.2 *Replacing* Data *from* the Base Directory

While editing Simulation data, if you need to revert to the original Alternative data (as it exists in the Reservoir Network module), you can *replace* the changed data in your Simulation directory with data *from* the Base directory.

To ***replace*** Simulation data with data ***from*** the Base directory:

- From the Simulation Control Panel, right-click on the Alternative to access the context menu, then choose **Replace from Base Directory**. The Replace Simulation Run Model Parameters from Base Directory dialog will appear (Figure 15.40).

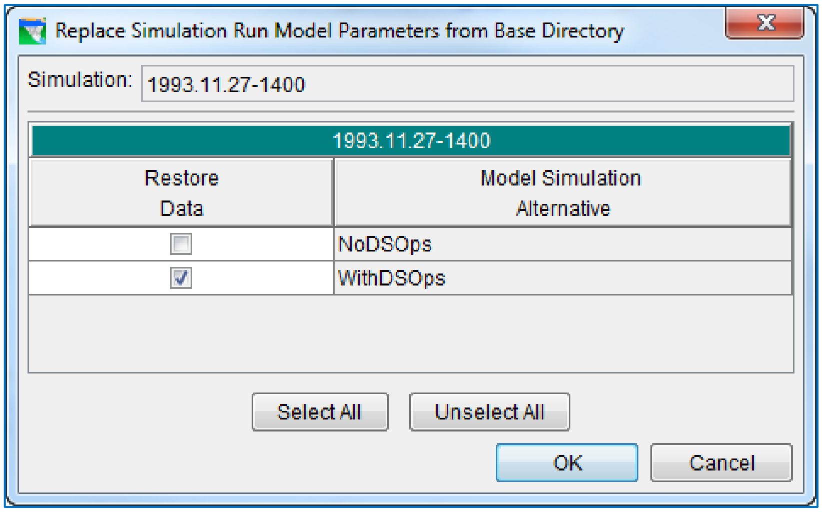


Figure 15.40 Replace Simulation Run Model Parameters from Base Directory

- Select the appropriate boxes in the **Restore Data** column beside the items you wish to restore from the Base directory to the Simulation directory.
- Click **OK** to replace the Simulation Alternative data you have selected with data from the Base directory.

The **Replace from Base...** option is also available from the Simulation menu.

15.9 Using HEC-DSSVue

Included within the framework of ResSim is **HEC-DSSVue**, a tool that allows you to access data stored in HEC-DSS database files.

When **HEC-DSSVue** is selected from the **Tools** menu (Figure 15.41) within the Simulation Module, the current "simulation.dss" file is opened.

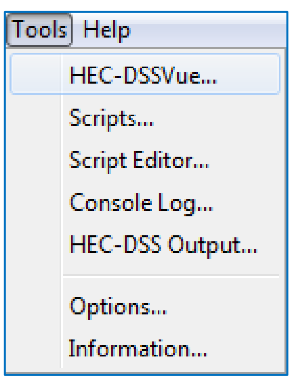


Figure 15.41 Accessing HEC-DSSVue from the Tools Menu

In the Main Window of HEC-DSSVue, a listing of pathnames that are contained in the simulation.dss file are provided, as shown in Figure 15.42.

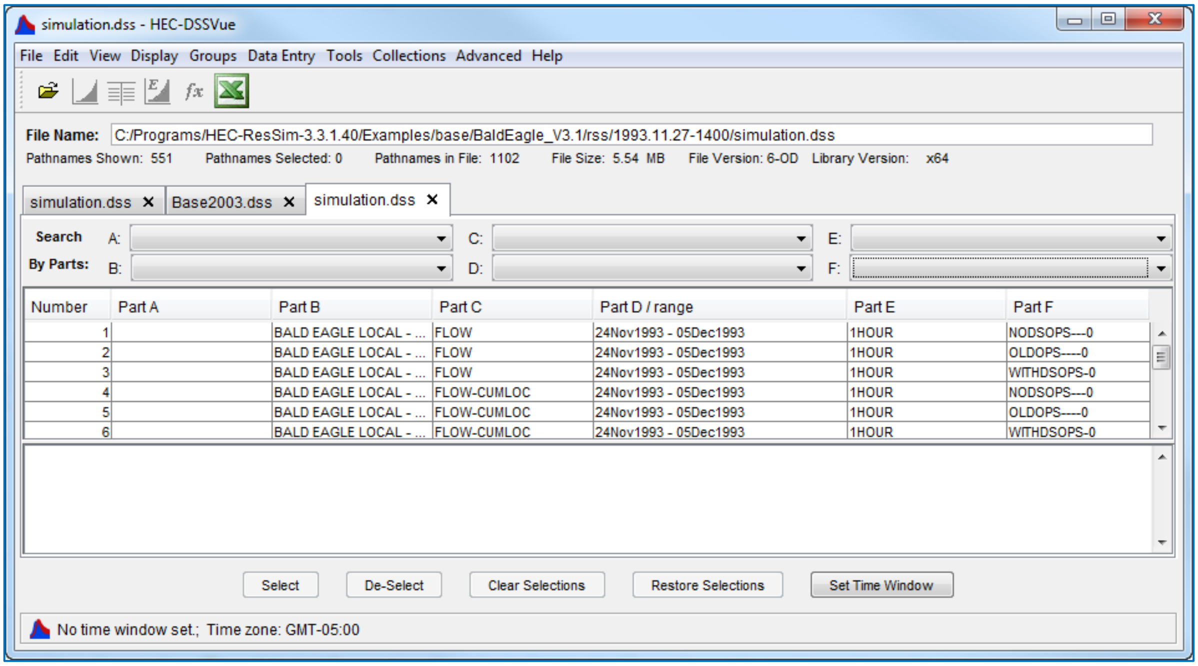


Figure 15.42 HEC-DSSVue Main Window Showing Pathname Listing

A screened listing of pathnames can be obtained by selecting a pathname part from the lists in the **Search by Parts** section of the window. For example, if you want a listing of "observed" records, you can select OBS from the F-part list, as shown in Figure 15.43. To obtain an unscreened listing of pathnames, select the "blank" area at the top of the list.

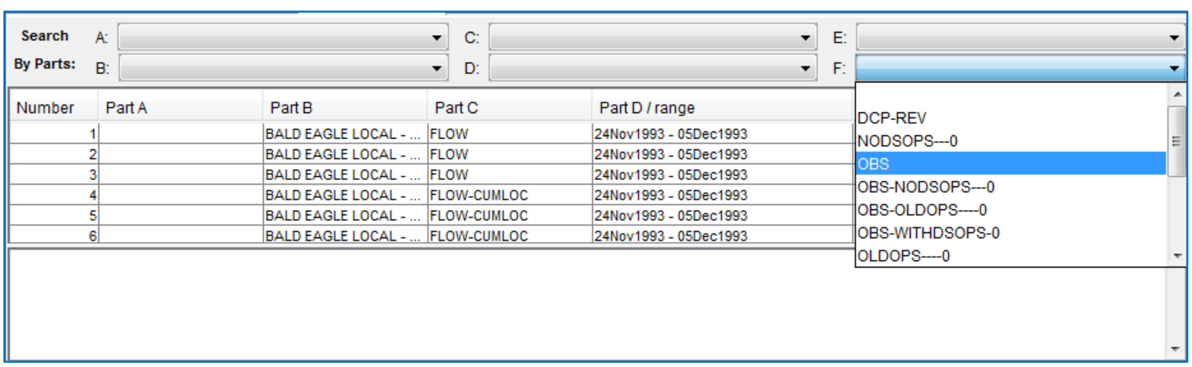


Figure 15.43 Screened Pathname Listing Showing Observed (OBS) Records

To select records to be plotted, tabulated, or edited, highlight the desired pathnames and click on **Select**. After one or more records are selected, the icons for the graph  and table  become active. Now, you can click on either icon to generate a plot (Figure 15.44) or tabulated values (Figure 15.45) of the selected records.

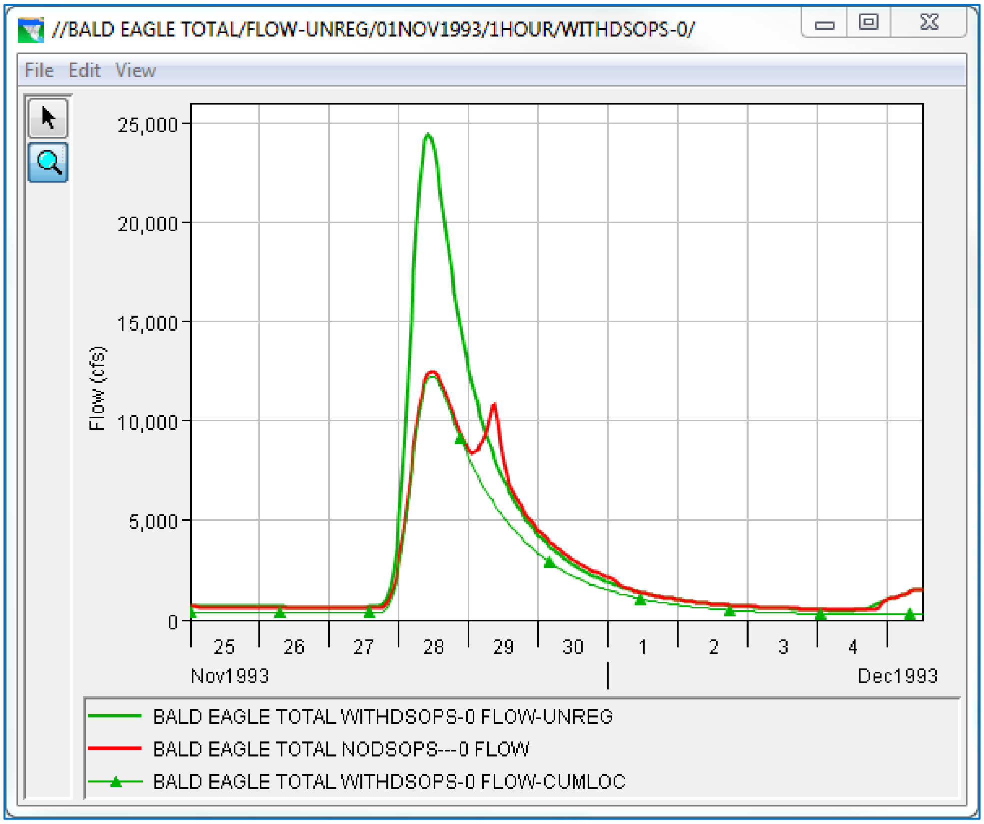


Figure 15.44 Example Plot Using HEC-DSSVue

//BALD EAGLE TOTAL/FLOW-UNREG/01NOV1993/1HOUR/WITHDSOPS-0/

File Edit View

Ordinate	Date	Time	BALD EAGLE TOT... FLOW-UNREG WITHDSOPS-0	BALD EAGLE TOT... FLOW NODSOPS---0	BALD EAGLE TO... FLOW-CUMLOC WITHDSOPS-0
Units			cfs	cfs	cfs
Type			INST-VAL	INST-VAL	INST-VAL
1	24 Nov 93	24:00	710	655	400
2	25 Nov 93	01:00	710	655	400
3	25 Nov 93	02:00	709	654	399
4	25 Nov 93	03:00	708	653	398
5	25 Nov 93	04:00	707	652	397
6	25 Nov 93	05:00	705	651	396
7	25 Nov 93	06:00	704	650	395
8	25 Nov 93	07:00	702	648	393
9	25 Nov 93	08:00	700	647	392
10	25 Nov 93	09:00	698	645	390
11	25 Nov 93	10:00	696	644	389
12	25 Nov 93	11:00	694	643	388
13	25 Nov 93	12:00	692	641	386
14	25 Nov 93	13:00	690	640	385
15	25 Nov 93	14:00	688	639	384
16	25 Nov 93	15:00	686	638	383
17	25 Nov 93	16:00	685	636	381
18	25 Nov 93	17:00	683	635	380
19	25 Nov 93	18:00	681	634	379
20	25 Nov 93	19:00	679	633	378
21	25 Nov 93	20:00	677	631	376
22	25 Nov 93	21:00	675	630	375

Figure 15.45 Example of Tabulated Data Using HEC-DSSVue

For a more detailed description of HEC-DSSVue, see Appendix D.

Chapter 16
Plotting Results

ResSim has many standard plots for viewing output, and you can customize and create your own, as well. This chapter describes some of the main plotting features available in ResSim.

16.1 Using Plots and Tables

Plots and tables in the Simulation Module offer detailed views of data and model results.

You can access plots using context menus in the Simulation Module's display area.

To access a **Plot** from the display area, right-click on a model element in the map display. The context menu will provide a list of one or more plot options, as illustrated in Figure 16.1.

Once you have opened a plot, you can also tabulate values by selecting **Tabulate** from the plot's **File** menu.

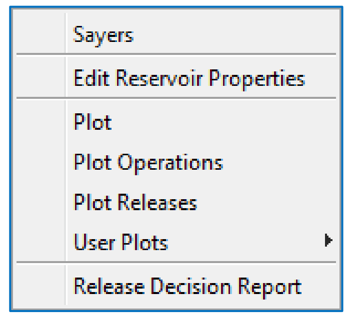


Figure 16.1 Reservoir Context menu—Plot Simulation Results

16.1.1 Features of Plots

ResSim plots offer a variety of information that will assist you with reviewing the results of a Simulation. Included in the information available from the default plots are reservoir elevation, storage, and release values as well as regulated and unregulated flow values. Figure 16.2 shows a default plot style illustrating reservoir results from a Simulation.

The plot window displays the location name in the title bar. Axis labels and a color-coded legend identify the data contained in the plot.

When a plot depicts the results of an Alternative, as in Figure 16.2, a dashed vertical line represents the start time of the Simulation. The Lookback (historic/observed) period occurs prior to the start time.

The **Zoom Tool**  allows you to view data closely at a specific time. To zoom in, hold the left mouse button down and outline the area you want to enlarge. To zoom out, click the right mouse button. To resize a plot, use the mouse to drag the edges of the window. It is possible to zoom in on one section of the entire plot (both the x and y variables simultaneously) or a range for either variable. To zoom in on a section of the plot, outline that area on the plot. To zoom in on a range (for either x or y variable), outline the desired range on the respective axis.

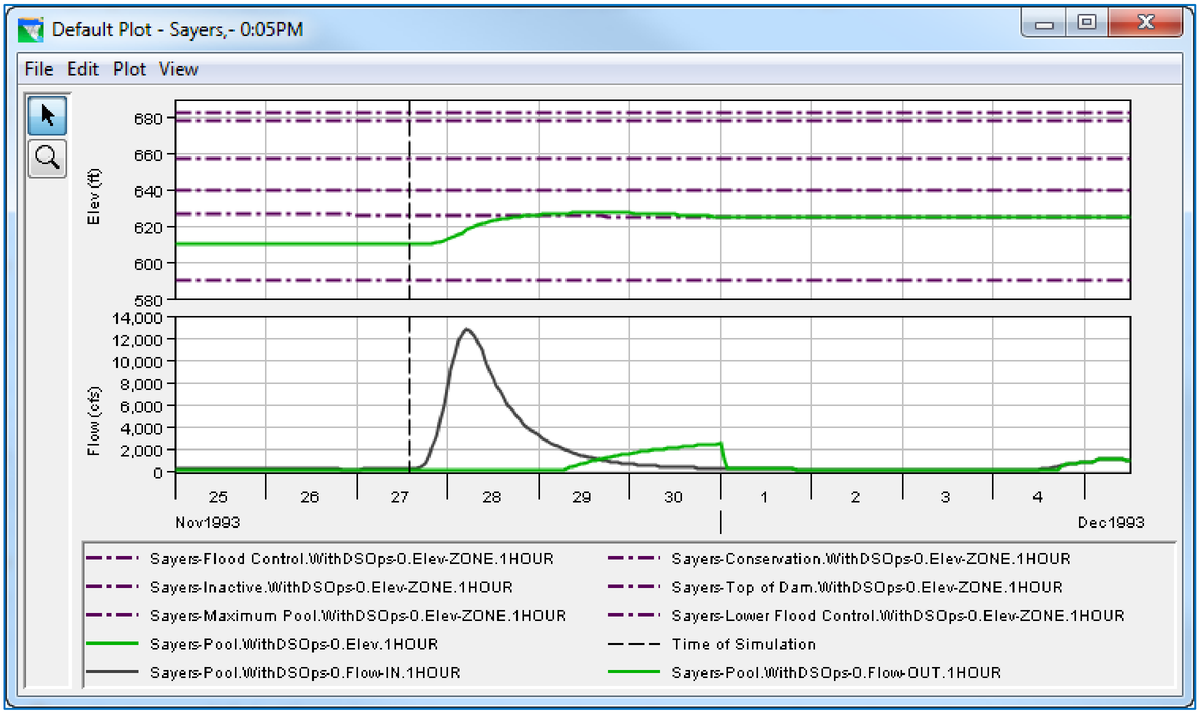


Figure 16.2 Sample Plot of Reservoir Results

From a plot window, the **View** menu provides additional controls for the display of results such as quickly displaying the entire plot after zooming in (**Zoom to All**), reposition or hide the legend (**Legend Placement/Hide Legend**), refresh the plot after modifying the alternative and recomputing (**Refresh**), and refresh the plot on the fly after recomputing (**Live Display…**).

16.1.2 Customizing Plots

You can customize the appearance of plots by using several properties editors that you can access from context menus:

- **Line Properties:** Right-clicking on a plot line or point will allow you to open a Curve Properties Editor to edit line colors, styles, and weights, as well as labels and quality symbols.
- **Background Properties:** Right-clicking on the background of a plot will allow you to add Markers on the X- or Y-Axis. Also, you can open a Viewport Editor where you can customize the border, background, and gridlines of the plot.
- **Axis Properties:** Right-clicking on a plot axis will allow you to set the Axis Type to "Log Axis" (or "Linear Axis"). Also, you can open an Axis Properties Editor where you can customize the axis scale and tic marks.
- **Label Properties:** Right-clicking on an axis label or plot legend will allow you to open a Label Properties Editor where you can add backgrounds and borders to the labels.
- **Legend Properties:** Right-clicking in an empty area in the legend region will allow you to open a Legend Properties Editor where you have a variety of options for

determining the appearance of the legend. Or, you can select Hide Legend from the right-click menu (to show the legend, select Show Legend from the View menu).

16.1.3 Creating User-Defined Plots

Since everyone has their own preferences regarding which variables to include in the context menu location plot selections, the **User Defined Plot** capability offers flexibility.

To create a User Defined Plot:

- Right-click on the element where you want the plot to appear, and select **Plot** from the context menu (Figure 16.3). The default plot for that location will appear.
- Choose **Select Variables** from the **Plot** Menu. The **Select Plot Variables** dialog will appear (Figure 16.4).

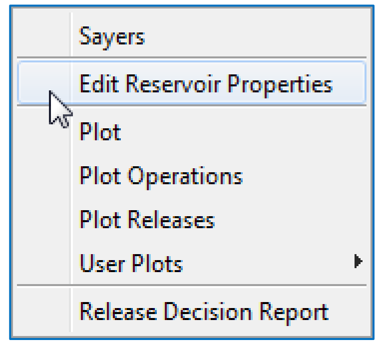


Figure 16.3 Context Menu for an Element

The **Available Variables** list shows the variables you can choose for your User Defined Plot. The **Selected** list shows variables you have selected.

To add variables to your User Defined Plot:

- Click on the variable in the **Available Variables** list, then click on the appropriate Plot Region or component in the **Selected** list. This allows you to specify where you want variables to appear in your plot.
- Click **Add** to add the variable you have chosen to the Selected list. If the added variable reflects different units, a separate Y-axis will be automatically added.

To remove a variable, click on its name in the **Selected** list then click **Remove**.

You can also **Move** a Plot Region up or down or **Remove** it by right-clicking on the plot region name and choosing the action desired from the context menu, as illustrated in Figure 16.5.

- Click **OK**. The Select Plot Variables dialog will close, and your User Defined Plot will open.

To save your User Defined Plot:

- From the **Plot** menu on the plot that appears, select **Save Plot Type...** The **Save Plot Type** dialog will appear (Figure 16.6). You can specify whether you want this User Defined Plot to be available to **All Applications** or to just **This Watershed only**. Enter a **Name** for the plot.
- Click **Save**. The Save Plot Type... dialog will close.

The new User Defined Plot will now be available when you select **User Plots** from the right-click context menu in the Display Area (Figure 16.7).

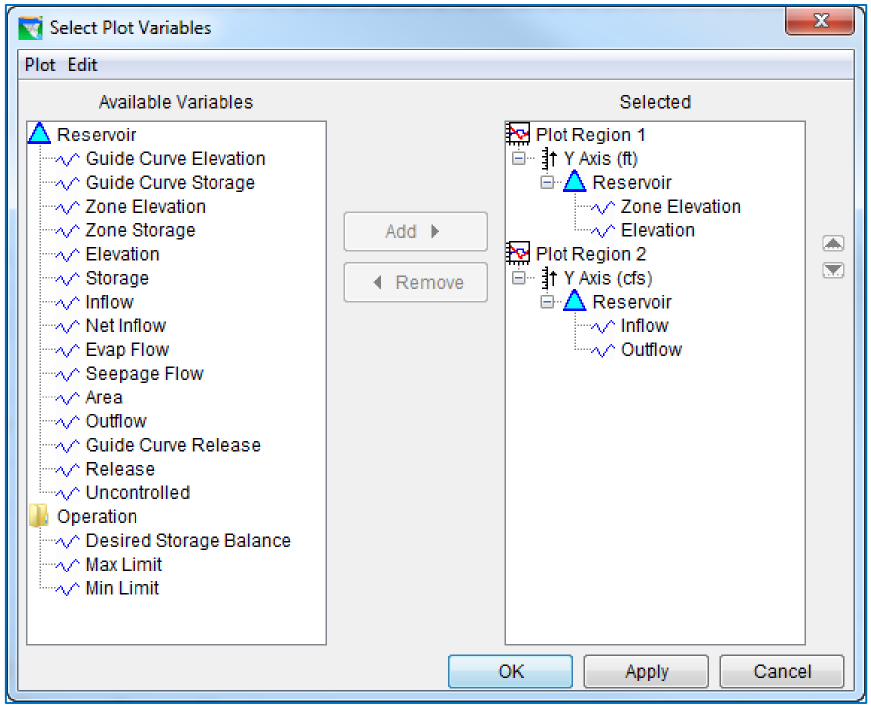


Figure 16.4 Select Plot Variables

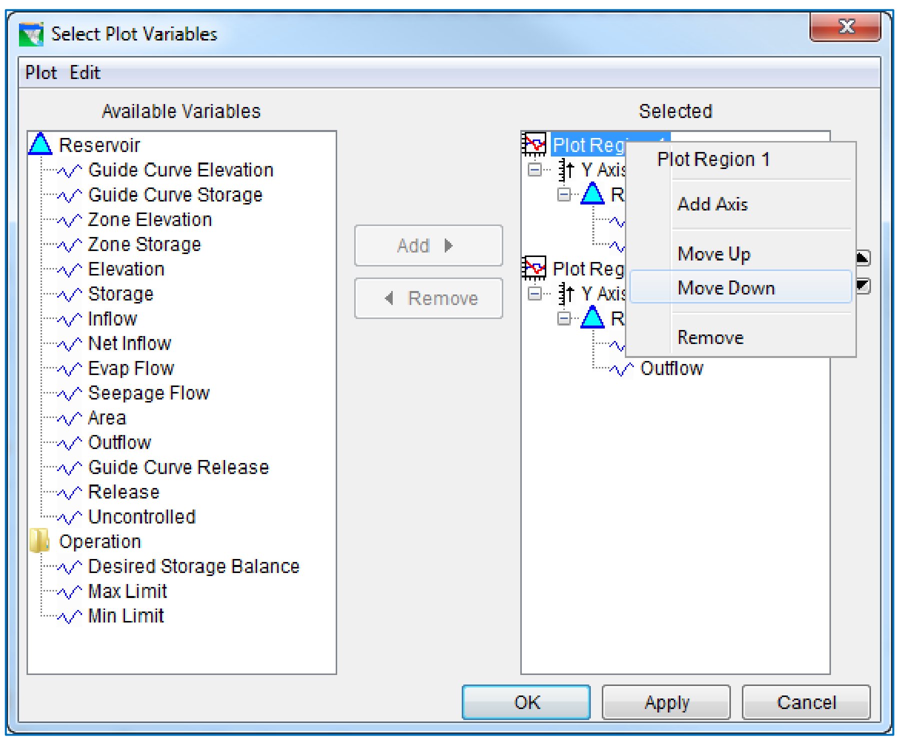


Figure 16.5 Select Plot Variables—Plot Region Context Menu

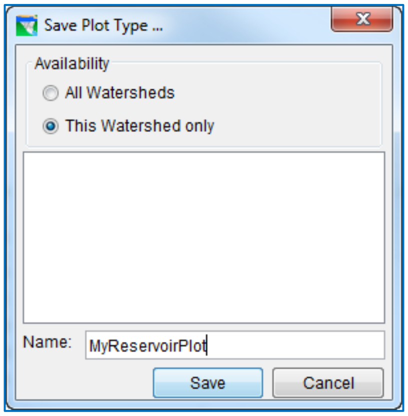


Figure 16.6 Save Plot Type

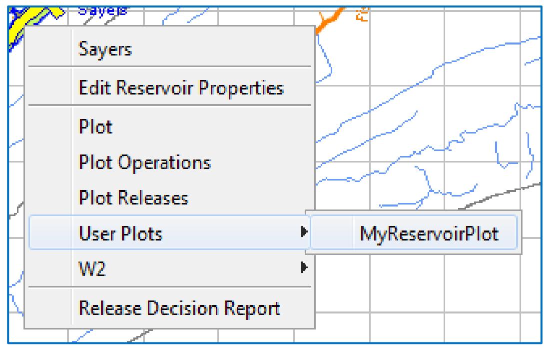


Figure 16.7 Context menu—User Plots

16.2 Viewing Data in Tabular Form

You can view plotted data in tabular form by selecting **Tabulate** from the **File** menu of the plot. Figure 16.8 shows an example.

//Sayers-Flood Control/Elev-ZONE/24Nov1993/1HOUR/WithDSOps-0/

File Edit View

Ordin...	Date / Time	Sayers-Fl... Elev-ZONE WithDSO...	Sayers-C... Elev-ZONE WithDSO...	Sayers-In... Elev-ZONE WithDSO...	Sayers-T... Elev-ZONE WithDSO...	Sayers-M... Elev-ZONE WithDSO...	Sayers-L... Elev-ZONE WithDSO...	Sayers-P... Elev WithDSO...	Sayers-P... Flow-IN WithDSO...	Sayers-P... Flow-OUT WithDSO...
Units		ft	ft	ft	ft	ft	ft	ft	cfs	cfs
Type		INST-VAL	INST-VAL	INST-VAL	INST-VAL	INST-VAL	INST-VAL	INST-VAL	INST-VAL	INST-VAL
1	24 Nov 1993, 2...	657.00	626.88	590.00	683.00	677.80	640.00	610.15	310	255.0
2	25 Nov 1993, 0...	657.00	626.86	590.00	683.00	677.80	640.00	610.15	310	255.0
3	25 Nov 1993, 0...	657.00	626.85	590.00	683.00	677.80	640.00	610.15	309	255.0
4	25 Nov 1993, 0...	657.00	626.84	590.00	683.00	677.80	640.00	610.15	309	255.0
5	25 Nov 1993, 0...	657.00	626.82	590.00	683.00	677.80	640.00	610.15	308	255.0
6	25 Nov 1993, 0...	657.00	626.81	590.00	683.00	677.80	640.00	610.15	308	255.0
7	25 Nov 1993, 0...	657.00	626.80	590.00	683.00	677.80	640.00	610.15	307	255.0
8	25 Nov 1993, 0...	657.00	626.78	590.00	683.00	677.80	640.00	610.15	306	255.0
9	25 Nov 1993, 0...	657.00	626.77	590.00	683.00	677.80	640.00	610.16	306	255.0
10	25 Nov 1993, 0...	657.00	626.76	590.00	683.00	677.80	640.00	610.16	305	255.0
11	25 Nov 1993, 1...	657.00	626.74	590.00	683.00	677.80	640.00	610.16	305	255.0
12	25 Nov 1993, 1...	657.00	626.73	590.00	683.00	677.80	640.00	610.16	304	255.0
13	25 Nov 1993, 1...	657.00	626.72	590.00	683.00	677.80	640.00	610.16	303	255.0
14	25 Nov 1993, 1...	657.00	626.71	590.00	683.00	677.80	640.00	610.16	303	255.0
15	25 Nov 1993, 1...	657.00	626.69	590.00	683.00	677.80	640.00	610.16	302	255.0
16	25 Nov 1993, 1...	657.00	626.68	590.00	683.00	677.80	640.00	610.16	301	255.0
17	25 Nov 1993, 1...	657.00	626.67	590.00	683.00	677.80	640.00	610.16	300	255.0
18	25 Nov 1993, 1...	657.00	626.65	590.00	683.00	677.80	640.00	610.16	300	255.0
19	25 Nov 1993, 1...	657.00	626.64	590.00	683.00	677.80	640.00	610.16	299	255.0
20	25 Nov 1993, 1...	657.00	626.63	590.00	683.00	677.80	640.00	610.16	298	255.0
21	25 Nov 1993, 2...	657.00	626.61	590.00	683.00	677.80	640.00	610.15	298	255.0
22	25 Nov 1993, 2...	657.00	626.60	590.00	683.00	677.80	640.00	610.15	297	255.0

Figure 16.8 Data in Tabular Form

Tip

To **sort** the data in most report tables:
Click on the column header. Each click will toggle the sort between ascending and descending order.

The **View** menu of the Tabular Data window offers six display options. The **Commas** option displays commas in numbers greater than one thousand. The **Reverse Order** shows the table starting with the last time period and ending with the first time period. The **Date and Time Separately** option splits the date and time into two separate columns. The **Date with 4 Digit Years** option displays the year with four digits instead of the default two. You can set the decimal place for viewing the data by using the **Show Decimal Places** option. Lastly, you can indicate how you want missing data displayed by using the **Show Missing As** option. You can also resize table columns by dragging their borders to the desired position with your mouse.

16.3 Printing and Exporting Plots and Tables

You can print ResSim plots and tables, copy and paste them into other applications, and specify export options for plots. See Appendix E for details.

Chapter 17 Viewing and Managing Reports

Reports are available from every module of HEC-ResSim. Many of the reports are module-specific, but some things remain the same no matter which module you are in—things like opening a report, selecting view settings, printing, and exporting. In addition, ResSim has a Report Builder for creating your own reports based on simulation results. This chapter covers the basics for viewing and managing reports and describes different types of reports available in the Simulation Module.

17.1 Viewing Summary Reports

Summary Reports are available from the **Reports** menu. In all Summary Reports, the Simulation name and Alternative appear at the top of the window, along with the Lookback date and time, the Start date and time, and the End date and time.

Summary Reports also have two menus. The **File** menu allows you to **Print** and **Close** the report (see Appendix F for information about printing). The **Options** menu lets you **Specify the Time** to review the simulation results for a single time period.

17.1.1 Reservoir Summary Reports

The **Reservoir Summary Report** (Figure 17.1) displays Average, Maximum, and Minimum result values for pertinent reservoir parameters.

To view the Reservoir Summary Report, select **Reservoir Summary** from the **Reports** menu.

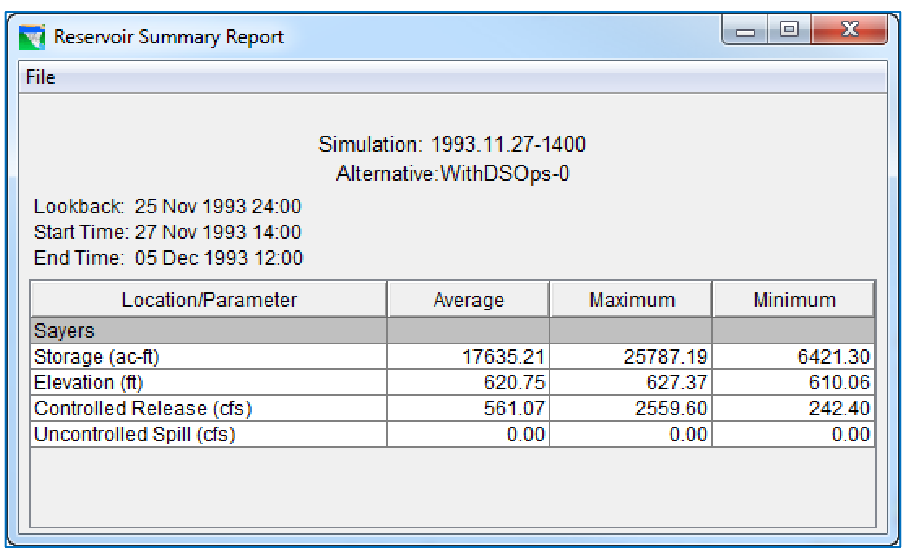
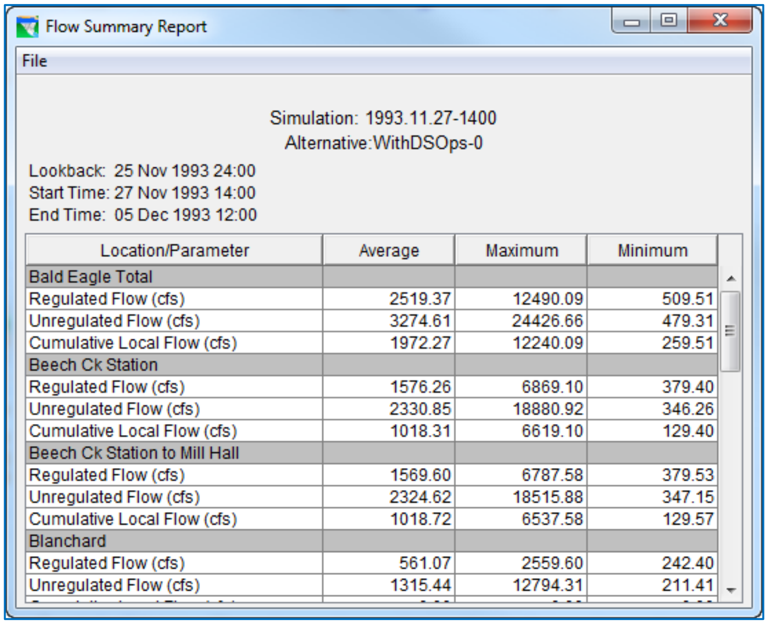


Figure 17.1 Reservoir Summary Report

17.1.2 Flow Summary Reports

The **Flow Summary Report** (Figure 17.2) displays Average, Maximum, and Minimum flow values for individual location parameters.

To view the Flow Summary Report, select **Flow Summary** from the **Reports** menu.



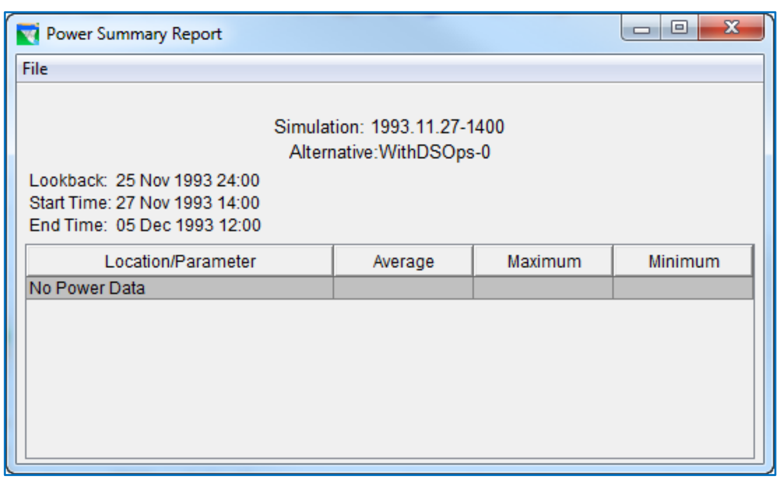
Location/Parameter	Average	Maximum	Minimum
Bald Eagle Total			
Regulated Flow (cfs)	2519.37	12490.09	509.51
Unregulated Flow (cfs)	3274.61	24426.66	479.31
Cumulative Local Flow (cfs)	1972.27	12240.09	259.51
Beech Ck Station			
Regulated Flow (cfs)	1576.26	6869.10	379.40
Unregulated Flow (cfs)	2330.85	18880.92	346.26
Cumulative Local Flow (cfs)	1018.31	6619.10	129.40
Beech Ck Station to Mill Hall			
Regulated Flow (cfs)	1569.60	6787.58	379.53
Unregulated Flow (cfs)	2324.62	18515.88	347.15
Cumulative Local Flow (cfs)	1018.72	6537.58	129.57
Blanchard			
Regulated Flow (cfs)	561.07	2559.60	242.40
Unregulated Flow (cfs)	1315.44	12794.31	211.41

Figure 17.2 Flow Summary Report

17.1.3 Power Summary Reports

The **Power Summary Report** (Figure 17.3) displays Average, Maximum, and Minimum values for individual location parameters.

To view the Power Summary Report, select **Power Summary** from the **Reports** menu.



Location/Parameter	Average	Maximum	Minimum
No Power Data			

Figure 17.3 Power Summary Report

17.1.4 Gate Summary Reports

The **Gate Summary Report** (Figure 17.4) displays Average, Maximum, and Minimum values for individual location parameters.

To view the Gate Summary Report, select **Gate Summary** from the **Reports** menu.

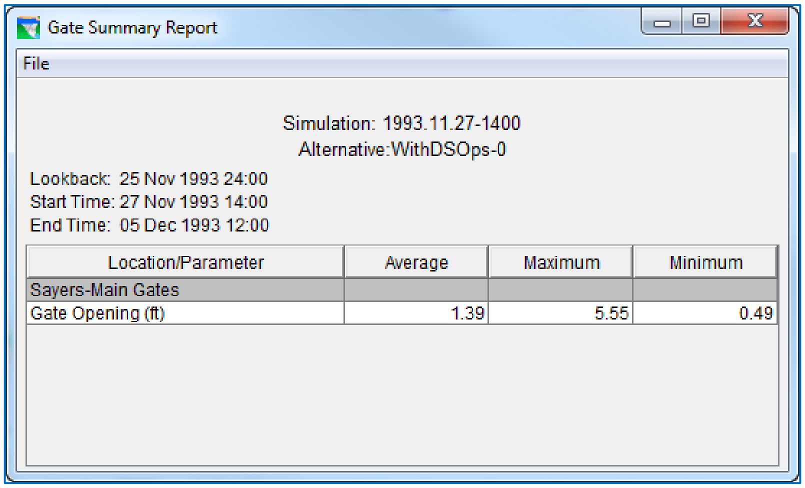


Figure 17.4 Gate Summary Report

17.1.5 Stage Summary Reports

The **Stage Summary Report** (Figure 17.5) displays Average, Maximum, and Minimum stage values for individual location parameters.

To view the Stage Summary Report, select **Stage Summary** from the **Reports** menu.

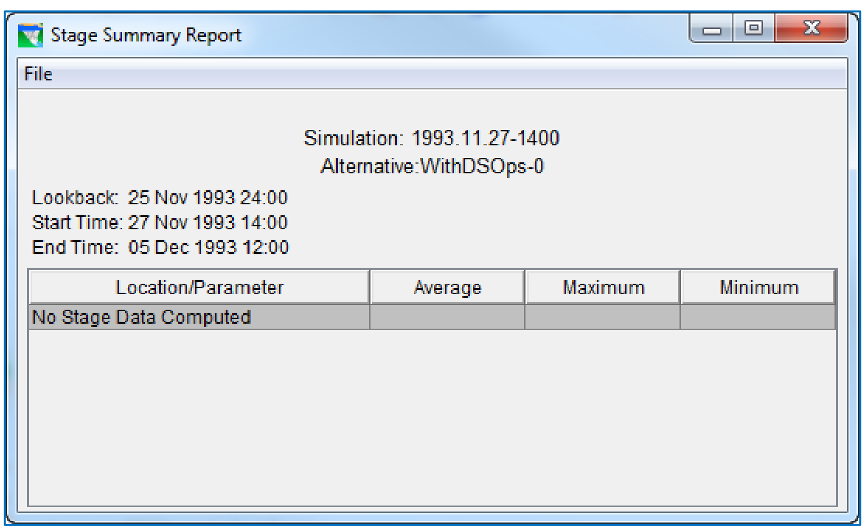
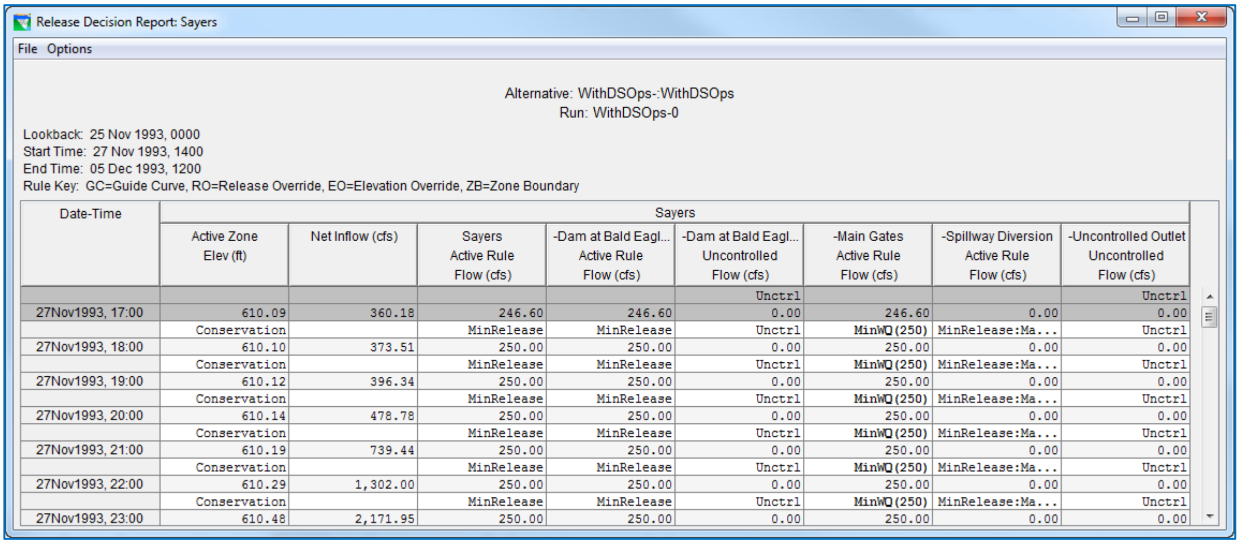


Figure 17.5 Stage Summary Report

17.1.6 Release Decision Reports

The **Release Decision Report** (Figure 17.6) displays a reservoir's *Active Zone*, *Elevation*, and *Net Inflow* for each timestep of the simulation. It also gives the *Active Rule* and *calculated release* for the reservoir pool, dam, and outlets for each timestep.



Date-Time	Sayers							
	Active Zone Elev (ft)	Net Inflow (cfs)	Sayers Active Rule Flow (cfs)	-Dam at Bald Eagl... Active Rule Flow (cfs)	-Dam at Bald Eagl... Uncontrolled Flow (cfs)	-Main Gates Active Rule Flow (cfs)	-Spillway Diversion Active Rule Flow (cfs)	-Uncontrolled Outlet Uncontrolled Flow (cfs)
					Unctrl			Unctrl
27Nov1993, 17:00	610.09	360.18	246.60	246.60	0.00	246.60	0.00	0.00
	Conservation		MinRelease	MinRelease	Unctrl	MinWQ(250)	MinRelease:Ma...	Unctrl
27Nov1993, 18:00	610.10	373.51	250.00	250.00	0.00	250.00	0.00	0.00
	Conservation		MinRelease	MinRelease	Unctrl	MinWQ(250)	MinRelease:Ma...	Unctrl
27Nov1993, 19:00	610.12	396.34	250.00	250.00	0.00	250.00	0.00	0.00
	Conservation		MinRelease	MinRelease	Unctrl	MinWQ(250)	MinRelease:Ma...	Unctrl
27Nov1993, 20:00	610.14	478.78	250.00	250.00	0.00	250.00	0.00	0.00
	Conservation		MinRelease	MinRelease	Unctrl	MinWQ(250)	MinRelease:Ma...	Unctrl
27Nov1993, 21:00	610.19	739.44	250.00	250.00	0.00	250.00	0.00	0.00
	Conservation		MinRelease	MinRelease	Unctrl	MinWQ(250)	MinRelease:Ma...	Unctrl
27Nov1993, 22:00	610.29	1,302.00	250.00	250.00	0.00	250.00	0.00	0.00
	Conservation		MinRelease	MinRelease	Unctrl	MinWQ(250)	MinRelease:Ma...	Unctrl
27Nov1993, 23:00	610.48	2,171.95	250.00	250.00	0.00	250.00	0.00	0.00

Figure 17.6 Release Decision Report

You can access the Release Decision Report by selecting **Release Decision** from the **Reports** menu. To go straight to a specific reservoir's Release Decision Report, select **Release Decision Report** from the reservoir's context menu (right-click the reservoir element).

The Release Decision Report **File** menu allows you to **Print** and **Close** the report.

The **Options** menu (Figure 17.7) allows you to select a different reservoir for the report. It also gives you the option to select a time window—either the default of the entire simulation period, or a specific, shorter time window.

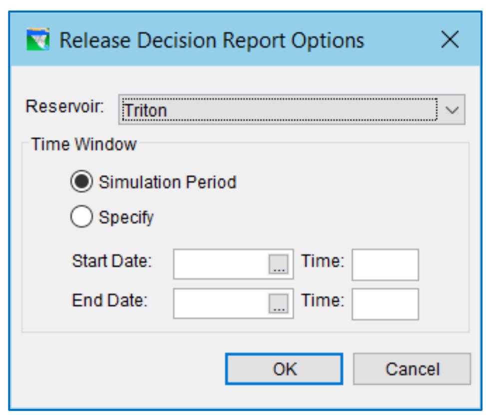


Figure 17.7 Release Decision Report Options

17.1.7 User Reports

The User Report editor provides you with the ability to create and edit user reports. **User Reports** make it possible to organize, manipulate, and display simulation results in customized tables. Output time series data specific to the current **active alternative** in the Simulation module is made available to the User Report editor.

To create, edit, or access **User Reports**:

- From the **Reports** menu, select **User Reports → Edit...** (Figure 17.8). The **User Report Editor** will appear, as shown in Figure 17.9.

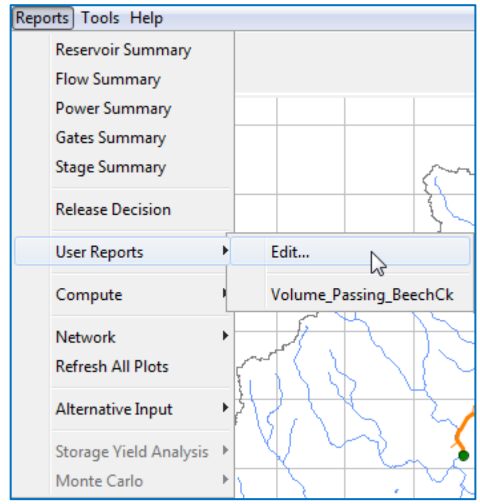


Figure 17.8 Simulation Module—Reports Menu—Accessing the User Report Editor

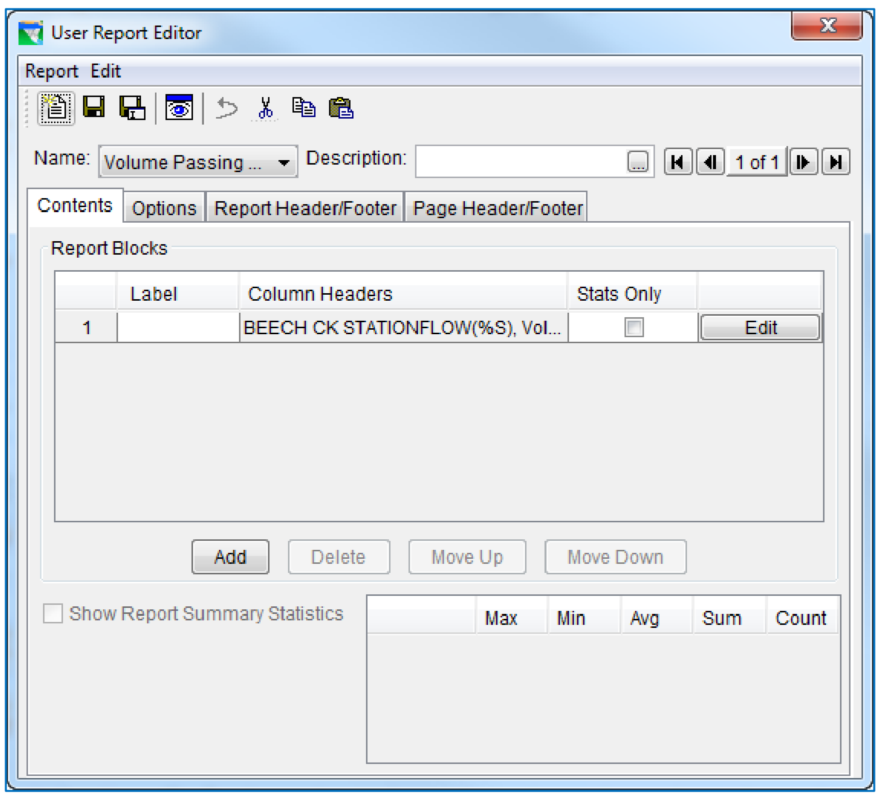


Figure 17.9 User Report Editor

- Create a new report template by selecting **New** from the **Report** menu. The **Create a New Report Template** tool (located in the upper left-hand corner of the editor) can also be used for creating a new report. The **Create New Report Template** dialog will appear (Figure 17.10) where you will give your report a **Name** and **Description** and select **OK**.

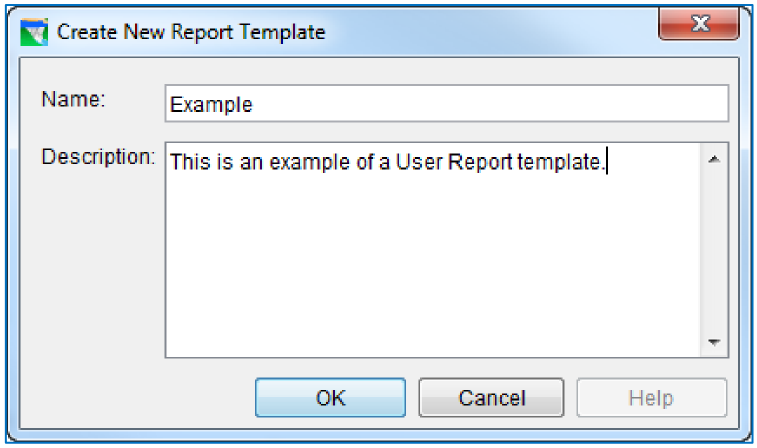


Figure 17.10 Create a New Report Template

- After creating a new report template (Figure 17.11), you will notice four tabs in the **User Report Editor**: **Contents, Options, Report Header/Footer,** and **Page Header/Footer.**

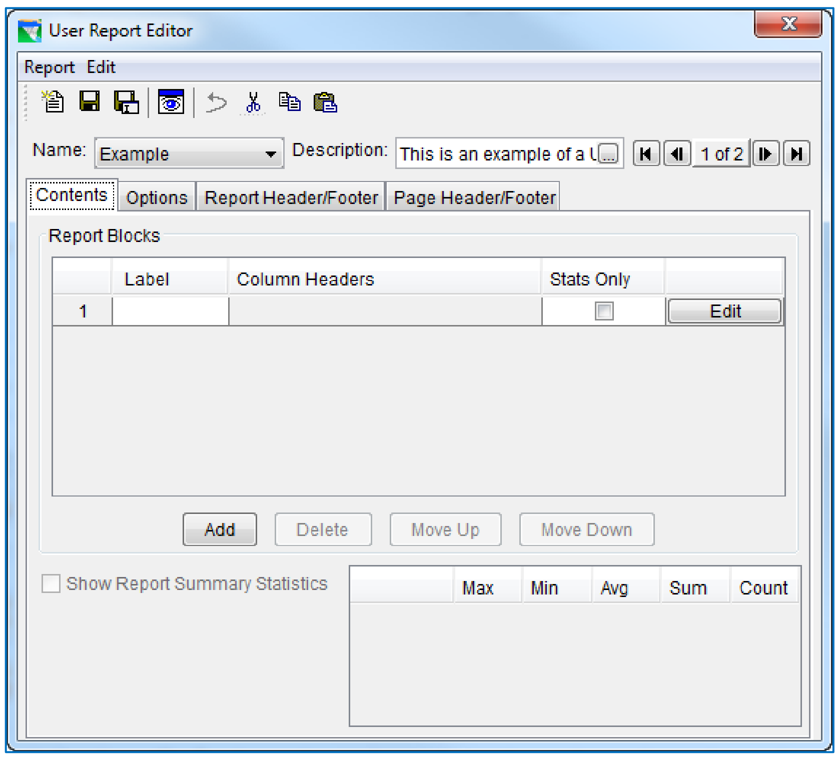


Figure 17.11 User Report Editor—After Creating a New Report Template

- From the **Contents** tab select **Edit** to choose the output time series that you would like to include in your first report block. The **Report Content Selection** dialog will appear (Figure 17.12).

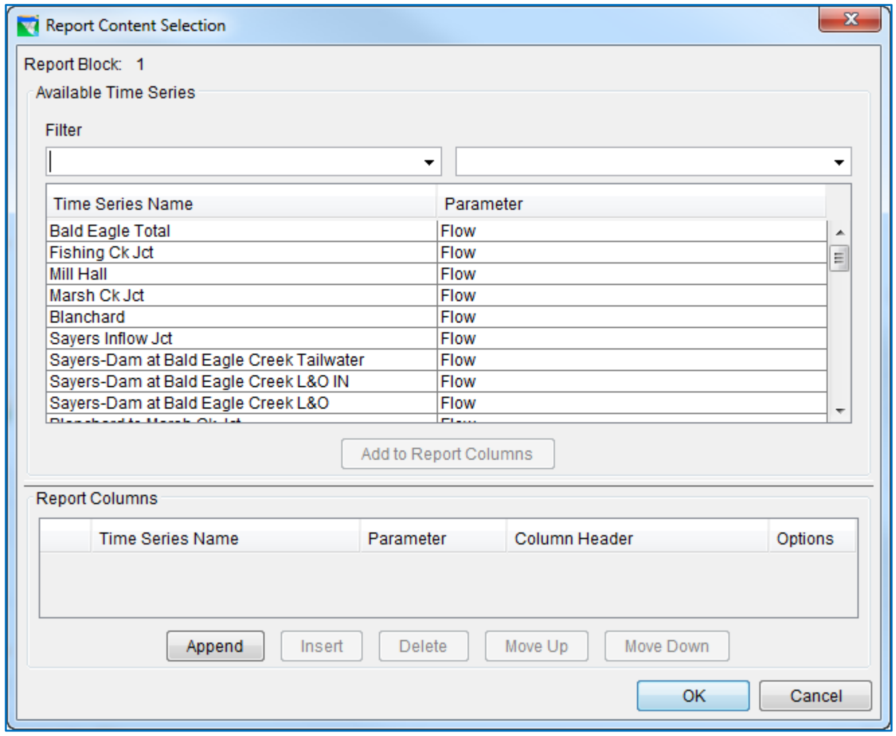


Figure 17.12 Report Content Selection

- To add a time series, highlight the **Time Series Name** and select **Add to Report Columns.** Each time series that is added will be represented as its own column in the current report block. As mentioned previously, only output time series data from the current **active alternative** in the Simulation module is available for report building.

Figure 17.13 is a display of the **Report Content Selection** dialog after two time series have been added to the report block under construction.

All elements in the **Report Columns** area of the **Report Content Selection** dialog are editable. To edit **Time Series Name**, **Parameter**, or **Column Header**, double click on the desired cell and make the appropriate changes.

Units for column headings can either be entered in manually or using the character string code **%S**. Typing this into the column header box (as shown in Figure 17.13) will result in the units automatically being displayed in the report. *Note that the code is case sensitive.*

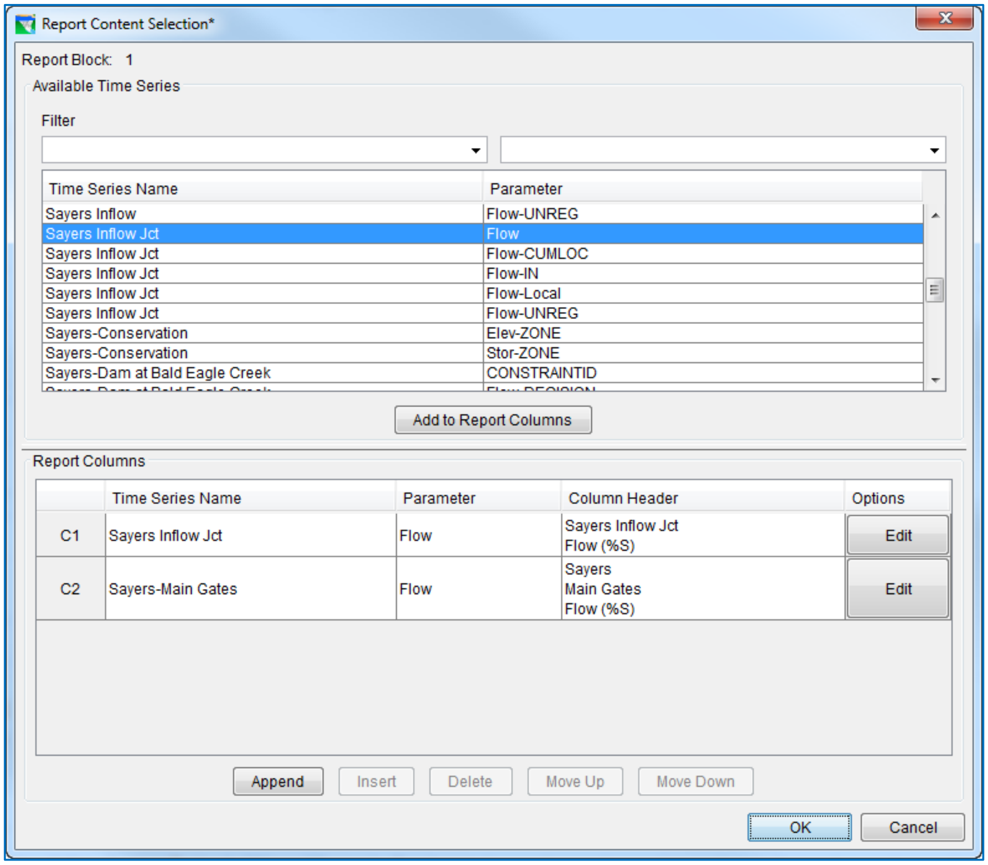


Figure 17.13 Report Content Selection—Time Series Added to Report Columns

The buttons at the bottom of the **Report Columns** area, **Append**, **Insert**, **Delete**, **Move Up**, and **Move Down**, can be used to add columns to the report and rearrange time series data into different columns. **Append** will add a blank column to the far-right side of the report (a row will be added to the template). **Insert** will add a blank column to the left of the column in the report that is selected in the **Report Columns** area (a row will be added before the selected row in the template). **Delete** will erase from the report whatever column is selected in the **Report Columns** area (the selected row in the template will be deleted). **Move Up** and **Move Down** will shift the selected column to the left and right in the report, respectively (the row in the template will shift up or down).

Select the **Edit** option to manipulate the time series data in each column. Doing so will bring up the **Report Column Options** dialog (Figure 17.14). The title bar at the top of the dialog will include the column currently being edited (in this case, **C1**). The Report Column reflects the contents of the Column Header (in this case, **SAYERS INFLOW JCT FLOW (%S)**).

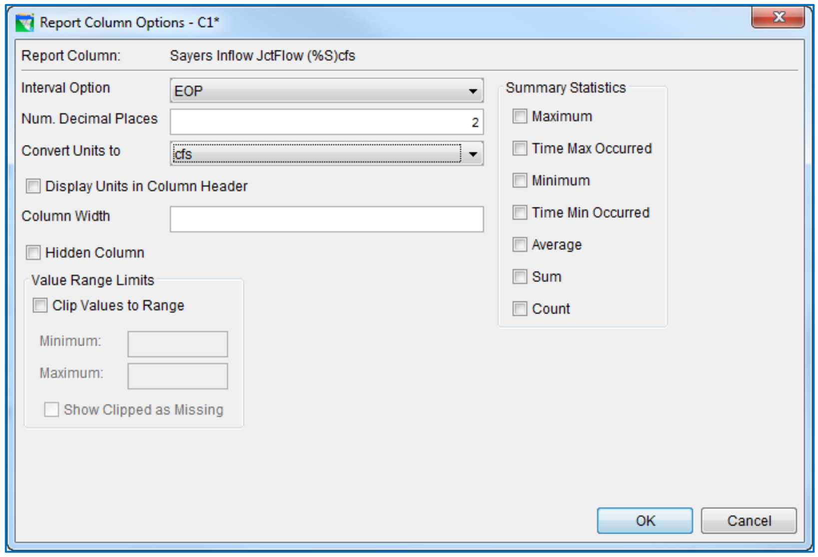


Figure 17.14 Report Column Options

For **Interval Option**, you are given four choices as to how you would like your data to be defined for each point in time: **SOP** (Start of Period), **EOP** (End of Period), **PAVG** (Period Average), and **PCUM** (Period Cumulative). This gives you the ability to specify what type of data is actually being reported for each time interval.

The number of decimal places reported for the time series can also be specified with **Num. Decimal Places**.

Units Multiplier (prefix) can be used to select a multiplier (1, 1000, or 1000000) that will be applied as a "prefix" to the units of the data reported in the column being reported. For example, you might choose 1000 to convert KW units to MW units. The data values will be divided by 1000 and the units label will be preceded by 1000 (e.g., 1000 KW). For example, for a value of 2526 KW, then the value displayed in the report column will change to 2.526 and the Units should reflect 1000 KW. Currently, however, the multiplier value is not automatically displayed in the column header. Therefore, you must manually enter the multiplier into column headings wherever appropriate. *Note: it also might be appropriate to increase the number of Decimal Places (see previous paragraph) when using this option for displaying units.*

The **Column Width** can also be specified. To manually enter a desired width, enter the number of characters wide that you would like your column to be in the **Column Width** box.

Checking the **Hidden Column** box will effectively prevent the column from being visible in the report. Selecting this option only prevents the column from being seen. The time series data, and all edits to the column, will still be attached to the report.

It is possible to withhold data that does not fall within a certain specified range by checking the **Clip Values to Range** box. After checking the box, specify the range (outside of which the data will not be shown) by entering the **Minimum** and **Maximum** values. Furthermore, you can check the **Show Clipped as Missing** box, which will replace the clipped data with either "M", "-M-", or a blank space. See options from the **View** menu of the **User Report** as subsequently discussed in Step 8.

The right side of the **Report Column Options** dialog contains **Summary Statistics** that can be calculated for the time series data in the current column being edited. These statistics include the following: **Maximum**, **Time Max Occurred**, **Minimum**, **Time Min Occurred**, **Average**, **Sum**, and **Count**. The statistics that are chosen to be calculated will be shown in the report at the bottom of the column.

Once all edits have been made to the column, select **OK** to save and return to the **Report Content Selection** dialog (previously shown in Figure 17.12), or select **Cancel**, which will return you to the **Report Content Selection** dialog and ignore all column edits you just made. In the **Report Content Selection** dialog, select **OK** to save all edits and return to the **User Report Editor** (previously shown in Figure 17.11). Selecting Cancel will return you to the **User Report Editor** but will not save any edits you made to the report block.

In the **User Report Editor**, select the **Options** tab (as shown in Figure 17.14).

By using the dropdown menu for **Reporting Interval**, you can select which report interval you would like to use. You can select **Each Step**, **Daily**, **Weekly**, **Monthly**, or **Yearly**. When choosing **Daily**, **Weekly**, **Monthly**, or **Yearly**, it will be necessary to specify what hour, day, or month you would like to report. For instance, if a daily reporting interval is used, you will need to choose which hour of the day you would like reported. When **Each Step** is chosen, data will be reported for all intervals, so it is not necessary to specify reporting hours, days, or months.

Default Minimum Column Width can also be specified. In the box to the right, column width, expressed in pixels, can be set to whatever value you find appropriate. The default column width is set to 100 pixels. You can increase the column width, but the minimum width will be at least 100 pixels (even if you enter a value less than 100).

In the bottom left corner are the **Date/Time Column** options which can be used to specify how you would like time to be reported. Choosing **Start of Period** will display the time at the start of each interval that data is reported. Choosing **End of Period** will display the time at the end of each interval, and choosing **Both** will display the beginning time and ending time of each interval. You can also check **Show Time**, which will display the hour of day for each time interval, and **Show Day of Week**, which will display the day of week for each time interval.

In the bottom right corner are the **Report Size and Location** options that can be used to specify how large you would like your report to be and where on the *screen* you would like it to appear. The size of your report can be set by entering dimensions (in pixels) in the **Report Size (w×h)** boxes. Furthermore, you can set the location where the report will appear on your computer screen by entering coordinates (in pixels) in the **Report Screen Location (x,y)** boxes. These screen coordinates should correspond to an origin based on the upper left hand corner of your computer screen (i.e., an "x, y" coordinate of "1, 1" would place the upper left corner of your User Report in the upper left corner of your computer screen). Since this display control affects the report both when it is displayed on the screen and when it is printed, you should only use this option when you are not printing the report.

- Select the **Report Header/Footer** tab in the **User Report Editor** to give your report a title, header, and footer, if desired (Figure 17.15).

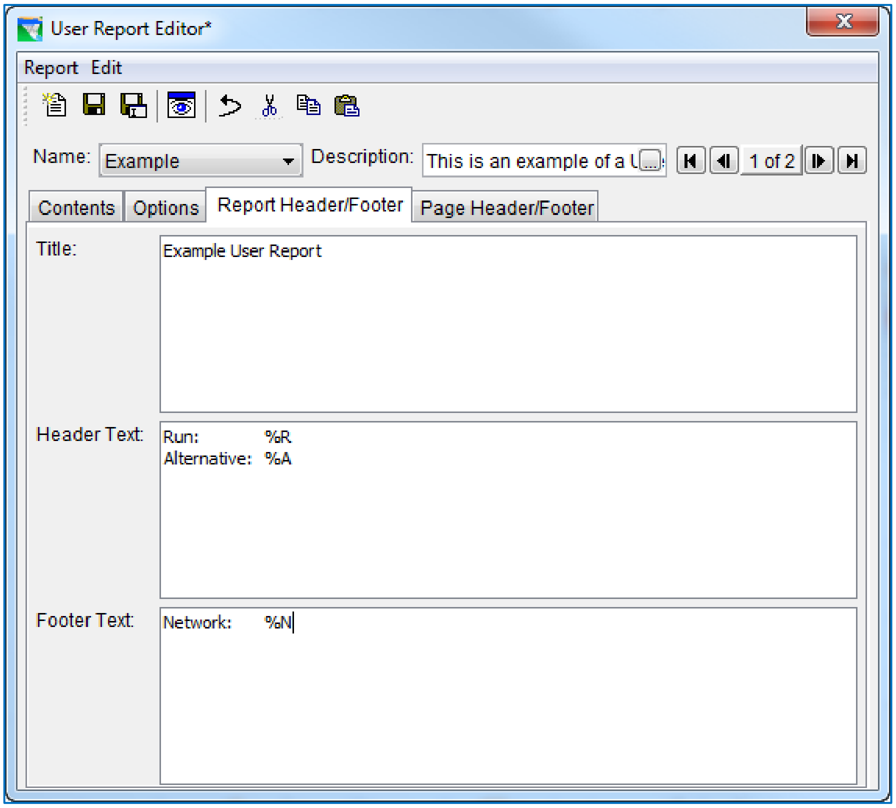


Figure 17.15 User Report Editor—Report Header/Footer Tab

The **Title** and **Header Text** will appear at the beginning of the report, while the **Footer Text** will appear at the end. Notice the character string codes (**%**) entered in Figure 17.15. These **codes** refer to run name (**%R**), alternative name (**%A**), and network name (**%N**), respectively.

Table 17.1 provides a list of the ***character string codes*** (which are *case sensitive*) that can be used to allow automatic viewing of special text fields in the report where text can be added or edited and in the Report Header & Footer.

Table 17.1 User Reports—Character String Codes

%R	Run name
%A	Alternative name
%N	Network name
%L	Lookback time
%B	Simulation Beginning (or start) time
%E	Simulation End time
%V	Build Version
%S	Units

- Select the **Page Header/Footer** tab in the **User Report Editor** to add a header and footer to the page of your report, if desired (Figure 17.16).

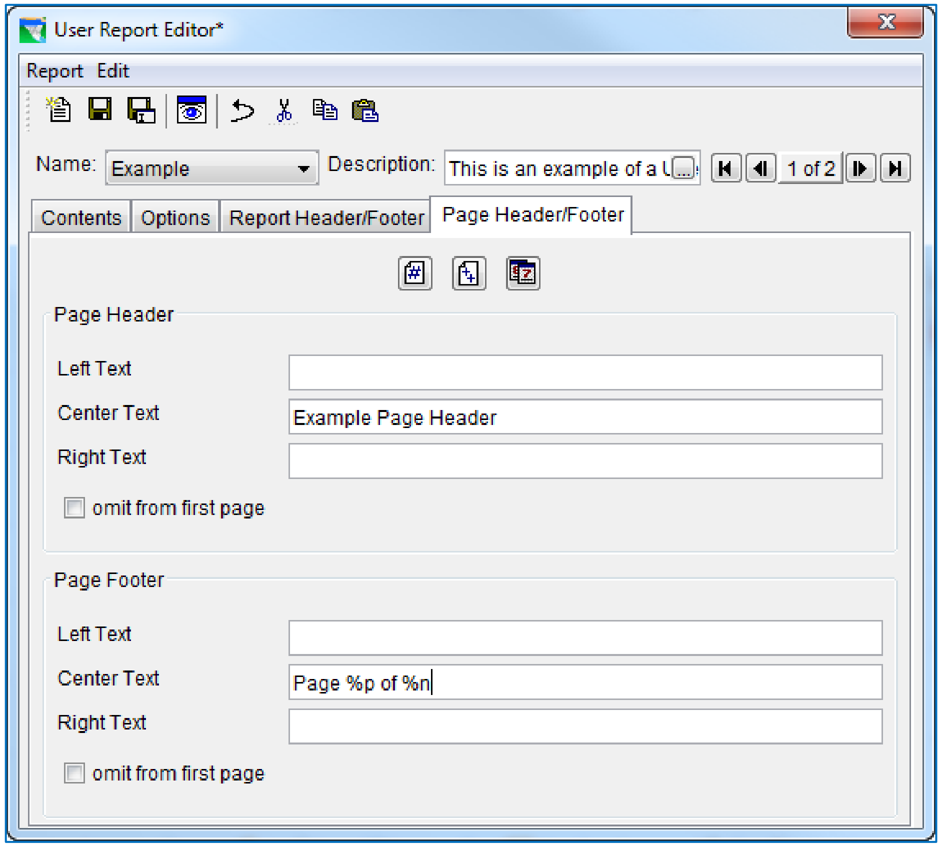


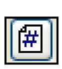
Figure 17.16 User Report Editor—Page Header/Footer Tab

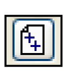
Instead of inserting a header and footer at the beginning and end of the report, a header and footer will be inserted at the top and bottom of each printed page. Table 17.2 provides a list of the ***character string codes*** (which are *case sensitive*) that can be used to allow automatic viewing of special text fields in the page header and/or footer.

Table 17.2 User Reports—Character String Codes for Header/Footer

%d	Date report is printed
%p	Page number
%n	Number of pages

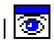
The three tool buttons located above the Page Header options can be pressed to insert the following codes (instead of manually typing in the %code):

 Add the page number (%p)

 Add the total pages (%n)

 Add the current date (%d)

If you do not want the Page Header and/or Footer information to appear on the first page of your report, you can check the box labeled **omit from first page**.

- In the **User Report Editor**, use the **Save and View Report** tool  to save your report template to disk and see a preview of your User Report.

A **User Report** contains the following menus:

- **Report**: Options include **Save As…**, **Print**, and **Close** (as shown in Figure 17.17).

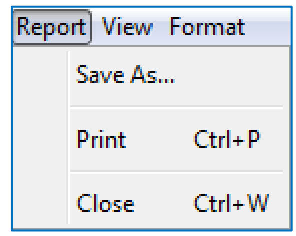


Figure 17.17 User Report—Report Menu

- **View**: Options include **Show Missing As** (blank, M, -M-) and **Show Commas** (as shown in Figure 17.18).

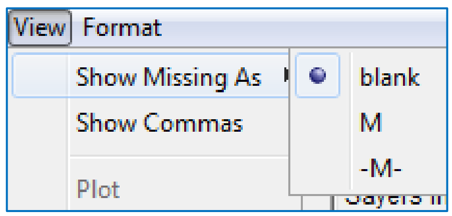


Figure 17.18 User Report—View Menu

- **Format**: Options include **Font** (for Title, Header, Footer, and Tables) and **Save Settings** (as shown in Figure 17.19).

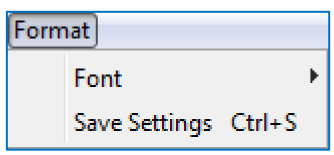
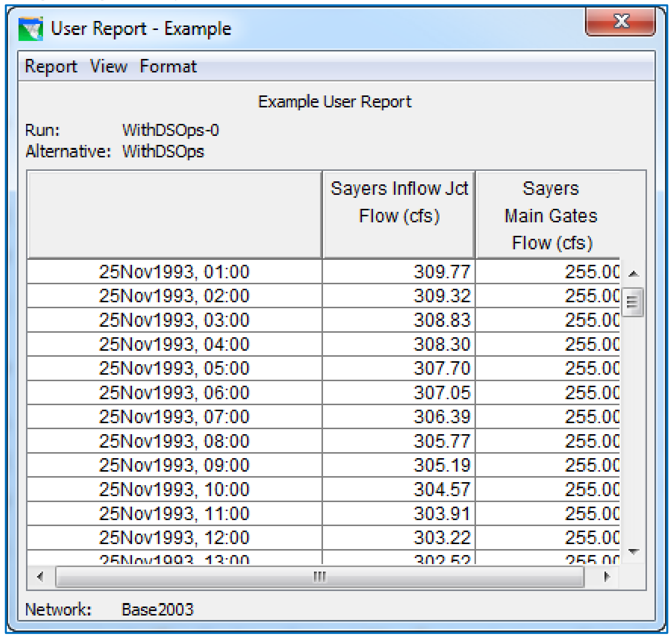


Figure 17.19 User Report—Format Menu

Figure 17.20 shows the "Beginning" (top of figure) and "End" (bottom of figure) of the **Example User Report** *(note the report title, header, and footer as well as the statistics printed at the end of each column)*.

"Beginning" of Report Block:



"End" of Report Block:

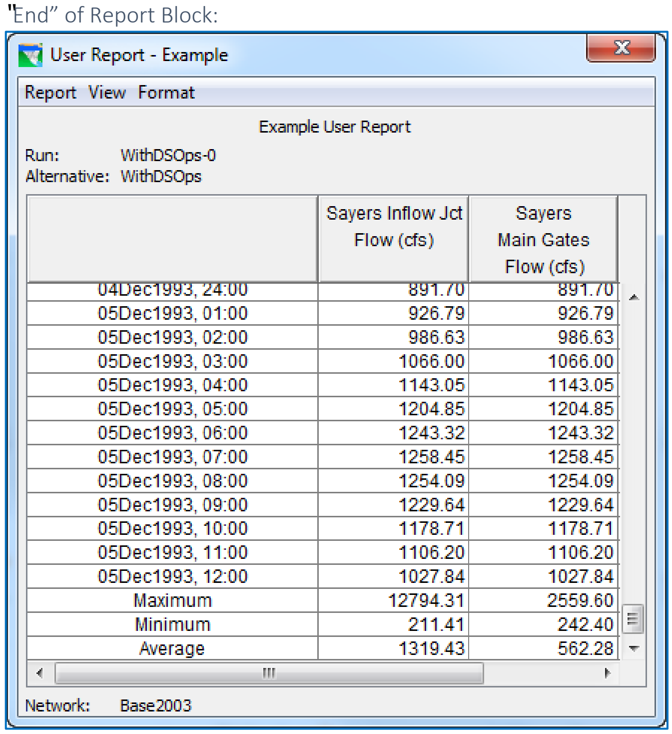


Figure 17.20 User Report Preview

- To add *another report block*, return to the **User Report Editor**. Under the **Contents** tab, select **Add** to insert another report block. Add one or more time series to the new report block using methods previously discussed and make desired edits to the newly created columns. Each report block can be given a title in the **Label** column in the **Report Blocks** area of the **User Report Editor**. Figure 17.21 shows the **User Report Editor** with two report blocks. In this example, the two labels Block 1 and Block 2 were entered for illustration purposes.

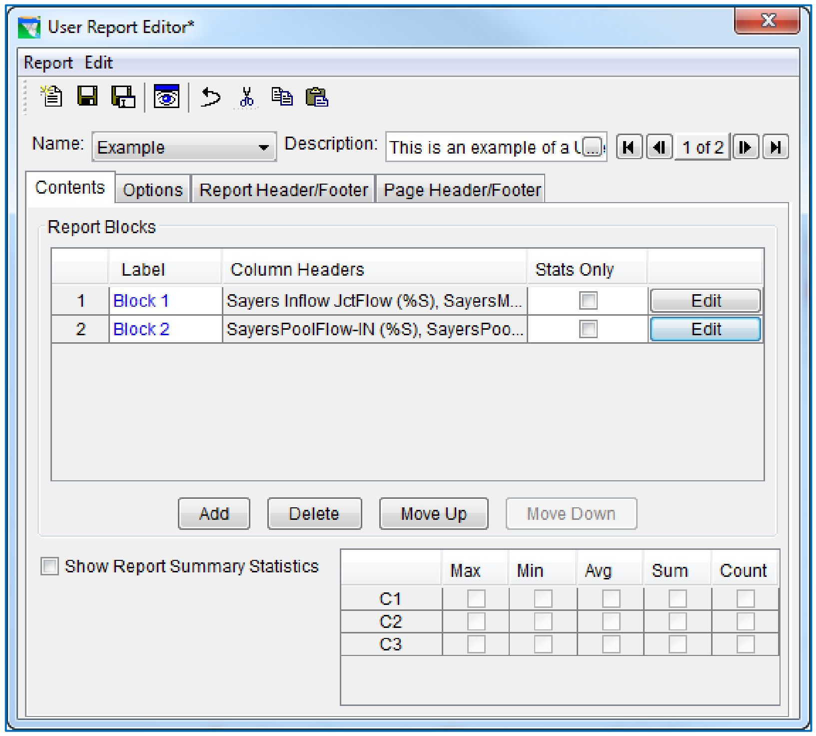
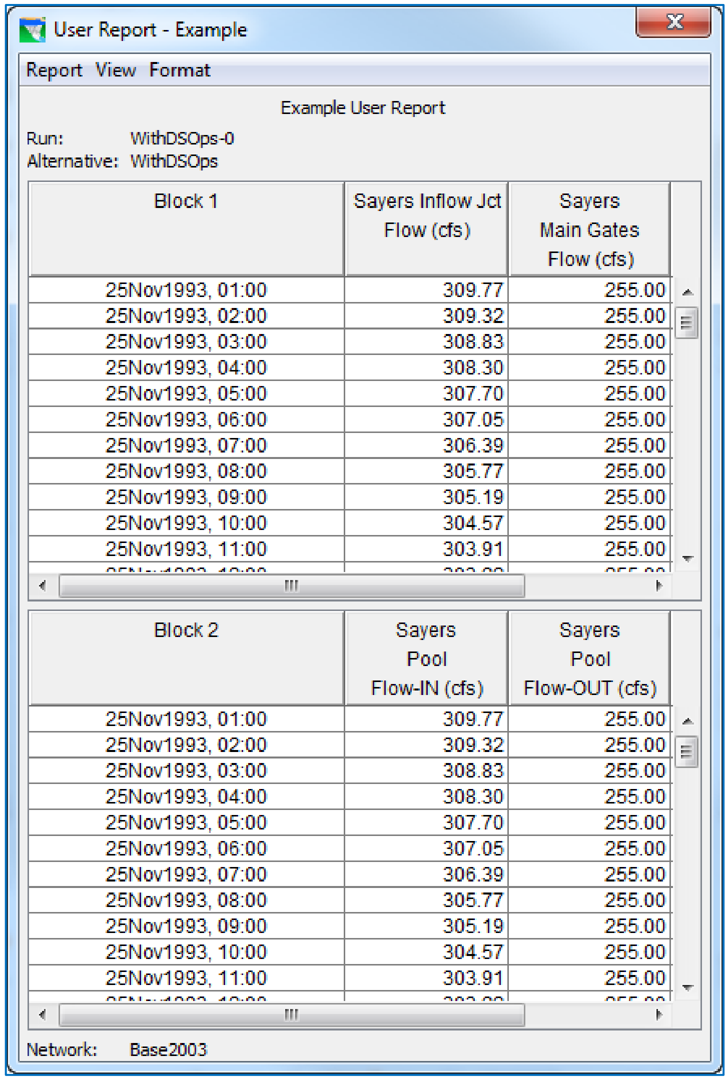


Figure 17.21 User Report Editor—Contents Tab—User Report Template with Two Report Blocks

Figure 17.22 shows a preview of the User Report after adding the second report block. Note that the labels assigned in the **User Report Editor** appear above the time column.



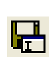
Block 1	Sayers Inflow Jct Flow (cfs)	Sayers Main Gates Flow (cfs)
25Nov1993, 01:00	309.77	255.00
25Nov1993, 02:00	309.32	255.00
25Nov1993, 03:00	308.83	255.00
25Nov1993, 04:00	308.30	255.00
25Nov1993, 05:00	307.70	255.00
25Nov1993, 06:00	307.05	255.00
25Nov1993, 07:00	306.39	255.00
25Nov1993, 08:00	305.77	255.00
25Nov1993, 09:00	305.19	255.00
25Nov1993, 10:00	304.57	255.00
25Nov1993, 11:00	303.91	255.00

Block 2	Sayers Pool Flow-IN (cfs)	Sayers Pool Flow-OUT (cfs)
25Nov1993, 01:00	309.77	255.00
25Nov1993, 02:00	309.32	255.00
25Nov1993, 03:00	308.83	255.00
25Nov1993, 04:00	308.30	255.00
25Nov1993, 05:00	307.70	255.00
25Nov1993, 06:00	307.05	255.00
25Nov1993, 07:00	306.39	255.00
25Nov1993, 08:00	305.77	255.00
25Nov1993, 09:00	305.19	255.00
25Nov1993, 10:00	304.57	255.00
25Nov1993, 11:00	303.91	255.00

Figure 17.22 User Report with Two Report Blocks

- Be sure to save your user report template after you have made all necessary edits. If you have not used the  tool after making your final edits, then be sure to save your User Report template using one of the following options:

You can use the tools available in the toolbar located near the top of the **User Report Editor:**

-  Save the Current Report Template using the current name.
-  **Save the Current Report Template under a new name** which will bring up a separate dialog that allows you to specify a new Name and Description for your report template.

You can also choose to use the **Save** commands available from the **Report** menu of the **User Report Editor**, as shown in Figure 17.23.

- Completed and saved **User Reports** can be directly accessed in the Simulation module from the **Reports** menu. Select **Reports** → **User Reports** to view a list of current user reports (Figure 17.24). In this example, the new user report created for this watershed is named "My_User_Report." Selecting "My_User_Report" will bring up the **User Report** previously shown in Figure 17.20.

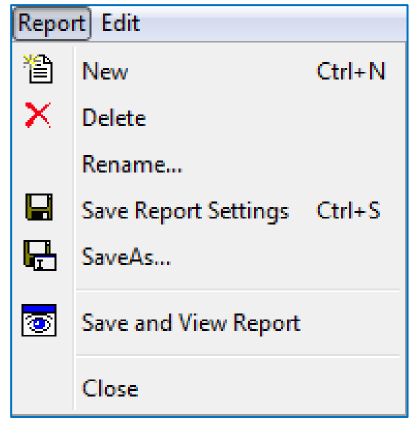


Figure 17.23 User Report Editor—Report Menu

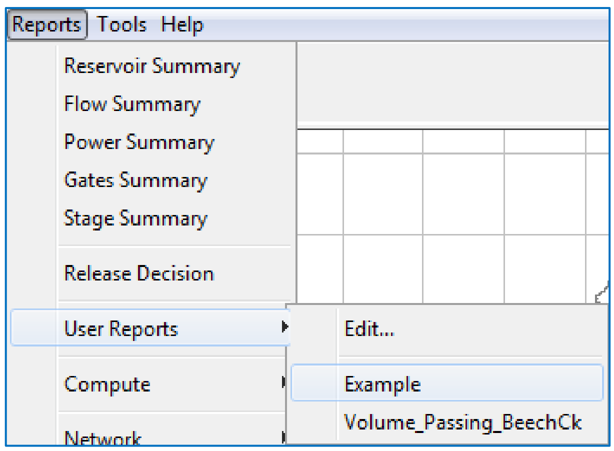


Figure 17.24 Simulation Module Reports Menu—Accessing Saved User Reports

17.1.8 Network Reports

The **Reservoir List**, **Reach List**, **Junction List**, **Diversion List**, and the **Network Connectivity Summary** report are also available from the **Reports** menu → **Network** option of the Simulation Module. Refer to Section 6.5 for more information about Network Reports.

17.2 Printing and Exporting Reports

From each individual report (or plot) there is a context menu that allows you to print the report, preview the report before you print, and export the report to an ASCII file. Right-clicking on the active report accesses the context menu. You can also print or export a report by selecting **Print** or **Export** from the report's **Report** (or **File**) menu.

17.2.1 Printing Reports

In one way or another, all ResSim reports are tables of information about your model. Hence, when you select **Print** from a report's context menu (Figure 17.25) or **Report** (or **File**) menu, the printed output will be a titled table containing the data from the active report. The printed report and its columns are influenced by the width of the report window and the reports columns, so if items in the columns of the report are not printing as you would like, you can resize the report window and/or the individual columns to adjust the printed report.

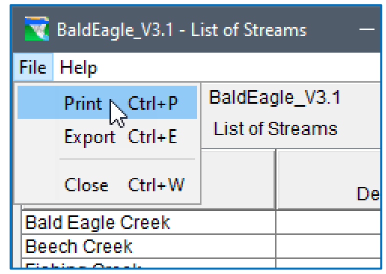


Figure 17.25 Report Print Option

To resize the report columns or the report window:

- Position the mouse pointer in the table header over one of the lines that separate the columns or over the window border.
- When the resize arrows  appear, hold down the left mouse button and drag the mouse right or left to indicate how wide or narrow you want the columns or dialog.

To print a report from the Watershed Setup module:

- Open a report. For example, from the **Reports** menu in the Watershed Setup module, select List of Streams (Figure 17.26).
- From the Report or File menu of the opened report, select Print.
- Or, right-click anywhere within the tabulated data of the report and select Print from the context menu (Figure 17.27).
- The Print Properties dialog will open (Figure 17.28).
- From the Page tab of the Print Properties dialog, you can:
    - Set the **Orientation** of your printed report—**Portrait** (default) or **Landscape**
    - Make the report columns fit on one page
    - Print the **Entire Table** (default) or just print the Selected Cells,
    - **Repeat** (Column) **Headers** on subsequent pages,
    - **Display Gridlines** (draw borders around each cell of the table).

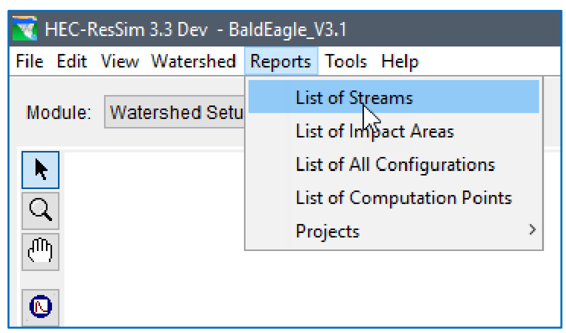


Figure 17.26 Selecting a Report

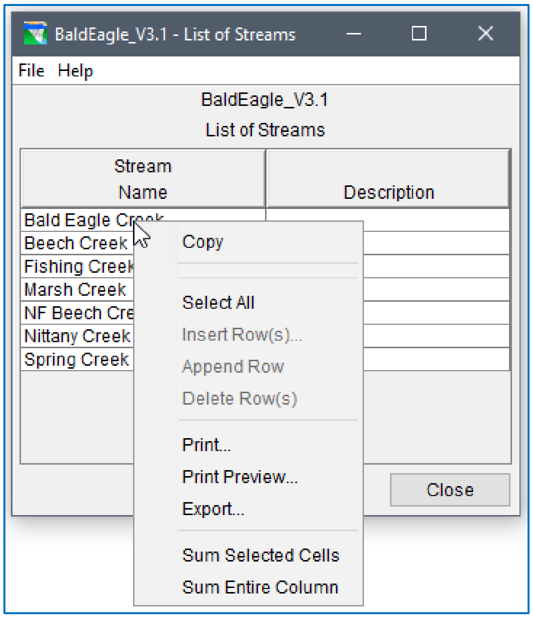


Figure 17.27 A Report's Context Menu

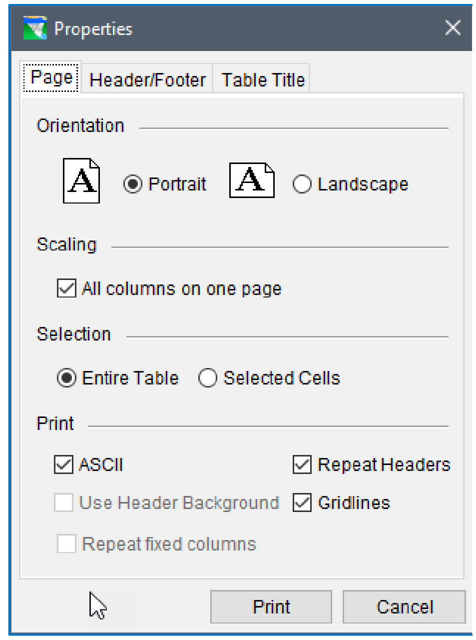


Figure 17.28 Print Properties Dialog

- By default, the report prints in **ASCII** format with a white background. If you would like the column headers to have a light grey background, uncheck ASCII and check **Use Header Background**. If header rows are included among the data in the table, those rows will also get a grey background, a little darker than the column headers.
- The **Header/Footer** tab contains two text boxes where you can enter text for a header and/or footer for the report. The header will be centered just below the top margin (and just above the report title). The footer be centered just above the bottom margin and will be separated from the table data by a blank line. The header and footer will appear on all pages of the report. When a header and footer are included, together they occupy approximately 3 lines of space on each page, thus reducing the number of lines of table data that will appear per page.
- The **Table Title** tab contains a text box with the default title shown. You can use this text box to change the title of the report. Also on the Table Title tab is a check box to allow you to choose whether to repeat the report title on subsequent pages.
- Once you have your print properties set, click **Print** and the system **Print** dialog will open (Figure 17.29). Select a printer then click **OK**. The report will be printed on the selected printer.

Tip

To save paper, use "Microsoft Print to PDF" to test your print settings before sending your report to a printer.

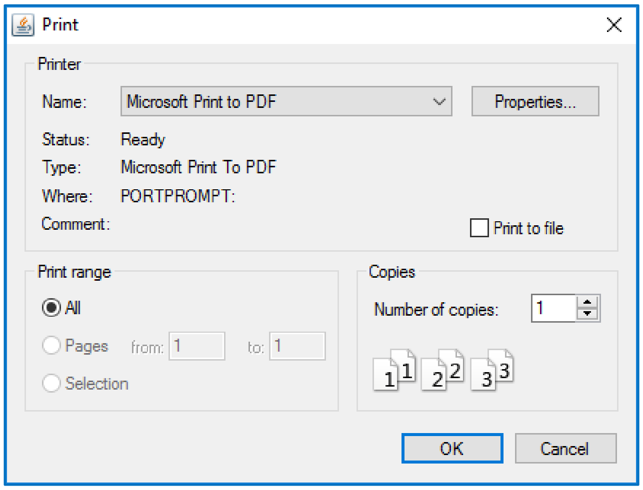


Figure 17.29 System Print Dialog

17.2.2 Print Preview

You can preview the report before printing. To do so:

- Open a report by selecting one from the Reports menu.
- Right-click anywhere within the tabulated data of the report and select Print Preview from the context menu.
- The Print Properties dialog (Figure 17.30) will appear. This Properties dialog differs from the one in Figure 17.28 only by the Print button, which is now labeled Preview.
- Configure your report and print properties as described in Section 17.2.1 above, then click the Preview button.
- The Print Preview dialog (Figure 17.31) will open. If you like what you see, you can send the report to a printer by clicking the Print button at the top of the dialog or you can close the Print Preview dialog by clicking the Close button.

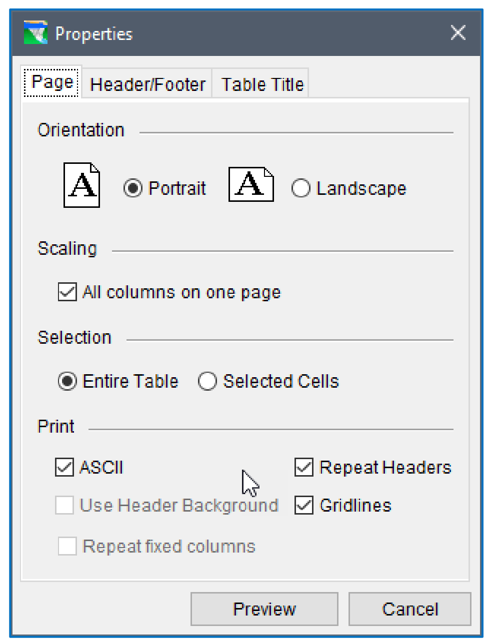
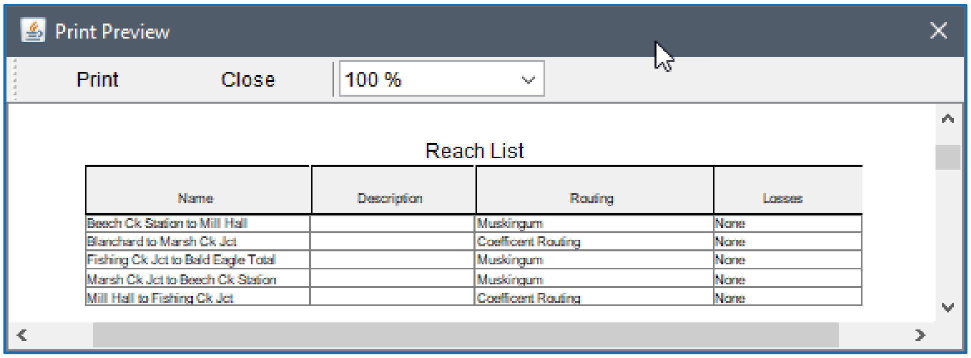


Figure 17.30 Print Preview Propeerties Dialog



Name	Description	Routing	Losses
Beech Ck Station to Mill Hall		Muskingum	None
Blanchard to Marsh Ck Jct		Coefficent Routing	None
Fishing Ck Jct to Bald Eagle Total		Muskingum	None
Marsh Ck Jct to Beech Ck Station		Muskingum	None
Mill Hall to Fishing Ck Jct		Coefficent Routing	None

Figure 17.31 Print Preview Dialog

17.2.3 Exporting Reports to a File

You can export reports to an ASCII text file. The primary reason to export the report to a file is to open it in another software product such as Microsoft Excel. To export a report:

- Right-click anywhere within the tabulated data of the report and select **Export** from the context menu. The **Table Export Options** dialog (Figure 17.32) will open. The options include:
  - **Field Delimiter**—the dropdown list includes TAB, SPACE, COMMA, COLON, and SEMI-COLON. If you plan to import the report into Excel, TAB and COMMA are the best delimiters to use.
  - **Fixed Width Columns**—although not a default setting, if you want the table to be more readable in a simple text editor, you may want to turn this option on. The cool thing about this option is that, unlike the printed report, the "fixed" width of each column in the exported report is at least the length of the longest string (value) in that column, so you will not get truncated data in your exported report.
  - **Quoted Strings**—some software products treat spaces as string delimiters, so you might need this option so that a multi-word name or description is seen as a single string. Be careful, if you select this option and fixed width string, the quoted string for each column will be padded with trailing blanks so that all the strings in a column have the same number of characters.
  - **Include Column Headers**—although the column headers are, by default, included in the report, you can turn them off if you wish.

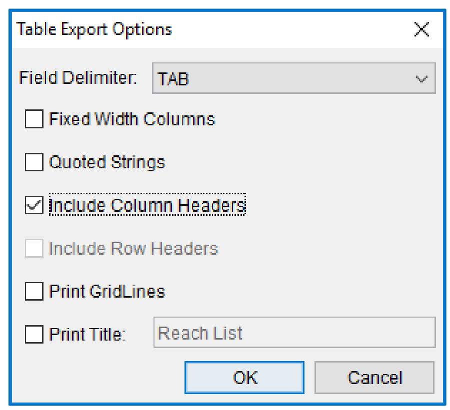


Figure 17.32 Table Export Options Dialog

  - o **Print Gridlines**—is another option you might want to consider but use with care. This turns on Fixed Width Column and then disables the Field Delimiters and turns on Fixed Column Width Options. This results in an exported report with fixed width columns of data separated by pipe characters (|) and rows of data separated by rows of dashes (-).
  - o **Print Title**—this option adds a title to the top of the exported report. If selected, a text box is available for you to enter a title.
- Once you have chosen your desired options, click OK.

The **Save** file browser will open. Use the browser to select where you want the file to be saved and enter a filename. If you want the filename to have an extension, you must enter your own; there is no default file extension. Then click **Save** to save the file and close the browser.

Chapter 18
Utility Scripting in ResSim

Scripting is the process of automating the actions of an interactive program for the purpose of being able to execute a prescribed set of actions with a simple selection. Scripting can simplify user operations and abstract complexity, ensure repeatability and consistency of results, reduce time required to generate results, and schedule an operation to occur one or more times in the future.

Scripting in ResSim takes two forms: The first form is represented by scripts that compute a value during each timestep of a simulation which will be used to influence the operation of one or more reservoirs; these are State Variables and Scripted Rules. State Variables and Scripted Rules were covered in Sections 12.2 and 12.3, respectively. The second form is represented by scripts that perform actions outside of the compute of an alternative; these scripts are often referred to as *utility* scripts because they perform such tasks as producing a custom plot of results, generating custom reports, and pre– or post–processing of output to produce additional information not directly generated by ResSim. This chapter will describe how to create, use, and manage this second form of scripting in ResSim, the utility scripts.

Both forms of scripting in ResSim use the Jython scripting language. Jython is a special implementation of the Python scripting language that enables scripts written in Jython to interact with programs written in Java. For further details on how to write Jython utility scripts that utilize the Java classes from ResSim and HEC-DSSVue, see the scripting chapter in the HEC-DSSVue User's Manual (HEC, 2009 or later).

Two interfaces are provided in ResSim for working with your scripts. Both are available from the **Tools** menu (Section 2.3.1.4) of any ResSim module:

Scripts…—The first option, **Scripts…**, opens the **Scripts** window, where you can collect buttons for launching your scripts. In addition, there are options available from the **Scripts** window for scheduling the launch of a utility script and for monitoring the status of a scheduled script.

Script Editor…—The second option, **Script Editor…** opens the interface you will use to create, edit, and test your scripts. Use the Script Editor to perform all your script file management.

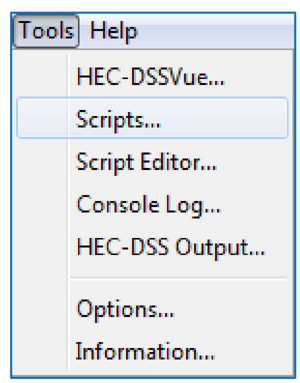


Figure 18.1 Tools Menu Scripts…

18.1 The Scripts Window

The Scripts window is simply a module-specific dialog where you can place buttons for launching your scripts. It is provided to make access to your scripts easier than it would be if you had to search for your script in a list of available scripts. Figure 18.2 illustrates the Scripts window for each module of ResSim; note that the name of the module from which you opened the **Scripts** window appears in the window's title bar. The script buttons that will appear in each module's **Scripts** panel are set by you and each panel is independent of the others. The idea is that scripts you would use in one module are not necessarily the same as those you would use in another. However, most scripts you will write will be used in the Simulation module, so, for simplicity, the following discussion of utility scripting in ResSim assumes you are working in the **Simulation** module.

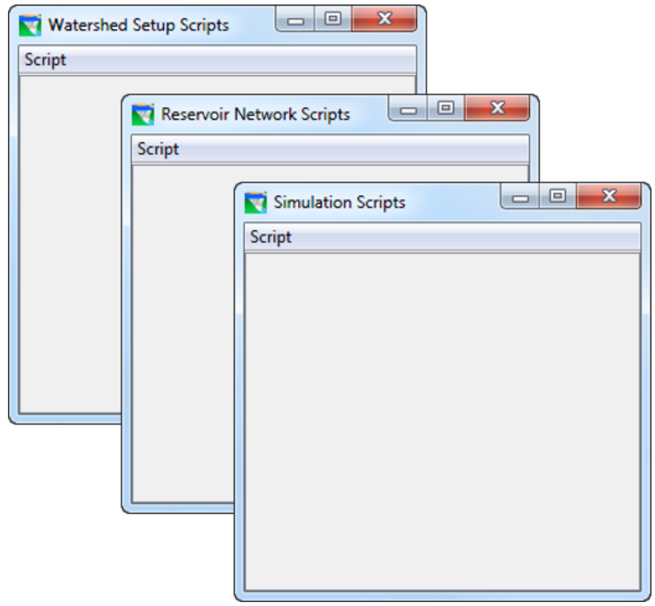


Figure 18.2 Scripts Windows for each Module

To open the **Scripts** window:

- Select Scripts... from the **Tools** menu of the main ResSim window.

The **Scripts** window contains one menu in its menu bar—**Script**. From the **Script** menu (Figure 18.3), you can choose from the following options:

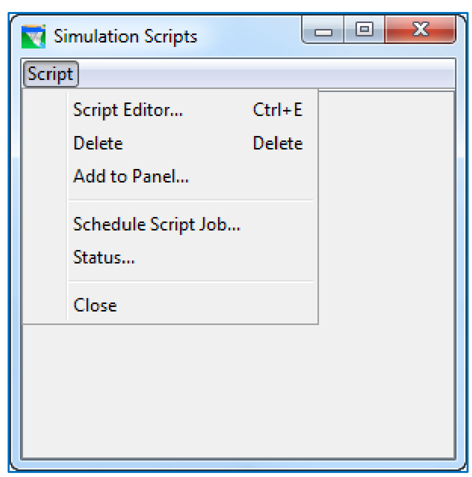


Figure 18.3 Scripts Window—Script Menu

Script Editor—in addition to accessing the Script Editor from the Tools menu of the main ResSim interface, you can access it from the Scripts panel.

Delete—select one or more scripts to delete from folder in the AppData area of user profile where ResSim stores your active scripts.

Add to Panel—add a script button to the **Scripts** panel of the main ResSim interface. This panel is at the bottom right of the Simulation module, below the compute button. The option opens a dialog that will also allow you to remove a button from the **Scripts** panel. **Add to Panel** cannot be used to add or remove buttons from the **Scripts** window; that is accomplished with the **Show in Script Selector** checkbox in the Script Setup pane of the **Script Editor**.

Schedule Script Job…—set a date and time to automatically run a selected script (as long as ResSim is running). The script can be scheduled to run once or on repeatedly on a regular interval (Figure 18.4).

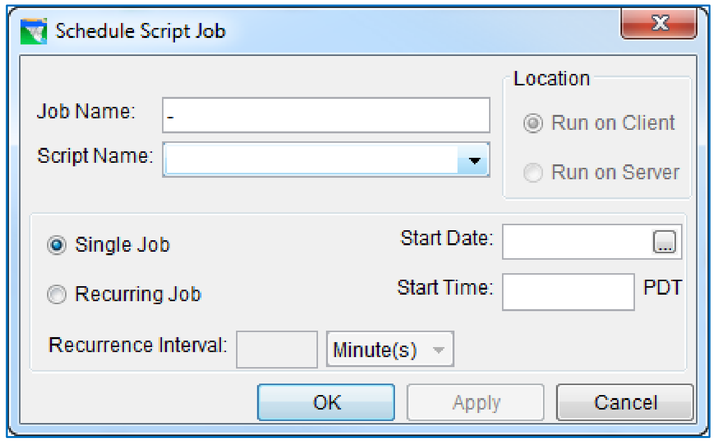


Figure 18.4 Schedule Script Job

Status…—opens the Script Job Status dialog (Figure 18.5) which lists the scheduled scripts their schedule and their status.

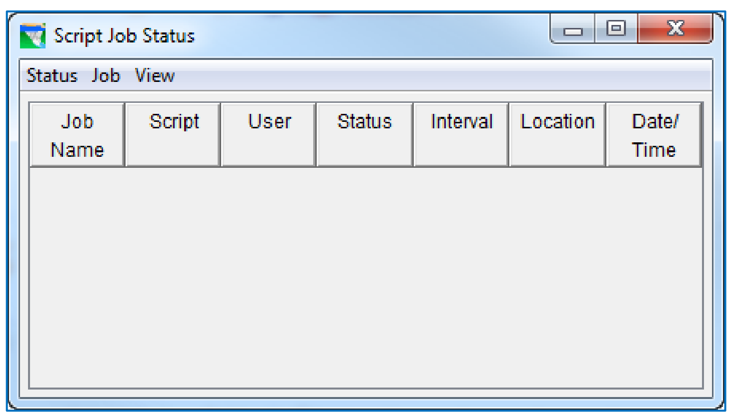


Figure 18.5 Script Job Status

Close—close the **Scripts** panel.

18.2 The Script Editor

The **Script Editor** option from the **Script** menu opens the **Script Editor** (Figure 18.6). The **Script Editor** contains a Menu Bar and three primary sections:

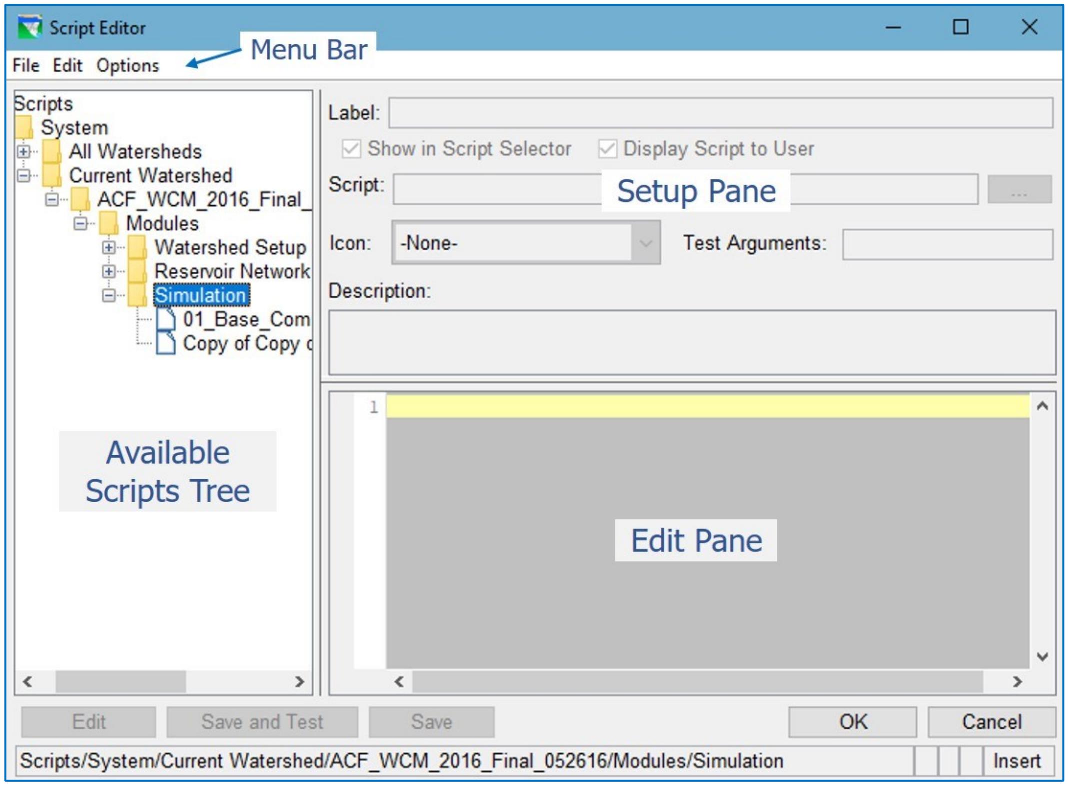


Figure 18.6 Script Editor

Available Scripts Tree—This tree is similar to a file system tree, but it displays the logical organization of how your scripts are stored. You can use this tree (or the **File** menu) to create, open, or import scripts in your watershed.

Setup Pane—This pane is for setting up a few properties of your script, including the label or icon that will be displayed on a script button, the name of the file containing the script, and a description of the script.

Edit Pane—This pane is where you will edit the script. The text editor of this pane recognizes Python source code and will colorize the text of your script to reflect key elements it recognizes such as comments, strings, keywords, constants, and parentheses.

The menu bar on the **Script Editor** includes the following three menus:

File—contains most of the standard options you'd expect to find in a **File** menu (Figure 18.7). One interesting aspect of the File menu is that some of the options are inactive (greyed out) depending on what is selected in the **Available Scripts Tree**.

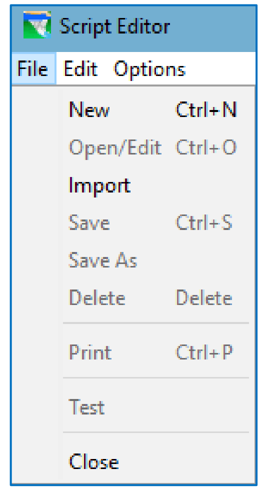


Figure 18.7 Script Editor File Menu

New—Before creating a new script, be sure to select the module the script will belong to in the Active Scripts Tree. When **New** is selected, the Setup and Edit panes become active and a few import lines are added to your new script. Be sure to give your script a name in the Label field; when you do, the **Script** field will display the filename of and path to the script.

Open/Edit—To open a script for editing, select an existing script from the **Available Scripts Tree**, then select **Open/Edit** from the **File** menu or click the Edit button below the tree. Or, you can just double-click on the script in the tree.

Import—When **Import** is selected, a File Browser will open to allow you to navigate your file system to select a script to import.

Save—Save the changes you have made to the script you have open in the **Script Editor.**

Save As—Save the script you have open in the Script Editor to a new file with a new name or label.

Delete—Delete the script you have selected in the Available Scripts Tree. If you do not have a script selected or open in the Script Editor, the Delete option will be unavailable.

Print—Print the script

Test—Run or execute the script you have selected in the Available Scripts Tree.

Close—Close the **Script Editor**. This action is also available through the **OK** and **Cancel** buttons at the bottom right of the **Script Editor**.

Note: Currently, ResSim does not edit or run utility scripts that are stored in the watershed folder. It stores them under the /users/ folder, which it creates in the AppData area of your profile on your computer. However, the most logical place to store your scripts IS in your watershed, so that if you move or share your watershed with a colleague, all necessary information is contained in the watershed.

Therefore, it is recommended that you maintain a utility script folder in your watershed directory. Before sharing the model with a new modeler, copy your current utility scripts from their location in the /users/ area* to a location in your watershed directory. The next modeler can then use the Import option to bring those scripts from their location in the watershed to the /users/ location, where ResSim will recognize them.

* C:\Users*yourUsername*\AppData\Roaming\HEC\HEC-ResSim\3.3\users*yourUsername**yourWatershedname*\

Edit—contains two sets of functions (Figure 18.8). The first set of functions are the standard text editing functions you might expect. Each entry for these functions in the **Edit** menu is followed by the shortcut keystrokes that can be used to apply the function without accessing it from the menu. The second set of functions in the **Edit** menu apply to the script file as a whole and could be considered "edits" to the **Available Scripts Tree**. These functions do not have shortcut keys.

The *text editing* functions include:

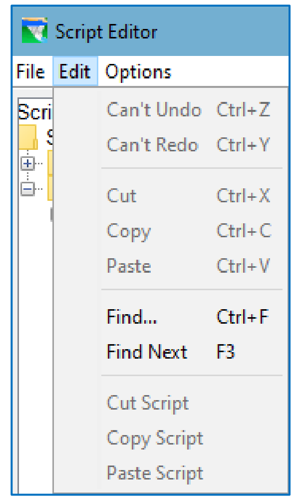


Figure 18.8 Script Editor Edit Menu

Undo—use this function to undo the last edit or edits to the text of your script. With each selection of this option, the last edit made to the text will be reversed and the text will be restored to its prior state. When there are no edits to *undo*, this menu entry displays in the menu as **Can't Undo** and is greyed-out.

Redo—use this function to redo the last undo. With each selection of this option, the last text edit that was reversed will be reapplied. When there are no edits to *redo*, this menu entry displays in the menu as **Can't Redo** and is greyed-out.

Cut—this function copies the selected text to the Windows Clipboard and deletes it from the script. If no text has been selected in the script edit panel, this function is greyed-out and cannot be selected from the menu and its shortcut keystrokes will perform no action.

Copy—This function copies the selected text to the Windows Clipboard. If no text has been selected in the script edit panel, this function is greyed-out and cannot be selected from the menu and its shortcut keystrokes will perform no action.

Paste—This function copies the text that was most recently placed on (copied to) the Windows Clipboard into the text of the current script at the current position of the text cursor.

Find—This option opens the **Find** dialog (as shown in Figure 18.9). Like most Find dialogs, you enter a string and select some options for how the find should operation. The

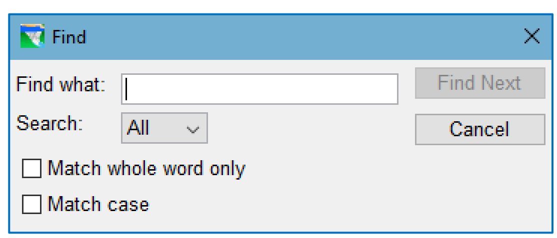


Figure 18.9 Script Editor—Edit Menu—Find Option

options include the search direction (All, Up or Down), whether the entered string represents a word and not a substring of a word, and whether or not case matters to the find.

Find Next—The **Find Next** function finds the next occurrence of the text that had been in the Find dialog the last time it was used. Find Next does not open the Find dialog, nor will it report failure if it doesn't find the string.

The ***Available Scripts Tree*** *edit* functions include:

Cut Script—This function deletes the currently selected script from the Available Scripts Tree and places it on the Clipboard. Because the script file is on the Clipboard, it can be pasted back into the current folder of the Tree or into another one.

Copy Script—This function copies the current script file onto the Clipboard.

Paste Script—This function copies the script file, if there is one, from the Clipboard into the current folder of the **Available Scripts Tree.**

The **Cut Script**, **Copy Script**, and **Paste Script** functions mimic the Delete, Save, and Save As functions from the File menu, *but they do not function as properly or consistently, so their use is not recommended.* Use the **File** menu functions.

Options—contains two features that influence how the tabs and spaces appear in the **Edit** pane (Figure 18.10).

Tab Size—to set how many spaces (blank characters) a tab represents.

Show Whitespace—this is a toggle that turns illustration of "whitespace" characters ON and OFF. When ON, tabs are drawn with arrows and spaces are drawn with dots. When OFF, tabs and spaces are drawn as blank characters.

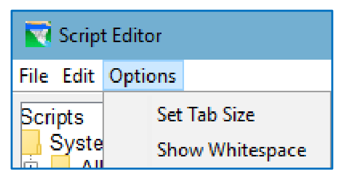


Figure 18.10 Script Editor—Options Menu

An example of using a Script might be to compute all the Alternatives contained within a Simulation (instead of running each Alternative one at a time). Figure 18.11 shows the contents of an example script to run all alternatives contained within a simulation. *NOTE: as previously described in Section 15.1.1 and indicated in Figure 14.5, the capability to run all alternatives in the simulation is available from the Simulation Menu → Run Manager.*

```
from hec.script import *
from hec.hecmath import *
from hec.heclib.dss import *
import time
import java.util.Vector
#
# The purpose of this script is to run all Alternatives in a ResSim Simulation.
# This script should be run from the Simulation module (==>Tools menu, Scripts..)
# There are two places where you will need to make revisions (search for REVISE)
# Have Fun and Good Luck!!!
#.................................. REVISE ................................
# the next line defines the name and location of this script's output log file
logFileName = "C:/Program Files/HEC/HEC-
ResSim/v3.0BetaVII/users/CWMS/BaldEagleDemo/scripts/Modules/Simulation/ResSimScript_log.out"
logFile = open(logFileName, "w")
#============================================================================
def log(str) :
	print str
	logFile.write("%s : %s\n" % (time.ctime(), str))
#============================================================================
def runSimulation(simulationName, alternativeName=None) :
	#---------------------------------------------------------------#
	# Make sure we're in the correct module and open the simulation. #
	#---------------------------------------------------------------#
	module = ClientAppWrapper.getCurrentModule()
	if `module` != "Simulation" :
		title = "Incorrect Module"
		msg = "This script must be run from the Simulation module."
		MessageBox.showError(msg, title)
		log("\n\n%s : %s\n" % (title, msg))
		return Constants.FALSE

	if not module.openSimulation(simulationName) :
		title = "Simulation Not Found"
		msg = simulationName
		MessageBox.showError(msg, title)
		log("\n\n%s : %s\n" % (title, msg))
		return Constants.FALSE

	#-----------------------------#
	# Run all the alternative(s). #
	#-----------------------------#
	if not alternativeName :
		runs = module.getSimulationRuns()
```

```
    else :
        run = module.getSimulationRun(alternativeName)
        if not run :
            title = "Alternative Not Found"
            msg = alternativeName
            MessageBox.showError(msg, title)
            log("\n\n%s : %s\n" % (title, msg))
            return Constants.FALSE
        runs = [run]

    for run in runs :
        print
        log("============================================")
        log("Computing %s" % run)
        log("============================================")
        #module.setActiveRun(run)
        module.computeRun(run, 5, Constants.TRUE, Constants.TRUE)
        log("============================================")
        log("%s done" % run)
        log("============================================")

    return Constants.TRUE
#================================================================================

#-------------------------#
# run the simulation #
#-------------------------#
#...................................... REVISE ...................................
# the next line defines the name of the Simulation containing Alternatives to be run
runSimulation("1993.11.29-0700")
logFile.close()
```

Figure 18.11 Example Script to Run Multiple Alternatives

References

Hydrologic Engineering Center, 2010. *Corps Water Management System (CWMS) Documentation, CWMS User's Manual, Version 2.0, October 2010*, U. S. Army Corps of Engineers, Davis, CA.

Hydrologic Engineering Center, 1998. *HEC-5, Simulation of Flood Control and Conservation Systems, User's Manual, October 1998, CPD-5*, U. S. Army Corps of Engineers, Davis, CA.

Hydrologic Engineering Center, 1995. *HEC-DSS (Data Storage System), User's Guide and Utility Manuals, March 1995, CPD-45*, U. S. Army Corps of Engineers, Davis, CA.

Hydrologic Engineering Center, 2009. *HEC-DSSVue, HEC Data Storage System Visual Utility Engine, User's Manual, Version 2.0, July 2009, CPD-79*, U. S. Army Corps of Engineers, Davis, CA.

Hydrologic Engineering Center, 2003. *HEC-FIA, Flood Impact Analysis, User's Manual, April 2003, CPD-81*, U. S. Army Corps of Engineers, Davis, CA.

Hydrologic Engineering Center, 2017. *HEC-WAT, Watershed Analysis Tool, User's Manual, Version 1.0, September 2017, CPD-88*, U. S. Army Corps of Engineers, Davis, CA.

U.S. Army Corps of Engineers, 1994. *EM 1110-2-1417, Streamflow and Reservoir Routing, Chapter 9, 31 August 1994*, Washington, D. C.

U.S. Army Corps of Engineers, 1987. *EM 1110-2-3600, Management of Water Control Systems, Engineer Manual, 30 November 1987*, Washington, D. C.

U.S. Army Corps of Engineers, 1991. *SSARR Model, Streamflow Synthesis and Reservoir Regulation, User Manual, January 1991*, Northwestern Division, Portland, OR.

HEC-ResSim

Reservoir System Simulation

User's Manual

APPENDICES

Version 3.3

February 2021

Appendix A
ResSim Application Settings

ResSim allows you to configure a variety of program options and preferences. You can create watershed locations for storing your watersheds, configure the fonts used in the user interface, and specify debug levels—to name just a few. These are managed by the ResSim **Options** Editor.

Other settings, mostly related to the environment that ResSim is running in, are also accessible. These settings can be viewed and managed through the **Application Properties** Editor.

This appendix describes both the **Options** Editor and the **Application Properties** Editor, all their tabs, and the various properties and settings that can be managed therein.

A.1 The Options Editor

To access the **Options** Editor:

- Select **Options…** from the **Tools** menu of any ResSim module.

As shown in Figure A.1, the **Options** Editor is arranged in *tabs*. Some tabs are module dependent, such as the **Simulation** tab which only appears when the **Options** Editor is accessed from the **Simulation** module of ResSim.

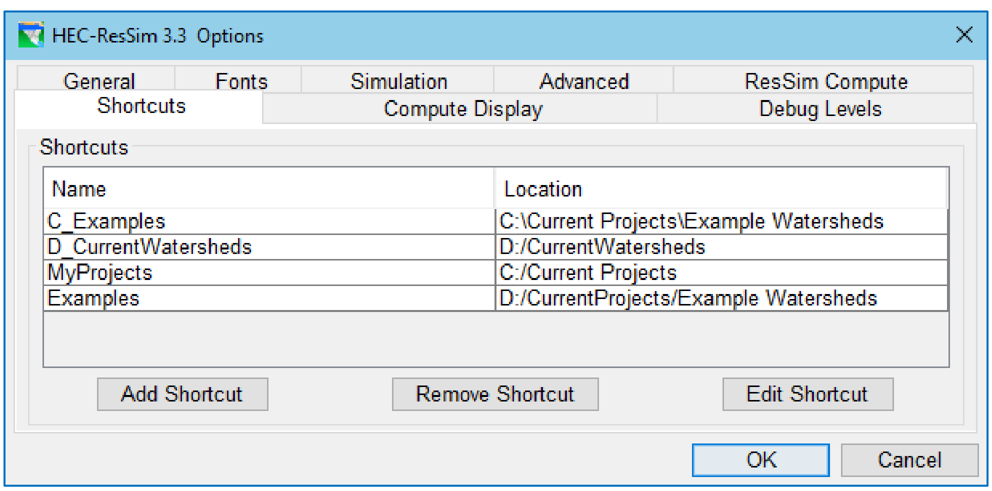


Figure A.1 ResSim Options Editor

Unless otherwise stated, any changes you make to settings in the **Options** editor are saved as global *User Preferences* and are retained until reset. In this context, global means: applies to *All* watersheds. ResSim's *User Preferences* are stored in the *AppData* folder of your Windows® user profile in a folder specific to the version of ResSim that you are currently working with. Since they are stored per version, each time you install a new version of ResSim, you will need to reset your preferences; they do not carry forward.

A.1.1 Shortcuts

A *Watershed Location*, or **Shortcut**, is the place on disk where you store your watersheds. You can have multiple watershed locations, although a single watershed cannot span multiple locations. Each watershed location is given an alias (or logical name) such as *My Watersheds* or *Current Projects*.

In prior versions of ResSim, you were required to create at least one watershed location before you could create a watershed. In version 3.3, that requirement has been removed; however there are still features in ResSim that expect your watershed to reside in a known watershed location so we recommend that you continue to create and use specified watershed locations (shortcuts) for storing your watersheds.

So, before you create or open a watershed, you should define at least one watershed location. To define a watershed location (a.k.a., a **Shortcut**):

- From the **Tools** menu, select **Options**. The **ResSim Options Editor** will open. See Figure A.1.
- Select the **Shortcuts** tab. The **Name** and **Location** of any watershed locations you have already defined will display in the **Shortcuts** list.
- To add a new location to the list, click the **Add Shortcut** button. The **Add Shortcut** dialog will open (Figure A.2)
- Enter a **Name** (or alias) for the new shortcut, then click **Browse…**. A file browser window will open. Use it to select the folder on disk where you want to store your watershed(s).
- Click **OK** to close the **Add Shortcut** dialog. The new watershed location (shortcut) will now appear in the list of **Shortcuts** on the **Shortcuts** tab (Figure A.1).

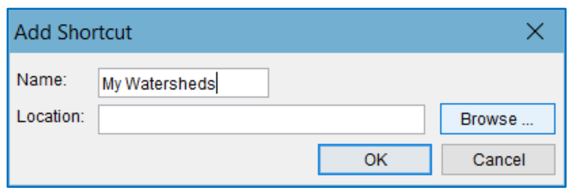


Figure A.2 Add Shortcut Dialog

A.1.2 Compute Display

When you compute a simulation in the Simulation Module, the ResSim **Compute Window** opens (Figure A.3).

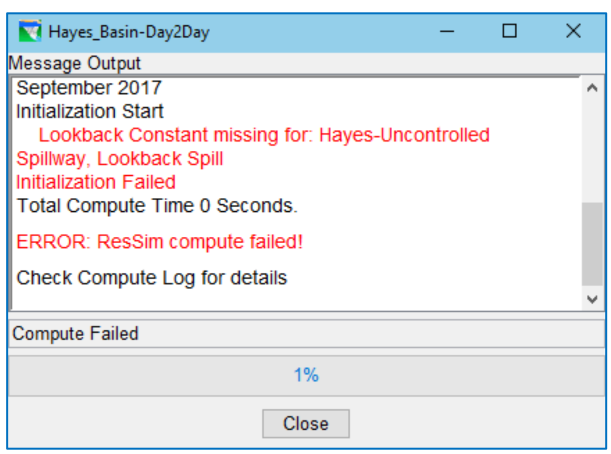


Figure A.3 ResSim Compute Window

The Compute Window reports on the progress of the currently computing alternative and displays messages generated by the ResSim compute engine.

The **Compute Display** tab of the **Options Editor** (Figure A.4) allows you to specify the text colors for the messages that are displayed in the Compute Window and to determine the format of the Compute Log.

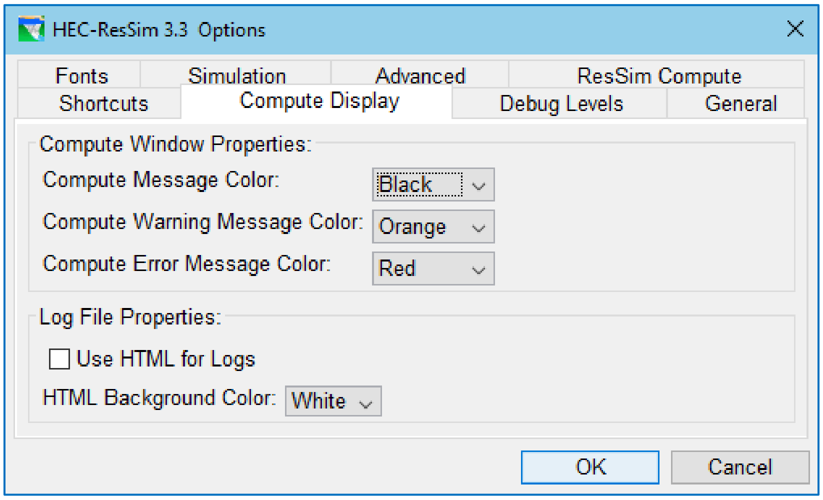


Figure A.4 Options Editor—Compute Display Tab

Compute Window Properties—contains the options for setting the text **Color** for:

- General **Messages**
- **Warning Messages**
- **Error Messages**

Log File Properties—this section contains options for specifying the format of the Compute Log:

- **Use HTML for Logs**—By default, the log is written in simple ASCII text format (the default) but if you would prefer that it be written in **HTML**, place a checkmark in the **Use HTML for Logs** checkbox.
- **HTML Background Color**—If the **Use HMTL…** checkbox is checked, then you can select a background color for display of the **HTML** formatted log.

A.1.3 Debug Levels

The **Debug Levels** tab (Figure A.5) lets you manage the level of diagnostic output (messaging) that ResSim will generate in the ResSim Console and Compute logs. The table on this tab lists **Log Flags** and their associated messaging **Levels** for various objects and features in ResSim that produce Level-controlled status messages. The default **Level** settings usually provide an adequate level of diagnostic output, but if you are having issues in a particular area, you may find it useful to increase the log **Level** (up to 10) for a relevant **Log Flag** to see if additional diagnostic information might be produced.

Note: Because ResSim *uses source code that is shared with other HEC applications, there are* Log Flags *in the list of Debug Levels that are not applicable to ResSim. For example, the Log Flag GridUtil is used by GridUtil & CWMS, not ResSim.*

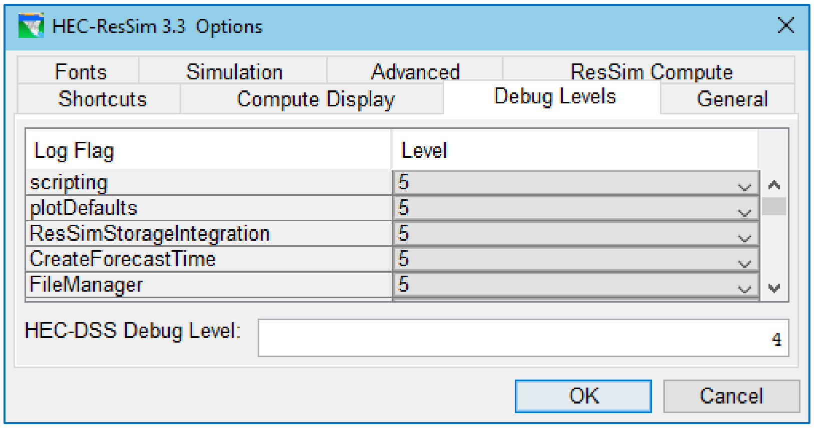


Figure A.5 Options Editor—Debug Levels Tab

HEC-DSS Debug Level—This log level is used by HEC-DSS to control the types of messages it produces to its log. The default value is 4 which will produce limited messaging in the DSS log for all DSS reads and writes. You can reduce DSS logging by entering a value of 1 or 2. Or you can increase the log level all the way up to 10 to get the maximum diagnostic output from DSS.

A.1.4 General

The **General** tab (Figure A.6) lets you manage settings that influence how ResSim behaves when opened or closed. These options are saved as user preferences for this version of ResSim and will remain as set until changed.

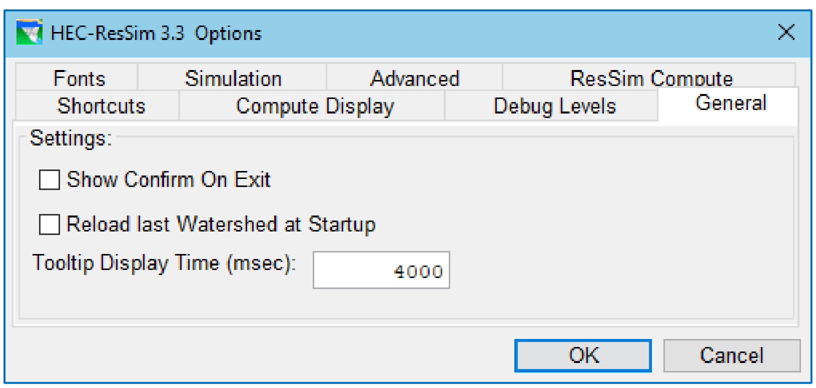


Figure A.6 Options Editor—General Tab

Show Confirm on Exit—When you first start using ResSim, you will find this option is checked by default. If you find it annoying that ResSim prompts you (Figure A.7) each time you try to close ResSim, then you can uncheck this option.

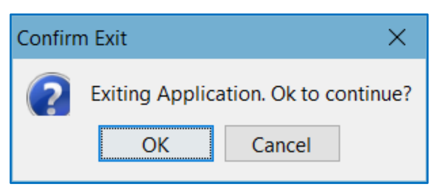


Figure A.7 Exiting Prompt

Reload Last Watershed at Startup—If you will be working on the same watershed for a while, this option is handy. Upon starting ResSim, it will reload the watershed you were working on when you last closed ResSim. This option is unchecked by default.

Tooltip Display Time—Tooltips appear when you pause your cursor over a user interface widget that has a tooltip defined for it. But they don't hang around long. This option lets you change how long a tooltip will be displayed before disappearing. The time is in milliseconds.

A.1.5 Fonts

The **Fonts** tab (Figure A.8) lets you specify the font that will be used for a variety of components (widgets) in the ResSim main user interface, windows, dialogs, and editors. These widgets include: Tabs, Labels, Descriptions for Radio Buttons and Checkboxes, Buttons, Lists and Text Fields.

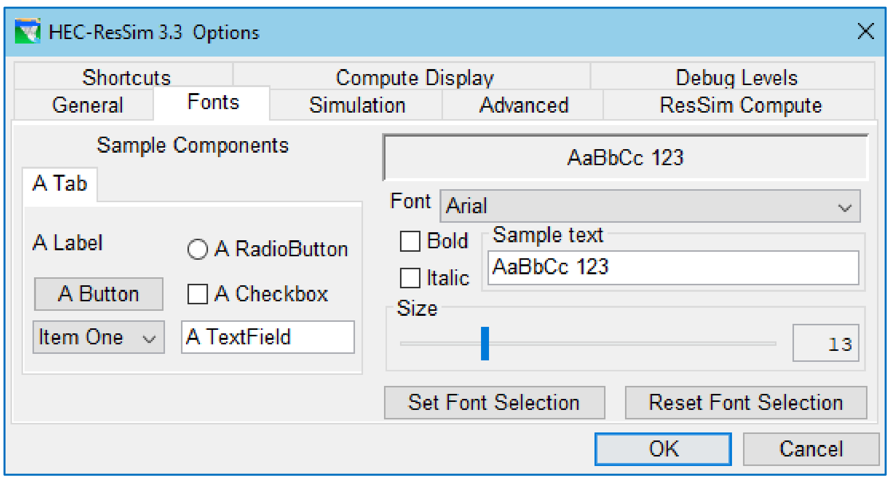


Figure A.8 Options Editor—Fonts Tab

Font—To change the font you see throughout ResSim, use the **Font** drop-down list to select your desired font. Use the fields below the **Font** selector to set the font **Size** and to make it **Bold** and/or *Italic*. You can even enter a different string of Sample Text to use to see how the font will look.

Set Font Selection—After making your selections, click the **Set Font Selection** button to apply your changes to the various widgets shown on the left side of the panel. If you like what you see, click **OK** to apply your selections and close the **Options** Editor. As ResSim will tell you, your new font settings will not be applied until the next time you start ResSim. *NOTE—you must click the **Set Font Selection** button before pressing **OK**, or your changes will not be applied.*

Reset Font Selection—If you don't like how your font settings appear on the **Sample Components**, click the **Reset Font Selection** button to start over (return to defaults) or press **Cancel** to discard your changes and close the **Options** Editor.

A.1.6 Simulation Options

The **Simulation** tab only appears in the **Options** Editor when accessed from the **Simulation** module with a watershed open. It provides some options that are reminiscent of those on the **General** tab but these options apply to the **Simulation** module.

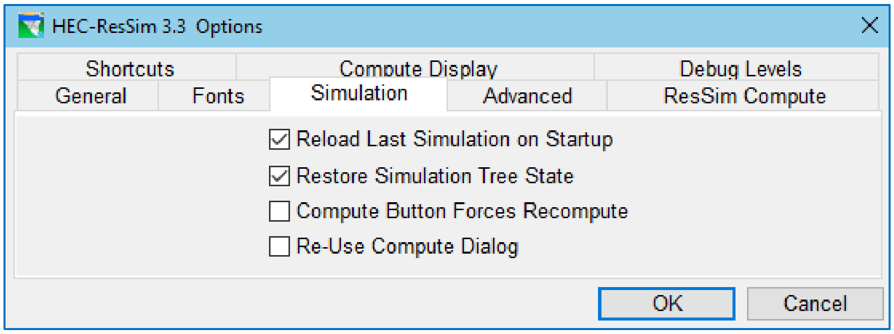


Figure A.9 Options Editor—Simulation Tab

Reload Last Simulation on Startup—If checked, this option will cause ResSim to load the simulation that you last had open in each watershed you open. This option will work even if you do not have **Reload Last Watershed at Startup** checked on the **General** tab.

Restore Simulation Tree State—The simulation tree state reflects the current settings of the tree in the **Simulation Control** panel. These setting include the identity of the active alternative and the checked/unchecked state of the checkboxes for each alternative.

Compute Button Forces Recompute—By default, this option is unchecked resulting in the following behavior:

The **Compute** button on the **Simulation Control** panel and the **Compute** option in an alternative's context menu will start a compute, but if ResSim thinks that nothing about the alternative has changed since the last compute, it will not actually perform the compute; instead, it will report in the **Compute Window** that it is "Skipping" the alternative. While in this default state, you can force a re-compute by holding the Ctrl key on your keyboard while clicking the **Compute** button or menu option.

By activating this option (placing a checkmark in the checkbox), no compute will be "skipped" even if no changes have been made.

Re-Use Compute Dialog—By default, the **Compute Window** (or Dialog) must be closed before a subsequent compute can be performed. By activating this option, the current compute must finish before another can be started, but the window will not need to be closed first.

A.1.7 ResSim Compute Options

The **ResSim Compute** tab (Figure A.10) gives you access to a number of settings that influence how ResSim performs its computations. Since you can only compute from the **Simulation** module in ResSim, the **ResSim Compute** tab only appears in the **Options** Editor when accessed from the **Simulation** module with a watershed open.

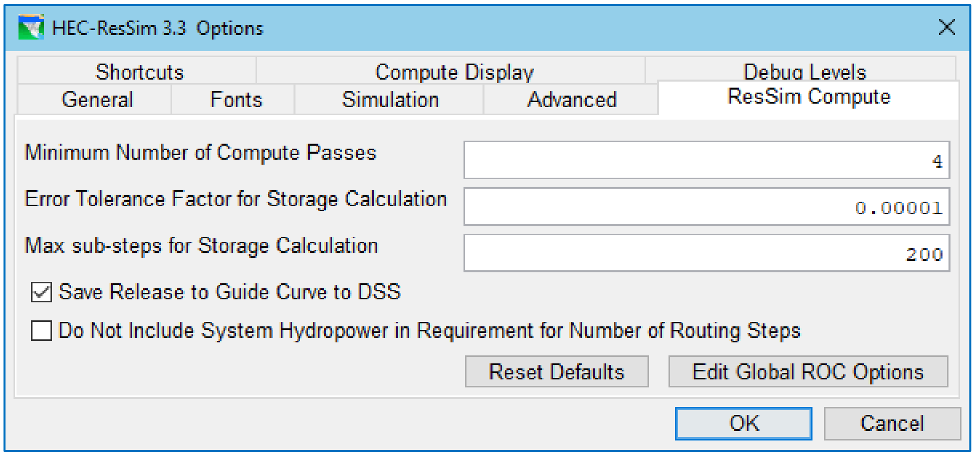


Figure A.10 Options Editor—ResSim Compute Tab

Minimum Number of Compute Passes—Each time ResSim computes through the time window is referred to as a "pass". Each time you tell ResSim to compute an alternative it will perform at least two passes—the idea is that it can use the results from a prior pass to improve the results of the current pass.

If the Minimum Number of Compute Passes is set to its default of 2, ResSim will perform two passes or more as mandated by the rule types being used in the alternative. During the first pass, ResSim evaluates only "at site" rules. It will ignore downstream and system rules, IF-Blocks, as well as any state variables and scripted rules. During the second pass, ResSim evaluates all active state variables and scripted rules, IF-Blocks, and it adds in the downstream control rules, if there are any. During the third pass, ResSim adds in system rules (like Tandem, Pump Schedule, and System Hydropower). If there are no downstream control or system rules, ResSim will stop after the second pass. If there *are* downstream rules but no other system rules, ResSim will stop after computing a third pass. If there *are* system rules, ResSim will compute a fourth pass, evaluating all the state variables, IF-Blocks, and rules, just as it did in the third pass.

The option to set the **Minimum Number of Compute Passes** is provided to let you determine if a greater number of passes should be computed than what ResSim will do by default. As such, ResSim will ignore any entry less than 2.

So, if you have downstream control rules in your model, an effective entry would be *greater* than 3. And, if you have system rules, an effective entry would be *greater* than 4.

Error Tolerance Factor for Storage Calculation—ResSim uses an estimate of the average pool elevation during a time step to determine release capacity of each outlet, so the longer the timestep, the more potentially inaccurate the estimate of average pool elevation could be, which would result in a potentially inaccurate estimate of the release capacity of each outlet. The computed releases from an uncontrolled outlet, like an overflow spillway, are often more affected by this potential inaccuracy than controlled outlets are; this is due to the fact that uncontrolled outlets always release *at capacity*, while controlled outlets often release at less than capacity.

To reduce the inaccuracies, a "storage integration" scheme was added to integrate storage over the timestep in an effort to better estimate the average elevation over the timestep. Unfortunately, this storage integration scheme can run into trouble converging if you have sharp inflection points in the storage-elevation data or in the outlet capacity curves, or if your reservoir storage is small compared to the release capacity. In addition, if/when the storage integration scheme has trouble converging, it can add a significant amount of time to the compute.

The **Error Tolerance for Storage Calculation** and **Max Sub-steps for Storage Calculation** parameters were added to give you control over the storage integration scheme.

This option, **Error Tolerance for Storage Calculation**, is the convergence criteria in units of storage (acre-feet or cubic meters) for the integration scheme. The default value is 0.00001. If you increase the tolerance (enter a larger value, like 0.001), the method will converge easier but may be slightly less accurate.

Max Sub-steps for Storage Calculation—This is the other parameter you can use to influence the storage integration computations. This value represents the maximum number of time slices the method can use for integration of storage over the timestep. The default value is 200. If your timestep is relatively short, ResSim should need fewer sub-steps to compute a valid estimate of average elevation. If you reduce the maximum number of sub-steps, the method occasionally may not converge which could result in slightly less accurate results.

Each model is different so you may need to run some tests to determine how sensitive your model is to these parameters. The storage integration scheme outputs warning messages to the console log and error messages to both the Compute Window (and log) and to the console log.

Save Release to Guide Curve to DSS—as part of the release decision process, each reservoir computes a "desired release to get to guide curve" for each timestep. But these flows are not, by default, written as part of the standard output to the DSS file. If you activate this option, you can find the *desired guide curve release* by filtering the pathnames in DSSVue to a C-part of **FLOW-GC-RELEASE**.

Do Not Include System Hydropower in Requirements for Number of Routing Steps—because routing effects (lag and attenuation) can affect downstream control and system operations, ResSim attempts to estimate these affects at the start of the compute process. These estimates result in a routing window (or Number of Routing (time)Steps) over which ResSim iterates in an attempt to determine an appropriate release from each reservoir.

As a system rule, the routing between **System Hydropower** reservoirs can, by default, influence the size of the total routing window. But, since **System Hydropower** does not have a target location for the system reservoirs to operate for, the influence of routing between the system hydropower reservoirs may not be relevant (e.g., if all the reservoirs in the system were on parallel streams). Use this option to turn **off** the influence of the **System Hydropower** rules on the routing window.

Reset Defaults Reset Defaults —Press this button to reset the options listed above to their default values.

Note: Rate of Change Rules have more impact than you might think... ResSim's decision logic tries to meet its objectives as fast as possible, usually within the current timestep. But rate-of-change constraints are in direct opposition to that tendency; their objective is to slow things down. To more fully address this objective, logic was added to the downstream control and guide curve methods to account for rate of change constrains. So, besides acting as normal release limit rules, Rate of Change rules also impact downstream control and guide curve by extending the time window over which the two methods try to meet their objectives. The options below are intended to give you some control over the impact of Rate of Change rules on Guide Curve operations.

Edit Global ROC Options —Shortly after rate of change impacts were added to the guide curve logic, user control over those impacts was added. To access the options for controlling the impact of rate of change rules on the guide curve logic, click the **Edit Global ROC Options** button. The **Global ROC Options Editor** will open (Figure A.11).

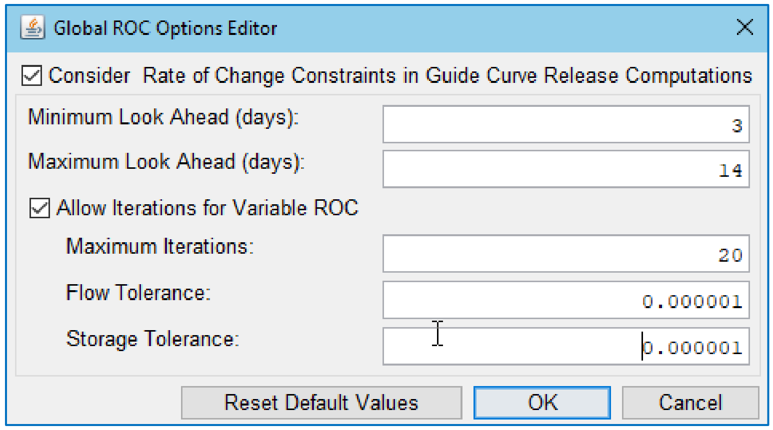


Figure A.11 Global ROC Options Editor

Consider Rate of Change in Guide Curve Release Computations—this checkbox acts as an ON/OFF switch for the application of Rate of Change rules upon the Guide Curve logic. By default, this option is ON by default. If you uncheck this option, all other parameters are deactivated.

Minimum Look Ahead—this option is the minimum number of days that the ROC rules will add to the time window over which the Guide Curve logic will attempt to get the reservoir back to guide curve.

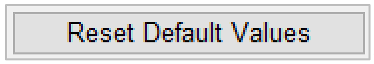
Maximum Look Ahead—this option is the maximum number of days that the ROC rules can add to the time window over which the Guide Curve logic will try to get the reservoir back to guide curve.

Allow Iterations for Variable ROC—Since rate of change rules can vary with inflow, outflow, or pool elevation, the impacts of the rate of change on the guide curve logic's time window can vary as a result of the decision that is currently being determined. This option allows you provide convergence criteria for evaluating the impact in this timestep. This feature is ON by default. The convergence parameters include:

Maximum Iterations—this value limits how many attempts ResSim can make to determine the time window impact on the Guide Curve logic per timestep.

Flow Tolerance—if the difference in flow between one iteration and another is less than this value, the iterations stop—the logic is considered to have reached "convergence".

Storage Tolerance—if the difference in ending storage between one iteration and another is less than this value, the iterations stop.



—If you have made changes to the Global ROC Options, you can return all the settings to the ResSim defaults by pressing this button.

Unlike most other settings in the Options Editor, the settings on the ResSim Compute tab are stored in the watershed, not as User Preferences.

A.1.8 Advanced Options

The **Advanced** tab gives you access to a couple of settings related to scripted State Variables and Rules. Since you can only interact with the **State Variable** and **Scripted Rule** editors from the **Network** and **Simulations** modules, the **Advanced** tab only appears in the **Options** Editor when accessed from the **Network** or **Simulation** modules.

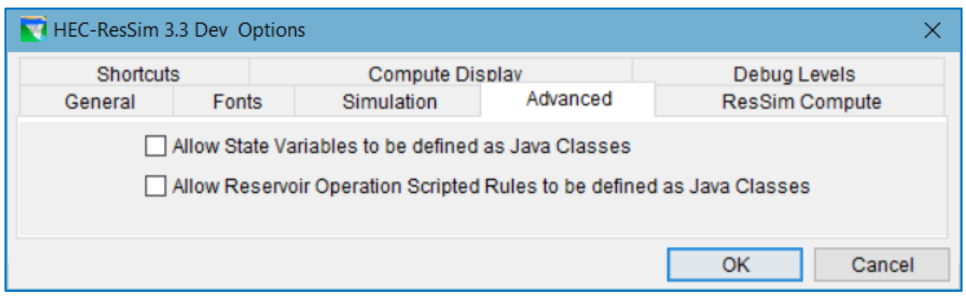


Figure A.12 Options Editor—Advanced Tab

Allow State Variables to be defined as Java Classes—by turning this option on, you active an option in the State Variable Editor that will allow you to identify a Java class file as the implementation of your state variable rather than the native Jython script. By writing your own Java class for your state variable you can improve compute time of your state variable by 20% or more. This option should be seriously considered for state variables scripts that are very long, complicated, and access a significant number of ResSim model objects and their methods.

Allow Reservoir Operation Scripted Rules to be defined as Java Classes—this option is exactly the same as the one above for State Variables, but it applies to scripted rules.

Since these settings impact how you create your operations in the current watershed, the settings on the Advanced tab are stored in the watershed, not as User Preferences.

A.2 The Application Properties Dialog

The **Application Properties** dialog is a viewer for a variety of properties related to the current watershed as well as the environment in which ResSim is running. Similar to a ***File->Properties*** dialog in a Windows application, the **Application Properties** editor provides details about your ResSim setup. These details include information about your local computer (your ResSim root directory, whether you are working in local or networked mode, and the IP address of your computer); your unique user properties (your user ID, user name, login, user directory path, preference directory, and access levels); watershed properties for the watershed you are viewing; server properties (application server, **File** manager, and login server); and system properties. Editing capability is limited and should be treated with care.

To open the **Application Properties** dialog:

- Select **Information** from the **Tools** menu of any ResSim module.

Like the **Options** editor, the **Application Properties** dialog is organized with tabs, as illustrated in Figure A.13.

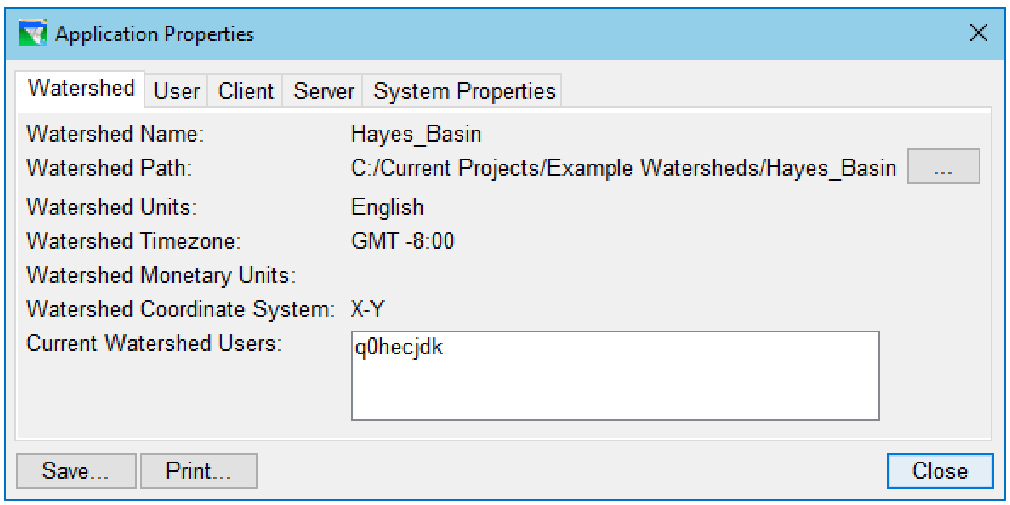


Figure A.13 Application Properties Editor—Watershed Tab

Note: ResSim was originally designed and built as a tightly integrated tool of a larger software development project (CWMS). Because of that effort, not only does ResSim share a lot of code with other HEC applications, it was also created to work in a client-server environment. Much of the information provided by the Application Properties dialog was considered important in that client-server framework but is of little to no interest when ResSim in run as a standalone program—which is how it is assumed you are running ResSim. Therefore, only those parameters and attributes that may be of interest in standalone have been documented here. For details on other parameters, you can refer to the CWMS User's Manual.

A.2.1 Watershed Properties

The **Watershed** tab of the Application Properties editor (Figure A.13) displays properties of the currently opened watershed including the:

- Name
- Path
- Units
- Time Zone
- Coordinate System

See Chapter 3 for details on how these properties are defined.

A.2.2 User Properties

The **User** tab of the **Application Properties** editor (Figure A.14) is primarily used for displaying user information. The only useful entry on this tab is the:

User Preferences Directory—the folder where your User Preferences are stored.

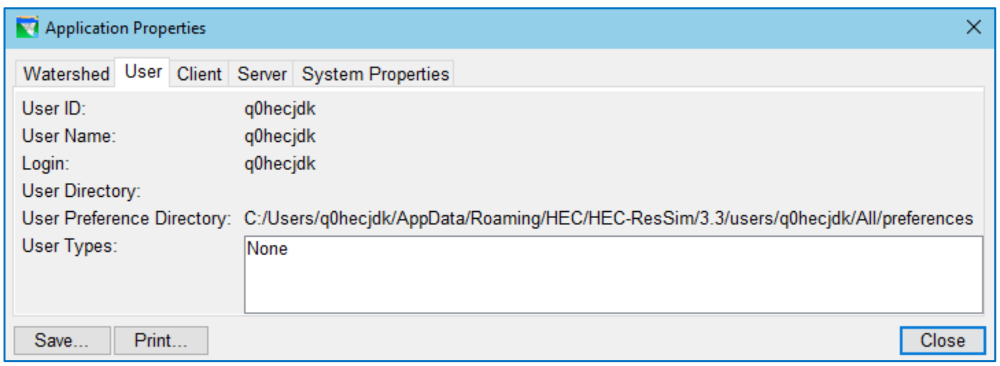


Figure A.14 Application Properties Editor—User Tab

A.2.3 Client Properties

The **Client** tab of the **Application Properties** editor (Figure A.15) displays information about ResSim and how it is running on your computer. The properties of interest include:

Base Directory—This is the path where you stored your watershed. If you identified a watershed location (shortcut) when you created the watershed, this is the directory the shortcut points to.

Java Version—This is the version number of the Java JRE that is running ResSim.

Jar Versions—This is a list of the program jars (Java Archives) that make up ResSim. Following each jar in the list is its build number. The jars should all have the same build number.

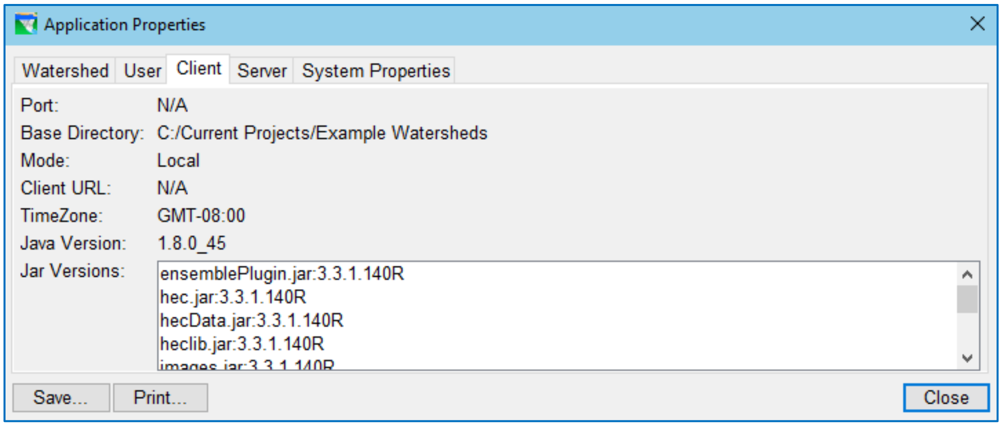


Figure A.15 Application Properties Editor—Client Tab

A.2.4 Server Properties

The **Server** tab of the **Application Properties** editor (Figure A.16) displays no useful information when ResSim is not running in Client-Server mode.

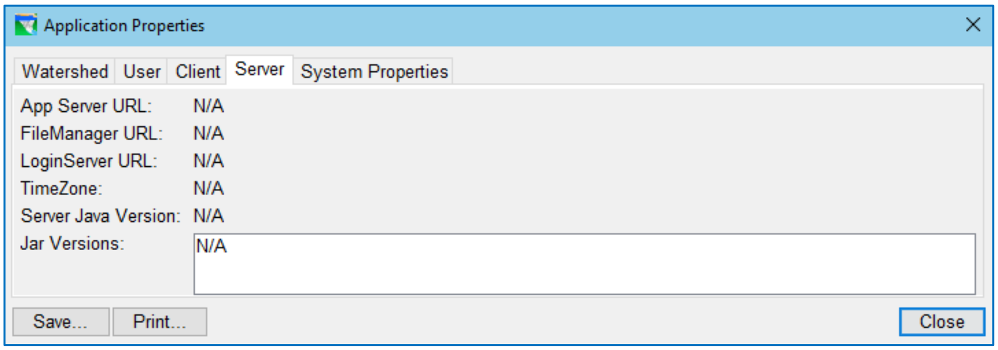


Figure A.16 Application Properties Editor—Server Tab

A.2.5 System Properties

The **System Properties** tab of the **Application Properties** editor (Figure A.17) displays a list of various environment variables and command line settings that are considered *system properties* of ResSim. Many of these variables are specified in the HEC-ResSim.config file that accompanies the HEC-ResSim.exe file which is used to launch (execute) ResSim. Although this tab has features necessary to edit these properties, don't use them.

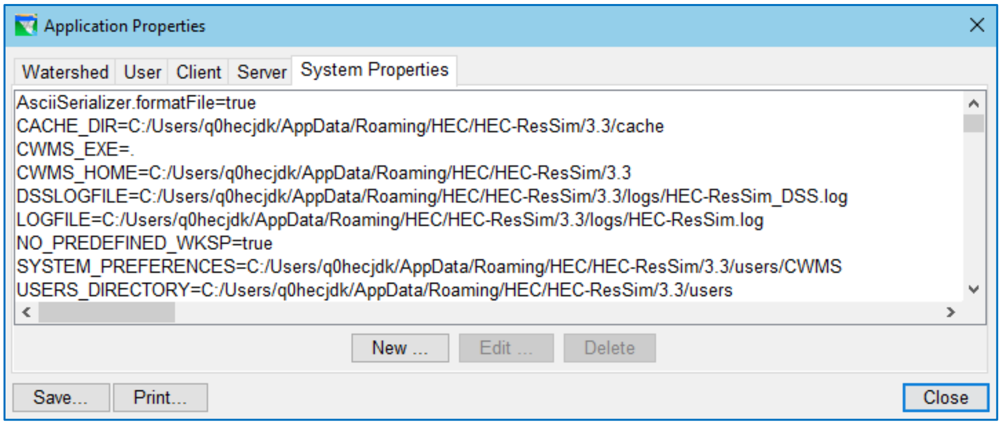


Figure A.17 Application Properties Editor—System Properties Tab

Very few, if any, of the System Properties should ever be changed by you, the user. But, if you must make changes to any of these properties, do so in the *HEC-ResSim.config* file (or its companion, the *HEC-ResSim - Personal.config* file located in your AppData area), not from this Editor.

Appendix B
Working with Map Display Layers

Layers in ResSim can be displayed, one on top of the other, in the Map Display area of ResSim. There are three types of layers used in the display: map layers, image layers, and ResSim layers. Map layers typically come from GIS shapefiles, DEMs, or DLGs and may contain features such as roads, political boundaries (cities, counties, etc.), rivers, subbasins, waterbodies, etc. Image layers are geospatially referenced images, which may also contain similar information, or perhaps photographic or satellite imagery. Each of these files, along with its associated data, is a called layer. In addition to these static map and image layers are ResSim schematic layers that are shown by default depending on the current module and the currently open dataset. Each ResSim module includes its own default layers -- Watershed Setup: Time Series Icons, Study, and Stream Alignment; Network: ResSim and Stream Alignment; Simulation: Model Schematic (which contains the ResSim layer) and Stream Alignment. These ResSim schematic layers contain elements that represent parts of the model schematic you create for your watershed.

Layers are considered hierarchical, which means that they may contain zero or more sub-layers. The first or top layer of a layer hierarchy is referred to as the primary or root layer. Each sub-layer usually represents a different type of component or element contained in the layer. Layers that do not have sub-layers usually contain only one type of drawing element or type. Most static map or image layers contain only one element type.

This appendix describes the various functions of the **Layer Selector** including:

- Adding Map Layers
- Setting the Watershed Coordinate System
- Editing Map Layer Properties
- Editing Schematic Layer Properties

B.1 The Layer Selector

The **Layer Selector** manages the organization and properties of the various Map Display Layers described above, including the Stream Alignment layer, the schematic layers, and the map layers.

To access the **Layer Selector** (Figure B.1) from any module:

- Select **Layers...** from the **View** menu.

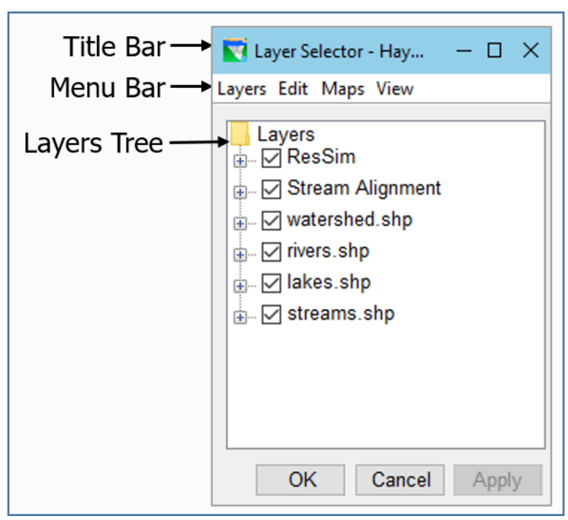


Figure B.1 Layer Selector

B.2 The Layer Selector Menus

The menus of the **Layer Selector** provide a variety of tools for managing the layers displayed in the Map Display area. If a command or option in a menu is *greyed-out*, the option is unavailable either because you have not turned on **Allow Layer Editing** from the **Edit** menu or the option cannot be used for the currently selected layer in the **Layers** tree. The menus of the Layer Selector include:

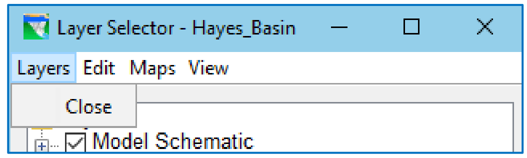


Figure B.2 Layers Menu

Layers (Figure B.2)—This menu is equivalent to a **File** menu. However, since the **Layer Selector** does not manage a file that contains its configuration settings, the only option in the **Layers** menu is **Close**, which will close the **Layer Selector**. Other ways to *close* the **Layer Selector** include:

- The **OK** and **Cancel** buttons at the bottom of the **Layer Selector**
- The **X** at the end of the Title Bar

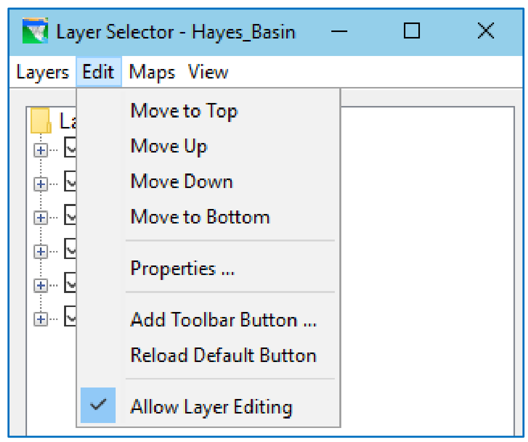


Figure B.3 Edit Menu

Edit (Figure B.3)—this menu provides tools for rearranging layers, viewing and editing layer properties, and adding and removing user toolbar buttons.

Allow Layer Editing—The **Layer Selector** uses a "locking" concept similar to that used by the **Watershed Setup** and **Reservoir Network** modules. The **Layer Selector** always opens in an "unlocked" state which limits the functions that you can perform on the existing layers and does not allow you to add layers. To enable all (or most) of the functionality in the **Layer Selector**, you must first "lock" it for editing by selecting **Allow Layer Editing** from the **Edit** menu. When editing is enabled, a checkbox appears next to the **Allow Layer Editing** menu item as shown in Figure B.3.

Move to Top,
Move Up,
Move Down,
Move to Bottom—These commands change the position of the currently selected layer in the Layers tree and how the layer is drawn in the Map Display. The current position of the selected layer will determine which Move commands are available in the Edit menu. When a layer is already at the top, the Move to Top and Move Up options will be unavailable.

Likewise, if a layer is at the bottom, the Move Down and Move to Bottom options will not be available.

Maps (Figure B.4)—This menu allows you to add and remove map layers, as well as import images as layers. The **Map Display Coordinates** option opens the **Geographic Region** editor. These functions are described in Section B.5.1 below.

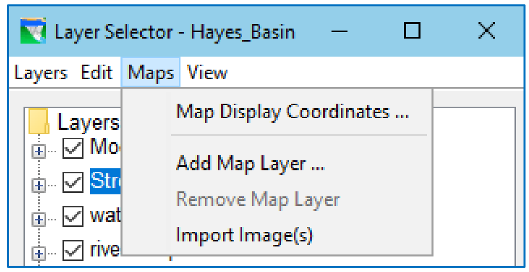


Figure B.4 Maps Menu

View (Figure B.5)—this menu has the following two options that impact the "view" of the **Layer Selector** itself:

Expand / Collapse—This option causes the selected layer in the Layers tree to display its sub-layers (or Legend). This is the same functionality as clicking on the plus-sign of the selected layer in the tree. Once you have expanded a layer, **Expand** changes to **Collapse** in the menu. Use **Collapse** to cause the selected layer to hide its sub-layers or Legend. This is the same functionality as clicking on the minus-sign of the selected layer in the tree.

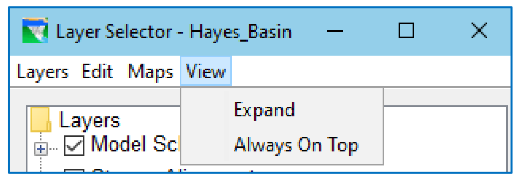


Figure B.5 View Menu

Always on Top—This option in the **View** menu is a "toggle switch". When active or ON, this option keeps the **Layer Selector** on top of all other windows on your desktop. If a checkmark precedes this option in the **View** menu, the option is active.

B.3 The Layers Tree

The Layers Tree is a tree widget used in the Layer Selector for displaying the hierarchical organization of the map, schematic, and image layers in your watershed. The top level of the tree (or root node) is the **Layers** folder, which contains all of the layers in the watershed relevant to the current module. Beneath the **Layers** folder are the *primary* layers. Each *primary layer* represents one of the layer types and is drawn as a branch in the tree with a plus/minus box, followed by a checkbox, then the layer name. A primary layer may have one or more components layers which is why a primary layer is referred to as a branch. Each component layer is drawn below the primary layer's checkbox with a plus-minus box and checkbox of its own.

In the following sections you will learn how to interact with the layers in the **Layer Selector** by using the options menus and context menus, the plus/minus boxes, the checkboxes, and the properties editors.

B.3.1 Controlling the Layer Display

The plus/minus box in front of each primary and component layer implements the same functionality as the **Expand** and **Collapse** options in the **View** menu.

Expand/Plus Sign—*Click* on a plus sign to expand a tree branch to display component sub-layers; if no sub-layers exist, you will see a legend for the layer (if a legend exists).

Collapse/Minus Sign—*Click* on a minus sign to collapse the tree branch or sub-branch below the minus sign.

You can also expand and collapse layers by double-clicking on the layer name (but don't be surprised if the **Properties** editor for that layer opens too.) Figure B.6 shows the **Layer Selector** with some of the Study Layer's sub-layers expanded.

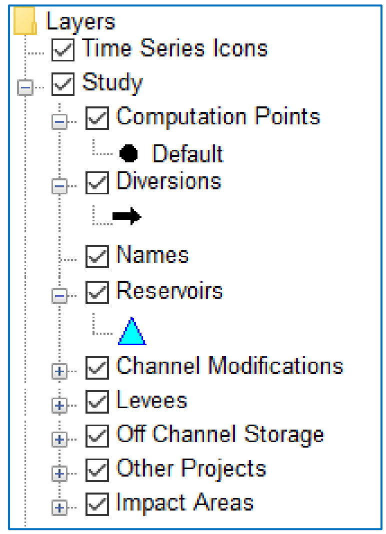


Figure B.6 Study Layer—Expanded

The checkbox that follows the plus/minus box turns the display of the layer ON or OFF in the **Map Display**. When a primary layer is un-checked (OFF), neither the primary layer nor its components layers will display. When checked (ON), the primary layer and all its *checked* component layers will display.

B.3.2 Viewing a Layer's Legend

When you *click* on the plus symbol to the left of a layer (or sub-layer) with no components or sub-layers of its own, the component's legend will be drawn below the component in the tree. <u>The *legend* is the icon or drawing style of the component *as it will appear in the Map Display*</u>.

The type of legend displayed is determined by the file formats or layer type. Figure B.6 shows the legend for Computation Points, Diversions, and Reservoirs, which are elements of the Study (watershed configuration) layer.

B.3.3 Accessing a Layer's Context Menu

You can access a primary layer's context menu by right-clicking on the layer in the **Layers** tree of the **Layer Selector**. Context menus offer commonly used commands that are also available in the **Layer Selector**'s **Edit** and **View** menus. Figure B.7 shows the context menu for the **Stream Alignment** layer.

If a command or option in a context menu is *greyed-out*, the option is unavailable either because you have not turned on **Allow Layer Editing** from the **Edit** menu or the option cannot be used for the currently selected layer in the tree.

The common commands available in a layer's context menu include:

Expand—opens the branch of a layer to display its component (sub-)layers. **Collapse** closes layer's branch to show only the primary layer. If a layer does not have sub-layers, **Expand** will show the layer's legend.

Move to Top,
Move Up,
Move Down, and
Move to Bottom—use these commands to rearrange the layers in the tree. The current position of an individual layer will determine which Move command are available. These options are described in detail in Section B.3.4.

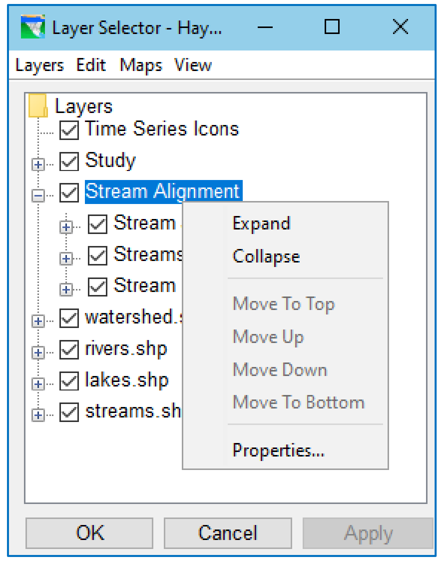


Figure B.7 Layer Selector—Schematic Layer Context Menu

Properties—opens a **Properties** editor for the layer, allowing you to change how the elements of that layer will be drawn in the Map Display area.

For *map* layers (Figure B.8), there are several additional options available in the context menu:

Show Legend option expands the map layer tree to display the map legend. This option changes to **Hide Legend** when the layer is expanded, allowing you to close the branch view of legends.

Change Label option allows you to change the name of the layer in the tree.

Set Scale for Zoom-in,
Set Scale for Zoom-out, and
Remove Scale Factors—use these commands to describe how the layer should appear in the Map Display relative to the zoom level.

Attributes Table...—this option will open a dialog showing a table of the attributes and their values for the elements in the selected shapefile layer.

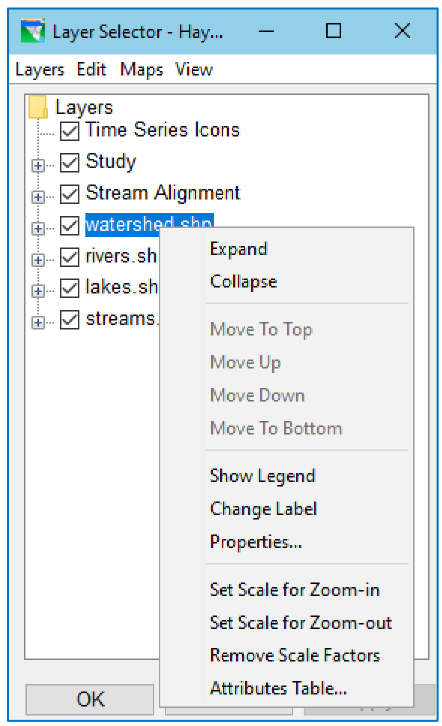


Figure B.8 Layer Selector—Map Layer Context Menu

B.3.4 Managing Layer Order

The **Layer Selector** controls how layers are arranged in the Map Display Area. When you add map layers to the watershed, the map layer is added to the bottom of the list of maps in the Layers Tree of the **Layer Selector**.

Use the **Layer Selector** to rearrange the order of the primary layers:

- From the **Edit** menu, select **Allow Layer Editing**.
- Select the layer you would like to move.
- From the **Edit** menu or from the layer's context menu, select:

  Move to Top—to move the layer to the front of the map display and to move the layer to the top of the layers in **Layers** tree.

  Move Up—to move the layer towards the front of the map display and to move the layer up in the **Layers** tree.

  Move Down—to move the layer toward the back of the Map Display and to move the layer down in the **Layers** tree.

  Move to Bottom—to move the layer to the back of the Map Display and to move the layer to the bottom of the layers in **Layers** tree.

To see your changes, click **Apply** (to keep the Layer Selector open) or click **OK** to close the **Layer Selector**.

B.4 ResSim Default Layers

ResSim has a three default layers, the **Time-Series Icon Layer**, the ***Schematic*** layer, and the **Stream Alignment Layer**. These layers are described in the following sections.

B.4.1 Time Series Icon Layer

Time-Series Icons are used in the CWMS CAVI to display data of various types for key locations in a watershed. They usually represent a gage and may contain datasets for each parameter measured at or derived for that gage.

The **Time Series Icon Layer** contains all of the time-series icons created in your watershed and may contain sub-layers that represent one or more dataset types.

> Although you can create Time-Series Icons and their layers in the Watershed Setup module in ResSim, **ResSim does not use time-series icons** nor does it display them in its Reservoir Network or Simulation modules. Refer to the CWMS User's Manual for additional details about Time-Series Icons.

B.4.2 Schematic Layer

The Schematic Layer has a different name, contains different component layers, and uses different display properties in each module of ResSim:

- In the **Watershed Setup** module, the schematic layer is called **Study.**
- In the **Reservoir Network** module, the schematic layer is called **ResSim**.
- In the **Simulation** module, the schematic layer is called **Model Schematic**.

B.4.2.1 Study Layer

The **Study** layer is the schematic layer that is displayed in the Watershed Setup module of ResSim. The **Study** layer contains all of the schematic elements that you can define in your watershed configurations including the various project types, impact areas, and computation points. The elements displayed by the **Study** layer in the **Map Display** area are a function of the currently selected **Configuration**.

When you are in the **Watershed Setup** module, the **Layer Selector** includes **Study** as a primary layer in the Layers tree (Figure B.9).

When you click on the plus sign in front of the **Study** layer in the **Layers** tree, the layer expands to show the set of component layers contained in the **Study** layer, including:

- Computation Points
- Diversions
- Names (labels for the projects, computation points, and impact areas)
- Reservoirs
- Channel Modifications
- Levees
- Off Channel Storage
- Other Projects
- Impact Areas

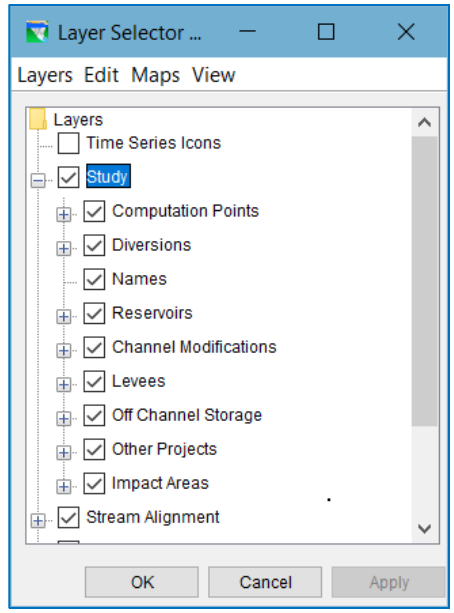


Figure B.9 Study Layer

ResSim creates a sub-layer for each component types even before you have defined any projects, computation points, or impact areas. Each component layer can be turned on and off without impacting the display of the other components.

B.4.2.2 ResSim Layer

The **ResSim** layer (Figure B.10) is the schematic layer that is displayed in the **Reservoir Network** module of ResSim. The **ResSim** layer contains all of the schematic elements that you can define in your reservoir networks. However, the elements displayed in the Map Display area of the **Reservoir Network** module as represented by the **ResSim** layer are a function of the currently open reservoir

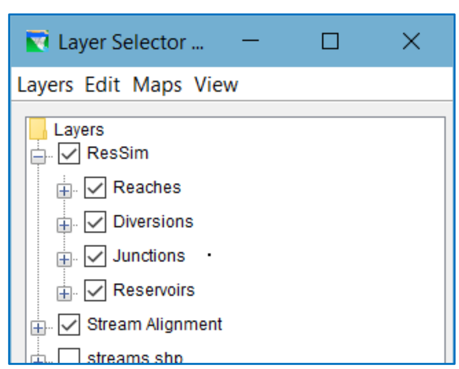


Figure B.10 ResSim Layer

network; if no network is open, only the stream alignment and map layers will be shown in the **Map Display** area and in the **Layers** tree of the **Layer Selector**.

When you are in the **Reservoir Network** module, the **Layer Selector** includes **ResSim** as a primary layer in the **Layers** Tree. When you click on the plus sign in front of the **ResSim** layer in the **Layers** Tree, the layer expands to show the set of component layers contained in the **ResSim** layer, including:

- Reservoirs
- Reaches
- Diversions (and Diverted Outlets)
- Junctions

B.4.2.3 Model Schematic Layer

The **Model Schematic** layer (Figure B.11) represents the model schematic in the **Simulation** module. It contains the **ResSim** layer and its component layers as described in Section B.4.2.2.

When you are in the **Simulation** module, the **Layer Selector** includes **Model Schematic** as a primary layer in the Layers tree.

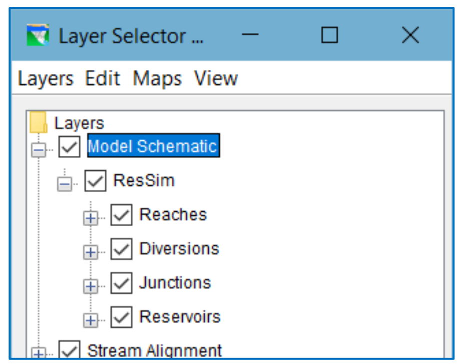


Figure B.11 Model Schematic Layer

The elements drawn in the **Map Display** area of the **Simulation** module as represented by the **Model Schematic** layer are a function of the currently *active* alternative. If no alternative is *active*, only the stream alignment and map layers will be shown.

B.4.3 Stream Alignment Layer

The **Stream Alignment** layer contains the Stream Alignment which represents the river system in your watershed. The **Stream Alignment** layer includes three component sub-layers (Figure B.12):

- Streams
- Stream Nodes
- Stream Junctions

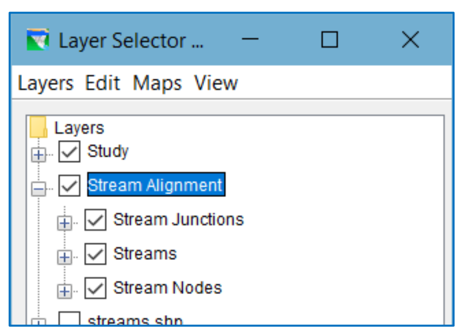


Figure B.12 Stream Alignment Layer

The **Stream Alignment** layer is available in the **Layer Selector** in all three ResSim Modules.

B.5 Map Layers

ResSim can display various types of maps in the geo-referenced Map Display area. These maps, displayed as map layers, are static images and are typically used as background for your model schematic to give the watershed "context". Examples of map layers include rivers, subbasins, county and state boundaries, etc. *Map* layers are not interactive.

Several map file formats are supported in ResSim including:

- ESRI® Shapefiles
- AutoCAD® DXF files
- ArcInfo® DEM files
- USGS Digital Line Graphs (DLG) files
- USGS Digital Elevation Model (DEM) files
- ASCII NET TIN files
- Raster images

B.5.1 Adding and Removing Map Layers

You will find it helpful to add maps to the watershed and the display area to provide a geographical reference for time-series icons, the stream alignment, and projects in your watershed.

B.5.1.1 Adding Map Layers

To add a background map to the **Map Display** area:

- First, copy any maps you wish to use to the *maps* folder of your watershed.
- From the **Edit** menu of the Layer Selector, select **Allow Layer Editing**.
- From the **Maps** menu, select **Add Map Layer…** A file browser will open to the default *maps* folder of your watershed.
- From the file browser, select the map you wish to add. *Click* **OK** to accept the chosen file and close the file browser.
- The selected map now appears in the Layer Selector Tree as a new primary layer (or branch). See "Adding a Map Layer" in Chapter 3 for more information.

B.5.1.2 Removing Map Layers

To remove a map layer from the **Map Display** area:

- Select the layer in the **Layers** tree of the **Layer Selector**
- Select **Remove Map Layer** from the **Maps** menu

B.6 Configuring Layer Drawing Properties

The **Layer Selector** provides three ways to access a **Properties** editor in order to view and configure the drawing properties of the selected layer. You can:

- Select a layer in the **Layers** tree, then select **Properties** from the **Edit** menu.
- *Right-click* on a layer in the **Layers** tree, the select **Properties** from its context menu.
- *Double-click* on a layer in the **Layers** tree.

Each of these techniques opens a **Properties** editor *specific to the layer type*. Each type of schematic layer, map layer, or image layer has its own editor for configuring the drawing properties of its components. For information about configuring the Time-Series Icon Layer, refer to the *CWMS User's Manual* (HEC, 2010).

B.6.1 Stream Alignment Layer Drawing Properties

When you select **Properties** for the **Stream Alignment** layer, the **Stream Alignment Properties Editor** (Figure B.13) appears.

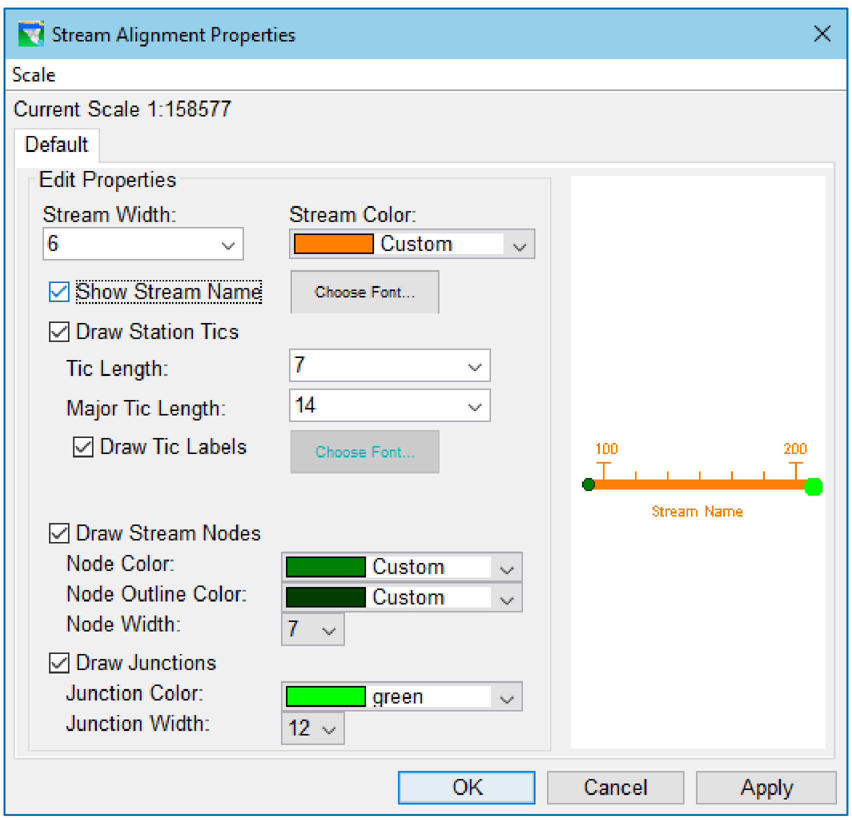


Figure B.13 Stream Alignment Properties Editor

The **Stream Alignment Properties** editor allows you to view and edit the following properties related to how the stream alignment is drawn in the **Map Display:**

Stream Width—Select the line width of the streams, in points, from the selection list. Possible values are 1—10. The default stream line width is 6.

Stream Color—Select the color of the stream lines from the list of pre-defined colors. The default stream color is a custom color of orange.

Show Stream Name—Use the checkbox to indicate if the stream name labels should be drawn with the streams in the **Map Display**. *Uncheck* the box to turn *OFF* the stream names.

Choose Font—This button opens the **Font Chooser** (Figure B.15). Use it to modify the font properties for the stream name labels. The stream names will be drawn using the **Stream Color.**

Draw Station Tics—Tic marks are drawn perpendicular to the stream line to mark approximate distance along the stream from downstream to upstream. Use the checkbox to indicate if the *stationing* tic marks should be drawn along the streams in the **Map Display**. *Uncheck* the box to turn *OFF* the tic marks. The tic marks and their labels will be drawn using the **Stream Color.**

Tic Length—Select the length of the minor tic marks, in points, from the selection list. The default minor tic length is 7.

Major Tic Length—Select the length of the major tic marks, in points, from the selection list. The default major tic length is 14.

Draw Tic Labels—Use the checkbox to indicate if the tic mark labels should be drawn with the tic marks of the streams in the **Map Display**. *Uncheck* the box to turn *OFF* the tic mark labels.

Tic Labels Font—*this* **Choose Font…** button will open the **Font Chooser.** Use it to set the font properties for tic mark labels.

Tip The stream alignment tic marks can strongly influence how the schematic layers are drawn, especially the labels for the various schematic elements. Once your stream alignment has been finalized, you probably won't need the station tic marks, so turn them off for a cleaner view of your schematic.

Draw Stream Nodes—Small circles are used to represent the stream nodes. By default, a stream node is created at the endpoints of every stream. Use the checkbox to turn ON or OFF the drawing of the stream nodes on the streams in the **Map Display**.

Node Color—Select the fill color of the stream nodes from the list of pre-defined colors. The default stream node color is a custom color of green.

Node Outline Color—Select the outline color of the stream nodes from the list of pre-defined colors. The default stream node color is a custom color of dark green.

Node Width—Select the diameter of the stream nodes, in points, from the selection list. Possible values are 1–10. The default node diameter (width) is 7.

Draw Junctions—Stream junctions are drawn at the connection of two or more streams; they appear as a larger circle (or halo) behind the smaller stream node circles that are created and drawn at the endpoints of every stream. Use the checkbox to turn ON or OFF the drawing of the stream junctions.

Junction Color—Select the color of the stream junction from the list of pre-defined colors. The default stream junction color is bright green, a custom color of green.

Junction Width—the size of the circle that forms the stream junction halo, in units of points.

The **Stream Alignment Properties** editor has a preview pane located on the right side of the editor. This preview pane allows you to view your changes before applying them.

B.6.2 Study Layer Drawing Properties

When you select **Properties** for the **Study** layer, the **Drawing Properties Editor** appears (Figure B.14). This editor has six tabs that allow you to view and edit properties of Reservoirs, Levees, Diversions, Channel Modifications, Computation Points, and Impact Areas in your watershed. The **Study** Layer is available only in the **Layer Selector** when opened from the **Watershed Setup** module.

B.6.2.1 Reservoirs

Reservoirs are drawn using four distinct elements:

- a **Triangle Element**—a re-shapeable polygon representing the reservoir pool,
- a **Storage Reach Element**—a thick line that follows the stream alignment and stretches from the inflow location(s) to the outflow location(s),
- a **Dam Element—**a rectangle drawn at the reservoir's outflow location(s), and
- a **Reservoir Name** label.

The **Reservoir** tab (Figure B.14) provides options for setting how reservoirs will appear in the **Map Display**. It is organized to present the drawing properties of each element of the reservoir separately.

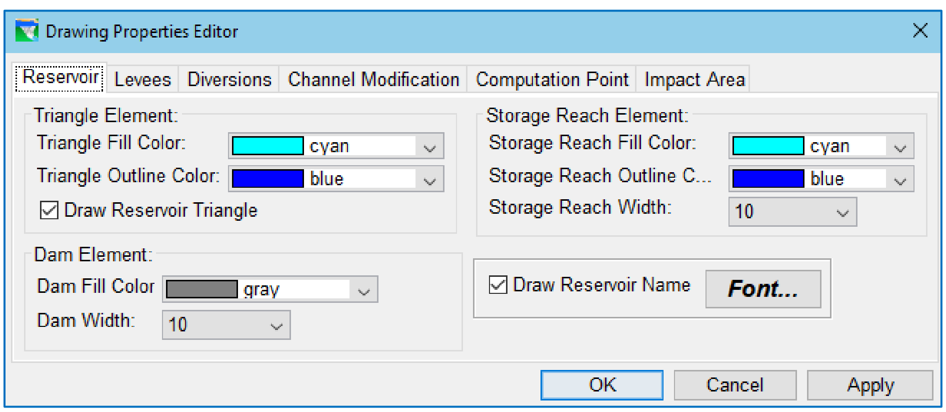


Figure B.14 Drawing Properties Editor—Reservoir Tab

The reservoir drawing properties you can modify include:

Triangle Fill Color—select the fill color of the reservoir pool polygons from the list of pre-defined colors. The default fill color is cyan.

Triangle Outline Color—select the color of the reservoir pool polygons outline from the list of pre-defined colors. The default line color is blue.

Draw Reservoir Triangle—Use the checkbox to indicate if the reservoir pool polygons should be drawn in the **Map Display** area. *Uncheck* the box to turn *OFF* the reservoir pool polygons.

Storage Reach Fill Color—select the fill color of the reservoir storage reaches from the list of pre-defined colors. The default fill color is cyan.

Storage Reach Outline Color—Select the color of the outline of the reservoir storage reaches from the list of pre-defined colors. The default outline color is blue.

Storage Reach Width—Select the width of the storage reaches, in points, from the selection list. Possible values are 1–10. The default width is 10.

Dam Fill Color—Select the (fill and outline) color of the dam rectangles from the list of pre-defined colors. The default color is grey.

Dam Width—Select the width of the dam rectangles, in points, from the selection list. Possible values are 1–10. The default width is 10. The length of the dam rectangle is a multiple of the width.

Draw Reservoir Name—Use the checkbox to indicate if the reservoir name labels should be drawn in the **Map Display**. *Uncheck* the box to turn *OFF* the reservoir names.

Font...—This button opens the **Font Chooser** (Figure B.15). Use it to select the **Font** and **Size** of the name label and whether it should be **Bold** and/or **Italic**. You can even change the **Sample Text** used to preview your selections. *Click* **OK** to save your settings and close the **Font Chooser**. The reservoir names will be drawn using the **Triangle Outline Color.**

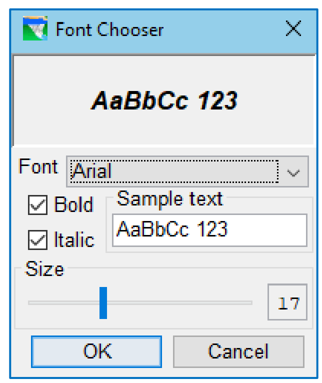


Figure B.15 Font Chooser

B.6.2.2 Levees

Levees are drawn as a line that parallels the stream alignment. The **Levees** tab (Figure B.16) provides options for setting the drawing properties of that line and its label.

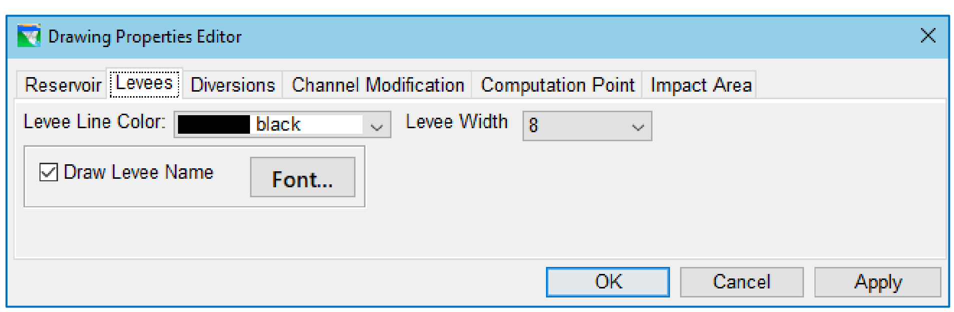


Figure B.16 Drawing Properties Editor—Levees Tab

The levee drawing properties you can modify include:

Levee Line Color—Select the color of the diversion line from the list of pre-defined colors. The default levee line color is black.

Levee Width—Select the diversion line width, in points, from the selection list. Possible values are 1–10. The default width is 8.

Draw Levee Name state—Use the checkbox to indicate if the name labels should be drawn with the levees in the **Map Display** area. *Uncheck* the box to turn *OFF* the levee names.

Font…—This button opens the **Font Chooser** (Figure B.15). Use it to modify the font properties for the levee name labels. The levee names will be drawn using the **Levee Line Color**.

B.6.2.3 Diversions

A diversion is drawn as a polyline with an arrowhead at the downstream end. The connectivity of the downstream end of a diversion is reflected in the color of the arrowhead.

The **Diversions** tab (Figure B.17), provides options for setting how diversions will appear in the Map Display area. The diversion drawing properties are presented in three sections—the **Line Element**, the **Arrowhead Element**, and the **Name** label.

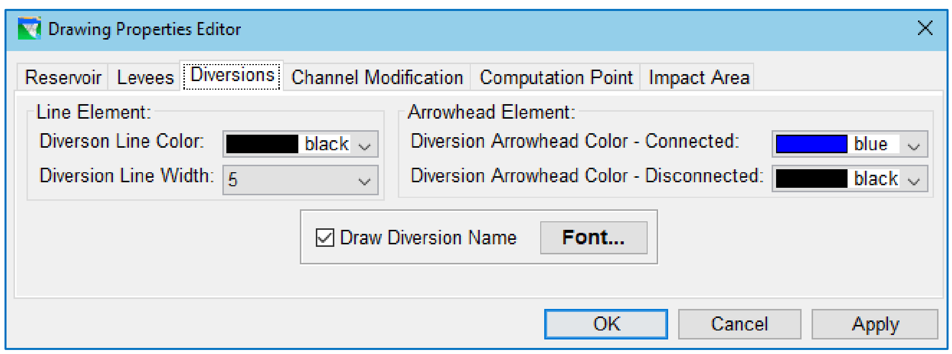


Figure B.17 Drawing Properties Editor—Diversions Tab

The diversion drawing properties you can modify include:

Diversion Line Color—select the color of the diversion line from the list of pre-defined colors. The default diversion line color is black.

Diversion Line Width—select the line width, in points, from the selection list. Possible values are 1–10. The default diversion line width is 5.

Diversion Arrowhead Color - Connected—This is the fill color of the arrowhead when the diversion is connected to a computation point at the downstream end. Select the color from the list of pre-defined colors. The default fill color is blue. The outline color of the arrowhead is always black.

Diversion Arrowhead Color - Disconnected—This is the fill color of the arrowhead when the diversion is **not** connected to anything at its downstream end. Select the color from the list of pre-defined colors. The default fill color is black. The outline color of the arrowhead is always black.

Draw Diversion Name—Use the checkbox to indicate if the name labels should be drawn with the diversions in the **Map Display** area. *Uncheck* the box to turn *OFF* the diversion names.

Font…—This button opens the **Font Chooser** (Figure B.15). Use it to modify the font properties for the diversion names. The diversion names will be drawn using the **Diversion Line Color**.

B.6.2.4 Channel Modifications

A channel modification is drawn as a thick line on top of the stream alignment. The **Channel Modification** tab (Figure B.18) provides options for setting how channel modifications will appear in the **Map Display** area.

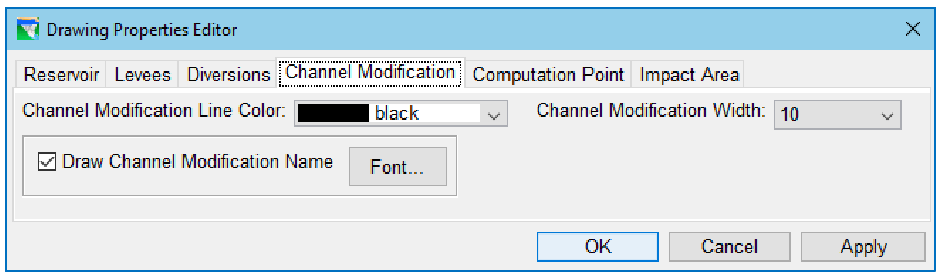


Figure B.18 Drawing Properties Editor—Channel Modification Tab

The channel modification drawing properties you can modify include:

Channel Modification Line Color—Select the color of the line drawn on top of the stream alignment from the selection list. The default color is black.

Channel Modification Width—Select the line width, in points, from the selection list. Possible values are 1–10; the default is 10.

Draw Channel Modification Name—Use the checkbox to indicate if the name labels should be drawn with the channel modifications in the **Map Display** area. *Uncheck* the box to turn *OFF* the channel modification names.

Font…—This button opens the **Font Chooser** (Figure B.15). Use it to modify the font properties for the channel modification name labels. The channel modification names will be drawn using the **Channel Modification Line Color**.

B.6.2.5 Computation Points

The **Computation Point** tab (Figure B.19) of the **Drawing Properties Editor** provides options for setting the computation point drawing properties, however *only* modifications to the **Computation Point Name** properties will *actually* impact what you see in the **Map Display**.

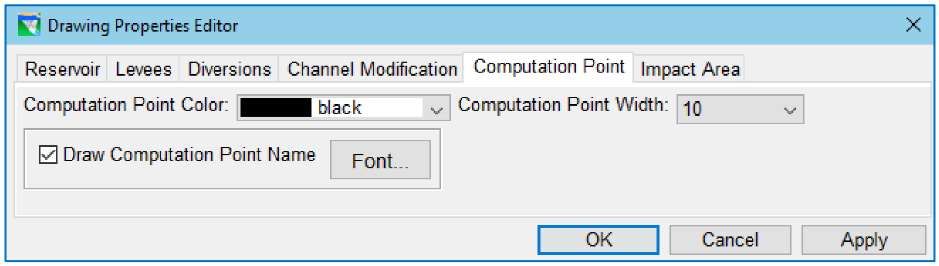


Figure B.19 Drawing Properties Editor—Computation Point Tab

The rest of the drawing properties for the computation point elements in the Map Display are managed by the Computation Point Layer Editor.

To open the Computation Point Layer Editor:

- Select Computation Point Layers from the Edit menu of the Watershed Setup module.

The **Computation Point Layer Editor** allows you to create computation point layers and set the drawing properties per layer. Every watershed is created with a default computation point layer—called Default. In ResSim watersheds, this is really all you need since ResSim cannot take advantage of additional computation point layers.

The computation point drawing properties available in the **Computation Point Layer Editor** (Figure B.20) include the **Computation Point**'s:

- Background Color—the outline of the symbol…
- Foreground Color—the fill color of the symbol…
- Symbol—the shape or text character used to represent the computation point on the map schematic.
- Symbol Size—a value, in points, between 4 and 25. A value of 10 is approximately equal to the size of a 10pt font.

See the HEC-WAT User's Manual for more details on Computation Point layers.

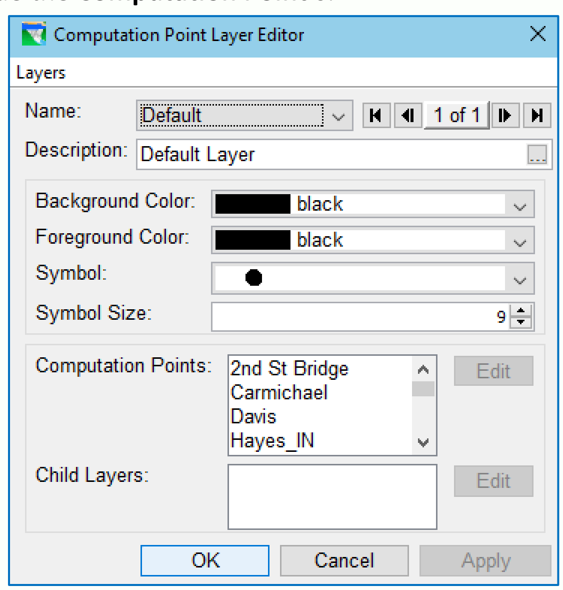


Figure B.20 Computation Point Layer Editor

The computation point *name* drawing properties you can modify from the **Drawing Properties Editor** include:

Draw Computation Point Name—Use the checkbox to turn ON or OFF drawing of the computation point name labels in the **Map Display**.

Font…—This button opens the **Font Chooser** (Figure B.15). Use it to modify the font properties for the computation point name labels. The computation point names will be drawn in black.

B.6.2.6 Impact Areas

The **Impact Area** tab (Figure B.21) provides options for modifying the Impact Area Name label properties. They include:

Draw Impact Area Name—Use the checkbox to indicate if the name labels should be drawn with the impact areas in the **Map Display**. *Uncheck* the box to turn *OFF* the impact area names.

Font…—This button opens the **Font Chooser** (Figure B.15). Use it to modify the font properties for the impact area name labels. The impact area names will be drawn in black.

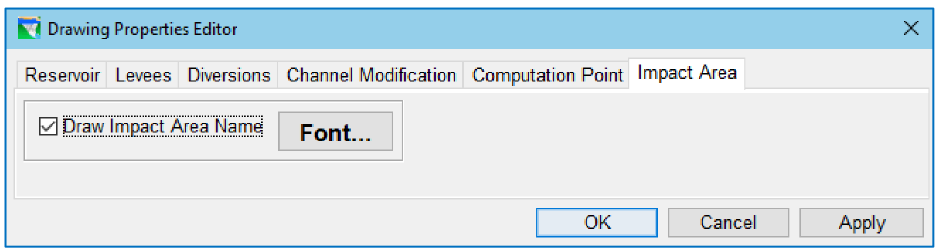


Figure B.21 Drawing Properties Editor—Impact Area Tab

B.6.3 ResSim Layer Drawing Properties

When you select **Properties** for the **ResSim** layer, the **ResSim System Draw Properties** editor appears (Figure B.22). This editor has four tabs that allow you to view and edit properties of the Reservoirs, Junctions, Reaches, and Diversions in your model schematic. The **ResSim** layer is available in the **Layer Selector** when opened from the **Reservoir Network** and **Simulation** modules. However, in the **Simulation** module, the **ResSim** layer appears as a sub-layer under **Model Schematic.**

B.6.3.1 Reservoirs

In the **ResSim** layer, reservoirs are drawn using the same four distinct elements as in the Study layer:

- a **Triangle Element**—a re-shapeable polygon representing the reservoir pool,
- a **Storage Reach Element**—a thick line drawn on top of the stream alignment; it stretches from the inflow junctions(s) to the outflow junctions(s),
- a **Dam Element—**a rectangle drawn behind the reservoir's outflow junction(s), and
- a **Reservoir Name** label.

The **Reservoir** tab (Figure B.22) of the **ResSim System Draw Properties** editor provides options for setting how reservoirs will appear in the **Map Display**, but there are fewer properties to set/modify than in the Study layer:

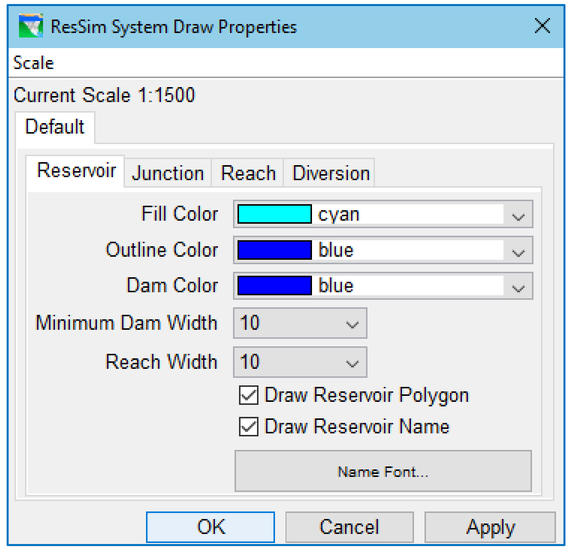


Figure B.22 ResSim Layer Draw Properties—Reservoirs

Fill Color—Select the fill color of the reservoir pool polygon from the list of pre-defined colors. The default fill color is cyan.

Note—the reservoir storage reach will be drawn with a dark version of the selected fill color; as such, the default fill color for the reservoir storage reach is a dark cyan.

Outline Color—Select the outline color of the reservoir pool polygon *and* the reservoir storage reach from the list of pre-defined colors. The default outline color is blue.

Dam Color—Select the fill and outline color of the dam from the list of pre-defined colors. The default dam color is blue.

Minimum Dam Width—Select the width of the dam rectangle, in points, from the selection list. Possible values are 1–15. The default width is 10. The length of the dam rectangle is a multiple of the width.

Reach Width—Select the line width, in points, from the selection list. Possible values are 1–15. The default storage reach width is 10.

Draw Reservoir Polygon—Use the checkbox to indicate if the reservoir pool polygons should be drawn in the **Map Display** area. *Uncheck* the box to turn *OFF* the reservoir pool polygons.

Draw Reservoir Name—Use the checkbox to indicate if the reservoir name labels should be drawn in the **Map Display**. *Uncheck* the box to turn *OFF* the reservoir names.

Name Font…—This button opens the **Font Chooser** (Figure B.15). Use it to modify the font properties of the reservoir name labels. The reservoir names will be drawn using the **Outline Color.**

B.6.3.1 Junctions

Junctions are drawn as filled circles. The **Junction** tab (Figure B.23) of the **ResSim System Draw Properties** editor provides options for modifying how junctions will appear in the **Map Display:**

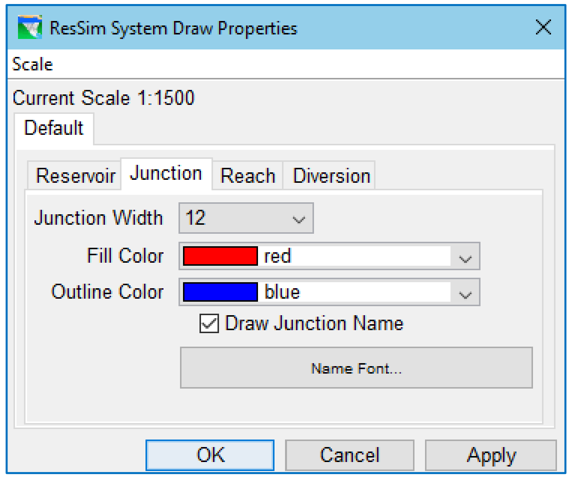


Figure B.23 ResSim Layer Draw Properties—Junctions

Junction Width—Select the diameter of the circle, in points, from the selection list. Possible values are 1–15. The default diameter is 12.

Fill Color—Select the fill color of the junction circle from the list of pre-defined colors. The default fill color is red.

Outline Color—Select the outline color of the junction circle from the list of pre-defined colors. The default outline color is blue.

Draw Junction Name—Use the checkbox to indicate if the junction name labels should be drawn in the **Map Display**. *Uncheck* the box to turn *OFF* the junction names.

Name Font…—This button opens the **Font Chooser** (Figure B.15). Use it to modify the font properties of the junction name labels. The junction names will be drawn using the **Fill Color.**

B.6.3.1 Reaches

Reaches are drawn as a thick line on top of the stream alignment. The **Reach** tab (Figure B.24) of the **ResSim System Draw Properties** editor provides options for modifying how reach will appear in the **Map Display:**

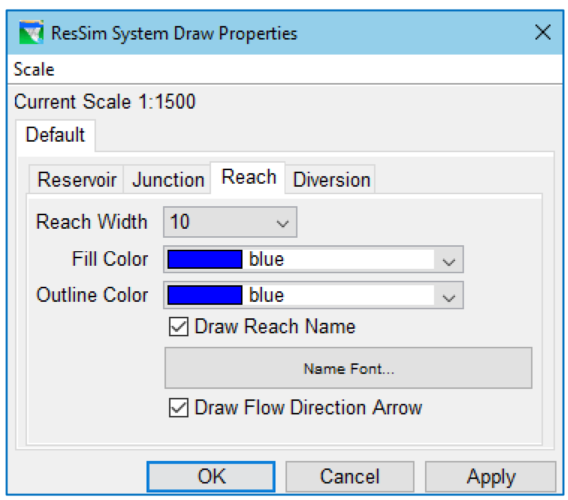


Figure B.24 ResSim Layer Draw Properties—Reaches

Reach Width—Select the width of the reach line, in points, from the selection list. Possible values are 1–15. The default diameter is 10.

Fill Color—Select the fill color of the reach line from the list of pre-defined colors. The default fill color is blue.

Outline Color—Select the outline color of the reach line from the list of pre-defined colors. The default outline color is blue.

Draw Reach Name—Use the checkbox to indicate if the reach name labels should be drawn in the **Map Display**. *Uncheck* the box to turn *OFF* the junction names.

Name Font…—This button opens the **Font Chooser** (Figure B.15). Use it to modify the font properties of the reach name labels. The reach names will be drawn using the **Outline Color.**

Draw Flow Direction Arrow—Use the checkbox to indicate if the flow direction arrows should be drawn at the midpoint of each reach in the **Map Display**. *Uncheck* the box to turn *OFF* the flow direction arrows.

B.6.3.1 Diversions

A diversion is drawn as a polyline with an arrowhead at the downstream end. The connectedness of a diversion is reflected in the color of the arrowhead.

The **Diversion** tab (Figure B.25) of the **ResSim System Draw Properties** editor provides options for modifying how diversion will appear in the **Map Display:**

Fill Color—Select the fill color of the reach line from the list of pre-defined colors. The default fill color is blue.

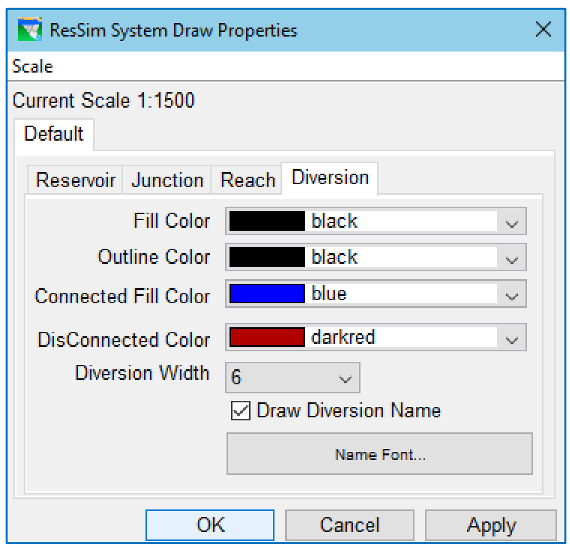


Figure B.25 ResSim Layer Draw Properties—Diversions

Outline Color—Select the outline color of the diversion reach line from the list of pre-defined colors. The default outline color is blue.

Connected Fill Color—Select the outline color of the reach line from the list of pre-defined colors. The default outline color is blue.

DisConnected Fill Color—Select the outline color of the reach line from the list of pre-defined colors. The default outline color is blue.

Diversion Width—Select the line width of the diversion, in points, from the selection list. Possible values are 1—15. The default line width is 6.

Draw Diversion Name—Use the checkbox to indicate if the reach name labels should be drawn in the **Map Display**. *Uncheck* the box to turn *OFF* the junction names.

Name Font…—This button opens the **Font Chooser** (Figure B.15). Use it to modify the font properties of the reach name labels. The diversion names will be drawn using the **Outline Color.**

B.6.4 Map Layer Drawing Properties

Map layers can be any of a number of formats supported by ResSim, including ArcView® Shapefiles, ArcInfo® DEM files, AutoCAD® DXF files, U.S Geological Survey (USGS) Digital Line Graphs (DLG) files, USGS Digital Elevation Model (DEM) files, ASCII NET TIN files, and raster images. So, when you select **Properties** for a map layer, the editor that opens is specific to the type of map. ResSim allows you to configure several options for each type of map (except for AutoCAD® DXF files).

To access the **Properties** editors for a map layer:

- Open the **Layer Selector** (Figure B.26) by selecting **Layers…** from the **View** menu of any module.
- In the **Layers** Tree, double-click on the map layer or right-click on it and select **Properties** from the context menu.

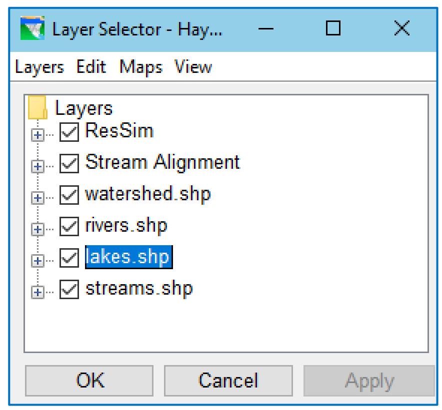


Figure B.26 Layer Selector

An editor specific to the type of map will open. The directory location of the map file in your ResSim watershed is shown at the top of each **Properties** editor.

The following sections describe most of the map layer editors currently available in ResSim.

B.6.4.1 Shapefiles (*.shp)

The shapefile spatial data format was developed by the Environmental Systems Research Institute, Inc. (ESRI) and is now a (mostly) open geospatial vector data format that can be used by ArcGIS and other geographic information system (GIS) software. Shapefiles store non-topological geometry and attribute information for the spatial features of a data set.

ResSim can use three types of shapefiles (*.shp): Point, Line, and Polygon. Each shapefile type has its own properties editor. Each properties editor is organized into 2 or 3 tabs. One of the tabs in each editor is the **Label** tab which looks and works the same in each editor and is described in its own section following the descriptions of the three shapefile editors.

ResSim saves the drawing properties of your shapefile(s) to a file with the same base name as your shapefile and the extension *.gdr*. This file will be stored in the *maps* folder for your watershed.

Point Shapefile Properties

If the map layer you wish to configure is a *point* shapefile (e.g., gages, control points), selecting **Properties** for the map layer will open the **Edit Point Properties** editor. This editor is organized in two tabs—**Style** and **Label**.

The **Style** tab lets you set the drawing properties of the Point elements in the shapefile. The drawing properties that you can modify are a function of the first field in the panel, **Draw Features using**. The options are:

One Style (Figure B.27)—under this option, the drawing properties are:

Style—the symbol used to represent each point.
Color—the color of the symbols.
Size—the size of the symbols, in points.

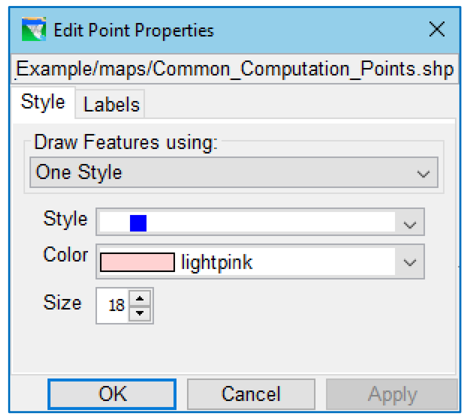


Figure B.27 Edit Point Properties Style Tab—One Style

Attribute Values (Figure B.28)—under this option, the drawing properties expand to:

Field for Values—select the *attribute* whose values will determine the color of each point.
Style—the symbol used to represent each point.
Size —the size of the symbols, in points.
Color—two colors are used to define the **Start** and **End** colors of a color ramp. ResSim will determine the number of unique values in the selected attribute field, then create a gradation set of colors ranging between the **Start** and **End** colors, with a different color in the gradation for each unique attribute value. The set of colors and their associated attribute values will be displayed in the **Color-Value** table.

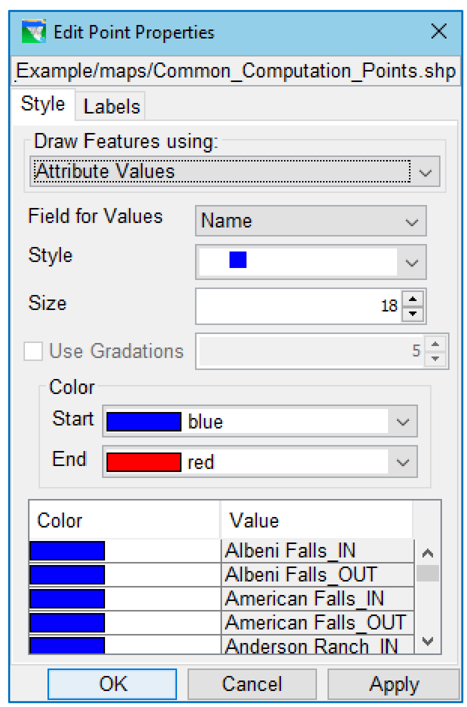


Figure B.28 Edit Point Properties Style Tab—Attribute Values

Line Shapefile Properties

If the map layer you wish to configure is a *line* shapefile (e.g., rivers and streams), selecting **Properties** for the map layer will open the **Edit Line Properties** editor. This editor is organized in two tabs—**Style** and **Label**.

The **Style** tab lets you set the drawing properties of the Line elements in the shapefile. The drawing properties that you can modify are a function of the first field in the panel, **Draw Features using:**. The options are:

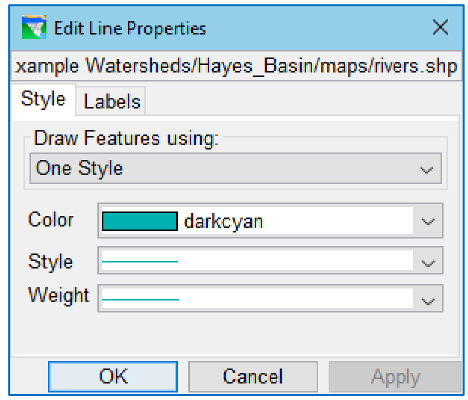


Figure B.29 Edit Line Properties Style Tab—One Style

One Style (Figure B.29)—under this option the drawing properties are:

Color—the color of the line that is drawn for each element in the shapefile.

Style—the line style, such as solid, dashed, dotted, etc.

Weight—the thickness of the line.

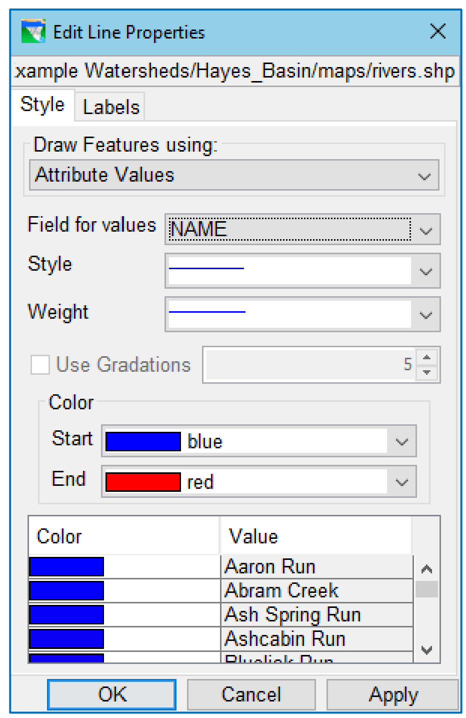


Figure B.30 Edit Line Properties Style Tab—Attribute Values

Attribute Values (Figure B.30)—under this option, the drawing properties expand to:

Field for Values—select the *attribute* whose values will determine the color of each line.

Style—the line style, such as solid, dashed, dotted, etc. used to draw each line in the shapefile

Weight—the thickness of the lines.

Color—two colors are used to define the **Start** and **End** colors of a color ramp. ResSim will determine the number of unique values in the selected attribute field, then create a gradation set of colors ranging between the **Start** and **End** colors, with a different color in the gradation for each unique attribute value. The set of colors and their associated attribute values will be displayed in the **Color-Value** table.

Polygon Shapefile Properties

If the map layer you wish to configure is a *Polygon* shapefile (e.g., lakes), selecting **Properties** for the map layer will open the **Edit Polygon Properties** editor. This editor is organized in three tabs—**Fill**, **Border**, and **Label**.

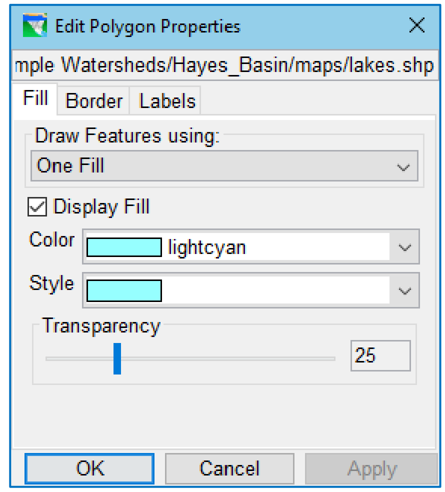


Figure B.31 Edit Polygon Properties—Fill Tab—One Fill

The **Fill** tab lets you set the fill drawing properties for the polygon elements in the shapefile. These properties are a function of the first field, **Draw Features using:**. The options are:

One Fill (Figure B.31)—under this option, the drawing properties are:

Color—the color of the interior of each polygon element in the shapefile.

Style—the fill style, such as solid, hashed, lined, etc.
Transparency—a value between 1 and 100; 1 is fully opaque; 100 is fully transparent.

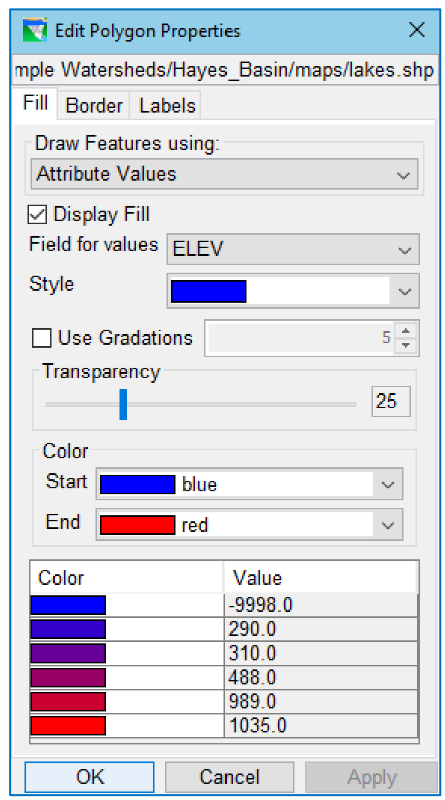


Figure B.32 Edit Polygon Properties—Fill Tab—Attribute Values

Attribute Values (Figure B.32)—under this option, the drawing properties expand to:

Display Fill—Use the checkbox to turn ON or OFF drawing the fill of the polygons in your shapefile.
Field for Values—select the *attribute* whose values will determine the fill color of each polygon
Style—the fill style, such as solid, hashed, lined, etc.
Transparency—a value between 1 and 100; 1 is fully opaque; 100 is fully transparent.
Color—the fill color properties take up the rest of the edit panel. But you really have only two color fields to set—**Start** and **End**. ResSim is going to determine the number of unique values in the

selected attribute field, then it is going to create a color gradation ranging between the Start and End colors with a different color in the gradation for each unique attribute value. The set of colors and their associated attribute value will be displayed in the **Color-Value** table.

Use Gradations—Use the checkbox to turn ON or OFF the use of gradations in the use of the color ram. If checked ON, you can set the number of gradation colors. The values of the selected attribute will then be divided into ranges, one range for each of the gradation colors.

The **Border** tab (Figure B.33) lets you set the outline drawing properties for each polygon element in the shapefile. Your options include:

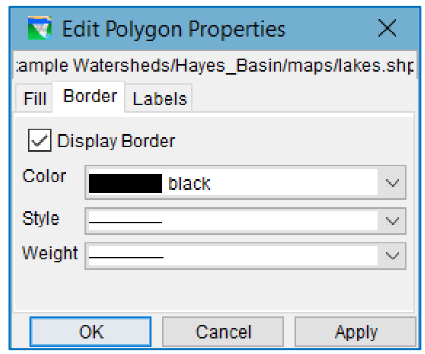


Figure B.33 Edit Polygon Properties—Border Tab

Display Border—Use the checkbox to turn ON or OFF drawing of the borders of the polygons.

Color—the color of the line that is drawn for around each polygon element in the shapefile.

Style—the line style, such as solid, dashed, dotted, etc.

Weight—the thickness of the line.

Shapefile Label Properties

The **Label** tab (Figure B.34) of each of the shapefile **Properties** editors look and function in exactly the same way. This tab lets you set whether or not the labels are drawn and with what font, size, and position.

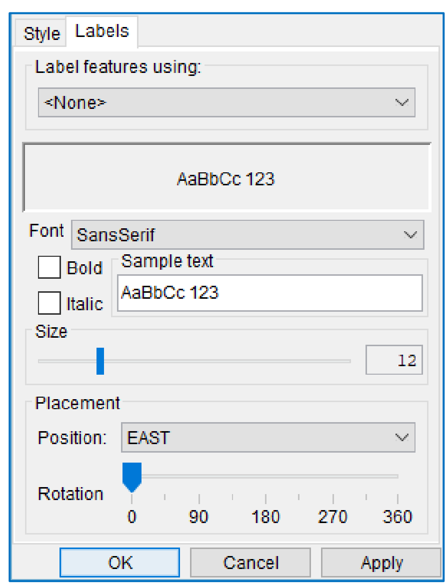


Figure B.34 Edit Shapefile Properties—Labels Tab

Label Features using:—To draw the labels, select the *attribute* whose values you want to use as the labels for element in the shapefile. To NOT draw labels for each element, select **<None>** (which will disable the selection of the rest of the label drawing properties.)

Font—select a font (typeface) to use to draw the labels

Bold—On/Off—makes the text **bold**

Italic—On/Off—makes the text *italic*

Size—the font size, in points.

Placement—select one of the eight points of the compass to determine where you want the label drawn relative to each element. For example, North would place the label centered above the element.

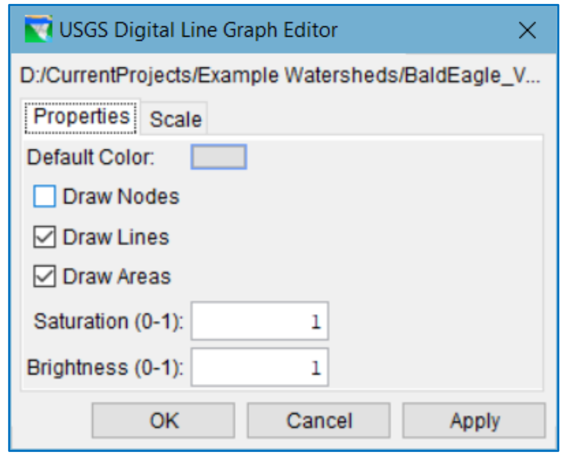


Figure B.35 USGS Digital Line Graph Editor for DLG Map Layer—Properties Tab

Rotation—select a rotation value to draw the label. Horizontal is 0. The increasing values are clockwise from the horizontal. Upside-down is 180.

All labels are drawn with respect to a label "handle" or position. In general, this handle is located near the center of the element plus a hard-coded offset; it is usually NOT obvious where the label handle is located because it can be affected by your **Position** selection. Try testing out a few **Position** and **Rotation** settings to understand how the various settings will affect where and how your labels will be drawn.

B.6.4.2 USGS Digital Line Graph Maps (*.dlg)

If the map layer you wish to configure is a USGS Digital Line Graph map (**.dlg*), selecting **Properties** from the map will open the **USGS Digital Line Graph Editor** (Figure B.35). When ResSim loads with a *dlg* file, it automatically creates a *dlgbin* file for use within the watershed. The dlgbin file contains the dlg information in binary form as well as your drawing properties.

On the **Properties** tab of the **USGS Digital Line Graph Editor**, you can modify the following drawing attributes:

Default Color—the button open a **Color Chooser** (Figure B.39) from which you can select the color you want the elements in your map drawn.

Draw Nodes,
Draw Lines,
Draw Areas—use the checkboxes to turn ON or OFF the drawing of the nodes, lines, or areas in your map.

Saturation—Control the intensity of the color used to draw the elements in your map by entering a decimal value between 0 and 1. The default value is 1.

Brightness—Control the amount of white in the color used to draw the elements in your map by entering in a decimal value between 0 and 1. The default value is 1.

On the **Scale** tab of the **USGS Digital Line Graph Editor** (Figure B.36), you can specify how the map will be displayed based on zoom level:

Always show layer—by default, the digital line graph layer is set to always display, regardless of the zoom level.

Only show layer in the following scale range—select this radio button if you want the map to display only for specific zoom ranges:

Only visible below 1: To set the scale (zoom range) so that the digital line graph layer becomes visible as you *zoom-in*, you can set the scale to the current scale by clicking **Set to Current** or enter a value in the box.

Only visible above 1: To set the scale so that the digital line graph layer becomes visible as you *zoom-out*, you can set the scale to the current scale by clicking **Set to Current** or enter a value in the box.

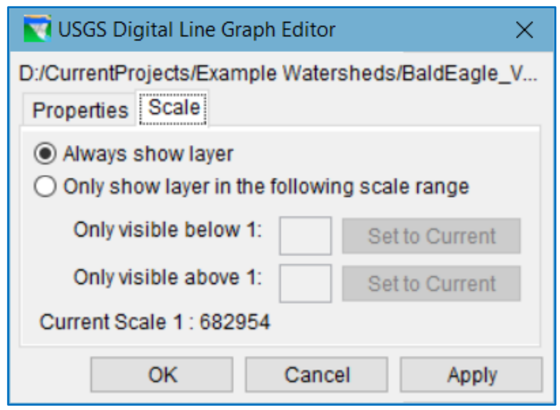


Figure B.36 USGS Digital Line Graph Editor—Scale Tab

B.6.4.3 ArcInfo® DEM, ASCII DEM, and ASCII NetTIN Maps

If the map layer you wish to configure is an ArcInfo® DEM (**.asc*), ASCII DEM (**.dem*), or ASCII NetTIN (**.net*) file, selecting **Properties** from the map opens the **Elevation Options** editor (Figure B.37). The **Elevation Options** editor is organized in two tabs—**Properties** and **Scale**.

The **Properties** tab allows you to define the properties for drawing the terrain map using color contours. The drawing properties include:

The Color Contour type—this is the selector in the upper left corner of the panel. The selection list includes: **Aspect Shading**, **Grayscale**, **Linear**, **Precipitation**, **Red-Green-Blue**, or **Terrain**.

Draw Edges—this option is specifically for ASCII NetTIN files. If checked, the edges of the triangles that make up an ASCII NetTIN file will be drawn.

Tic Interval—used for marking the color contours

Contour Limits—set the Maximum/Minimum values for **Contour Limits**. By default, Contour Limits are set to **System Specified Min/Max Values**

based on the elevation data in stored in the map. If you deselect the checkbox, you can enter custom values.

Brightness—the relative lightness or darkness of a color from 0 to 1 (black to white).

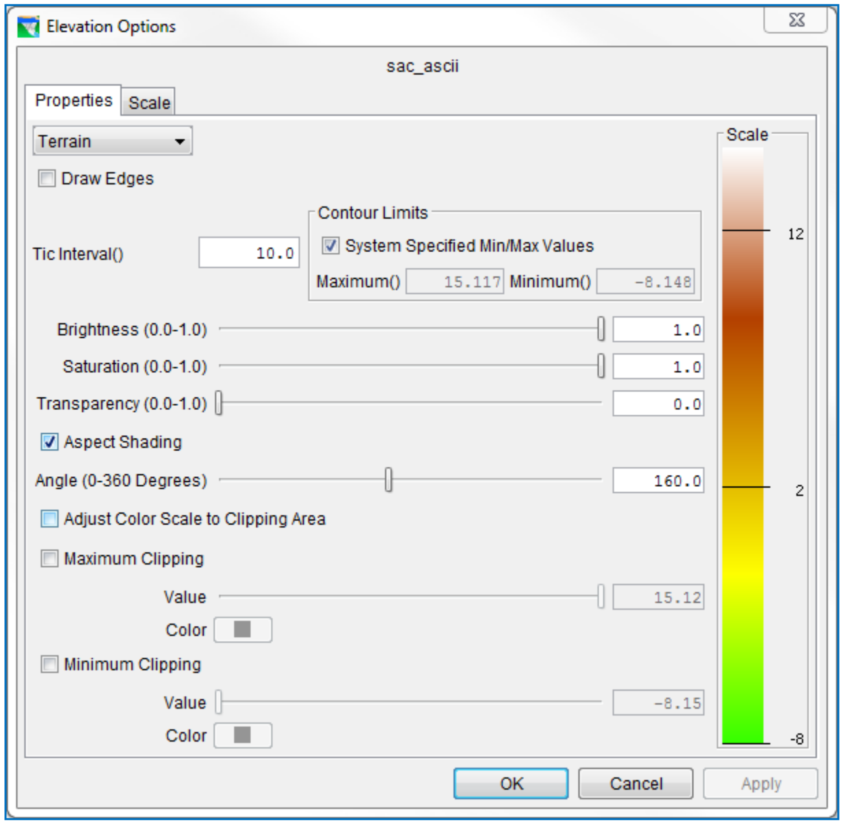


Figure B.37 Elevation Options Editor

Saturation—the intensity or density of the color, measured from 0 (grey) to 100 (vivid color).

Transparency—the level of opacity, measured from 0.0 to 1.0, of your gridded data layer.

Aspect Shading—On/Off—If you are using elevation maps, you may wish to choose **Aspect Shading** as the color contour and activate the **Aspect Shading** option. Aspect Shading uses a single color and makes the map appear in relief by placing an imaginary light source above the map and shading the elevation contours. When the **Aspect Shading** option is On, use the **Angle** slider to adjust the angle of the light source.

Maximum Clipping and **Minimum Clipping**—On/Off—by default, these options are unchecked (Off). If you activate these options, the **Value** sliders and **Color** buttons become available for setting the end points of the color ramp.

The **Value** sliders allow you to specify the amount of Clipping within the Contour Limits you have specified; you can also type values into the text boxes.

Color—these buttons open the **Color Chooser** (see Section B.8 for a description of the **Color Chooser**).

Scale—The color ramp on the right side of the **Properties** tab will reflect your drawing property settings.

On the **Scale** tab of the **Elevation Options Editor** (Figure B.38), you can specify how the map will be displayed based on zoom level:

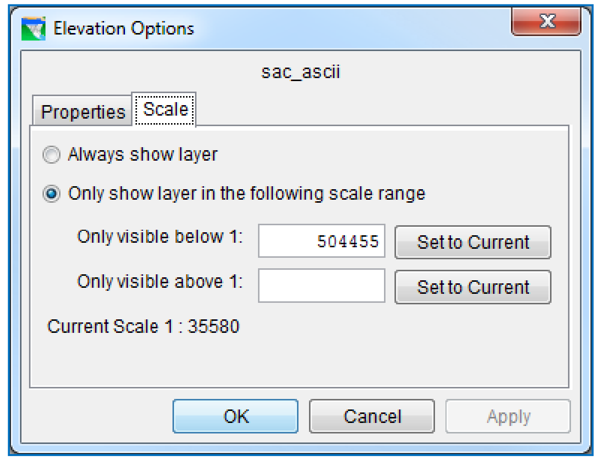


Figure B.38 Elevation Options—Scale Tab

Always show layer—by default, the digital line graph layer is set to always display, regardless of the zoom level.

Only show layer in the following scale range—select this radio button if you want the map to display only for specific zoom ranges:

Only visible below 1: To set the scale (zoom range) so that the digital line graph layer becomes visible as you *zoom-in*, you can set the scale to the current scale by clicking **Set to Current** or enter a value in the box.

Only visible above 1: To set the scale so that the digital line graph layer becomes visible as you *zoom-out*, you can set the scale to the current scale by clicking **Set to Current** or enter a value in the box.

B.7 Using the Color Chooser

Most drawing properties editors only provide a limited list of colors to choose from for setting the color of a particular layer element. However, some Properties editors let you use the **Color Chooser** to pick a color for the layer element. The **Color Chooser** affords great flexibility for select default colors. The **Color Chooser** has three tabs: **Swatches**, **HSB**, and **RGBA**, offering three methods for choosing a color. For each method, the preview area shows you how your selected color will look—review it carefully before applying your changes (clicking **OK**).

B.7.1 Swatches

To select a color from the **Swatches** tab (Figure B.39), click on the swatch of the color you want. The **Recent** panel keeps track of your choices so you can return to them easily.

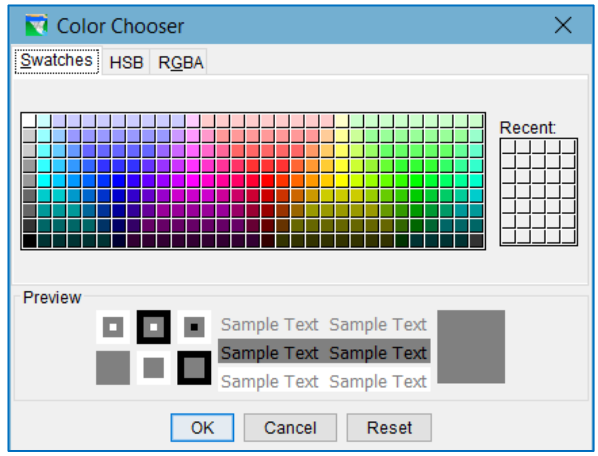


Figure B.39 Color Chooser—Swatches Tab

B.7.2 HSB Colors

HSB stands for **H**ue, **S**aturation and **B**rightness. Hue is the basic color. Saturation is how dark the color is (how much black is added). Brightness is how light the color is (how much white is added). To select a color from the **HSB** tab (Figure B.40):

- Start by selecting the **Hue**—select the **Hue** radio button then select a color by moving the rainbow slider bar in the middle of the tab panel. A value will appear in the **Hue** textbox representing your color selection and the associated values of Red, Green, and Blue that make up that hue will be shown.
- The large box to the left of the slider bar will fill to show the range of saturation and brightness you can select from for the Hue you chose. Click in the box to pick a shade of your selected Hue. The **Saturation** and **Brightness** values will update to reflect your selection. The Red, Green, and Blue fields will also update to show the equivalent RGB values for your selected color.

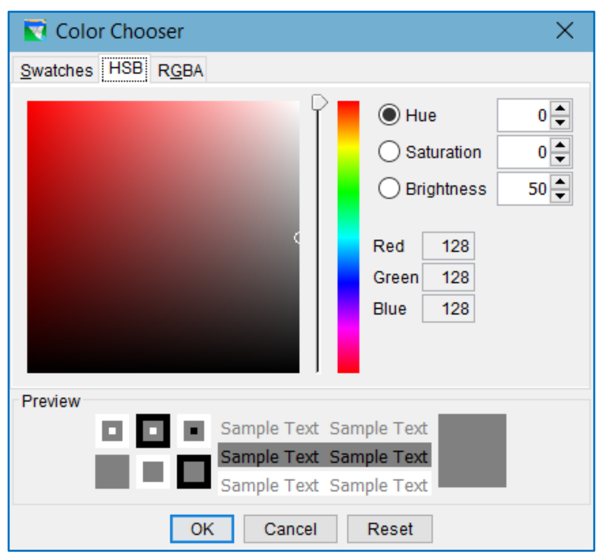


Figure B.40 Color Chooser—HSB Tab

If using Hue is not effective for finding the color you want, you can also select a color by choosing the **Saturation** or **Brightness** radio buttons and adjusting your color selection with the slider bar and the shade box.

Hue (H) is measured in a circle from 0 to 359 degrees (0=red, 60=yellow, 120=green, 180=cyan, 240=blue, 00=magenta).
Saturation (S) is the intensity or density of the hue, measured from 0 (grey) to 100 (vivid color).
Brightness (B) is the relative lightness or darkness of a color from 0 (black) to 100 (while).

B.7.3 RGBA Colors

To select a color from the **RGBA** tab (Figure B.41), use the sliders or type in values to select the **Red**, **Green**, **Blue,** and **Alpha** values to produce the color you want.

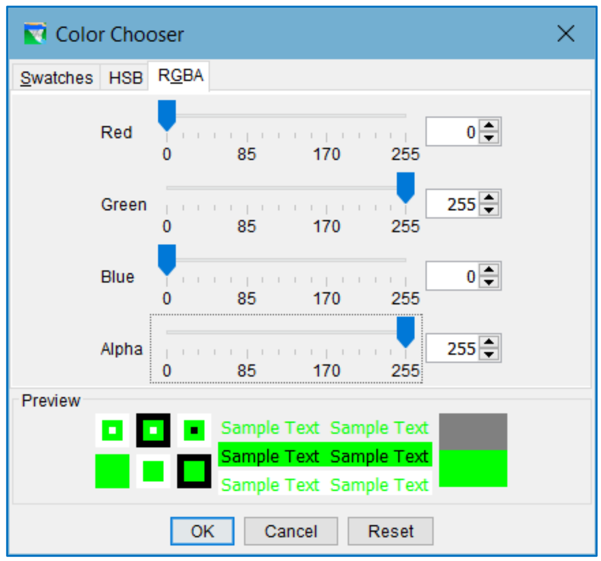


Figure B.41 Color Chooser—RGBA Tab

We all know what **Red**, **Green**, and **Blue** are, but when mixing these colors as *light*, not paint, most folks don't know how to produce certain familiar colors, so Table B.1 has some examples.

Alpha is another aspect of defining the color of light that may be unfamiliar, but it is really just *opacity* or *transparency*. The larger the alpha value, the darker (and denser) the color; the smaller the alpha value, the lighter and more transparent the color.

Table B.1 Example Colors in RGB Values

Color	**Red**	**Green**	**Blue**
Black	0	0	0
Med. Grey	128	128	128
White	255	255	255
Yellow	255	255	0
Green	0	128	0
Lime Green	0	255	0
Cyan/Aqua	0	255	255
Blue	0	0	255
Navy	0	0	128
Maroon	128	0	0
Red	255	0	0
Purple	128	0	128
Magenta/Fuchsia	255	0	255

B.8 Creating User Toolbar Buttons

BLUF (Bottom Line Up Front)

The results of recent testing suggest that *the User Toolbar Buttons may be more trouble than they are worth*. These buttons seem to have more scope than they should, i.e., when added to one watershed, they may show up in other watersheds. You may find yourself Removing the Toolbar Buttons repeatedly from both the original watershed as well as watersheds that shouldn't show them. And, the Reload Default Button function doesn't appear to clear them out everywhere or permanently.

A **User Toolbar Button** acts as a shortcut for controlling the view of a layers in your **Map Display** without having to open the **Layer Selector**. These buttons are useful when you need to have frequent control over layers that are being displayed. **User Toolbar Buttons** are module specific, which means that if you create a button in the **Watershed Setup** module, it will not appear in the Simulation module unless you create it (again) in that module.

Clicking a **User Toolbar Button** turns the associated layer ON or OFF in the **Map Display**, in much the same way as the checkbox in the **Layers** Tree in the **Layer Selector**.

Right-clicking on a toolbar button opens a context menu that gives you access to the layer's properties editor and the ability to select and deselect sub-layers to be displayed. The options available from a toolbar button's context menu depend upon the associated layer's layer type.

Hovering your cursor over a toolbar button displays a tooltip with a description of the button or the layer it is associated with.

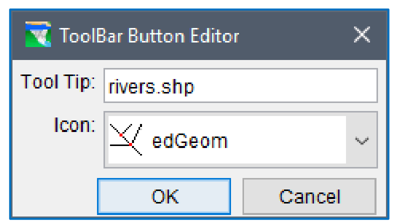


Figure B.42 Toolbar Button Editor

To *create* a toolbar button:

- Open the **Layer Selector** and select a layer in the **Layers** tree.
- From the **Edit** menu, select **Add Toolbar Button**. The **Toolbar Button Editor** (Figure B.42) opens.
- The layer's name appears in the **Tool Tip** field, but you can enter a more descriptive tooltip if desired.
- Select an icon from the **Icon** list to display on the button. If you forget to select an icon, a toolbar button will *not* be created.
- Select **OK** to finish creating the Toolbar Button. **Cancel** will abort the process.

The toolbar button will appear in the Module toolbar above the Map Display Area as shown in Figure B.43.

To *remove* a toolbar button:

- Select the layer in the **Layers** Tree of the **Layer Selector**
- From the **Edit** menu, select **Remove Toolbar Button**.

To *remove all* toolbar buttons that you have defined:

- Select **Reload Default Button** from the **Edit** menu in the **Layer Selector**.

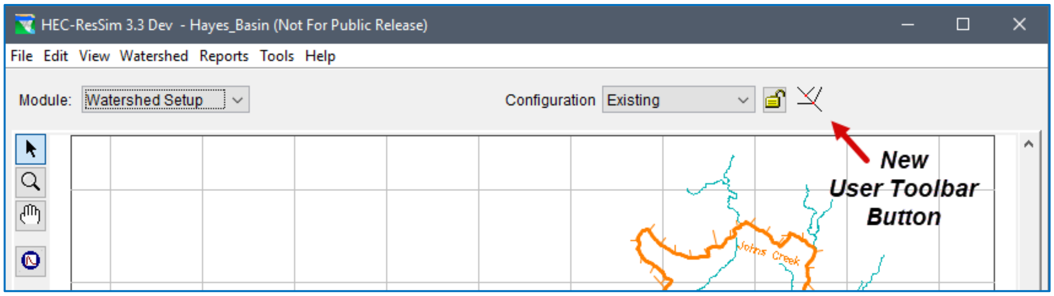


Figure B.43 User Toolbar Button Added to Main ResSim Window

B.9 Defining the Watershed Coordinate System

In Chapter 3, the section titled "Setting the Geographic Coordinate System" explains how to specify the coordinate system of your watershed by accessing the **Display Coordinate Information**. This Appendix section provides more detailed information related to setting up the coordinate system for your watershed.

B.9.1 Accessing Display Coordinate Information

The **Display Coordinate Information** editor is accessible from the **Layer Selector** which can be opened from any of the three ResSim modules. To open it:

- Select **Layers** from the **View** menu in the Main ResSim window. The **Layer Selector** will open
- Select Map Display Coordinates from the Maps menu of the Layer Selector.
- In the Display Coordinate Information editor, select Edit.
- The **Map Coordinate Information** editor will appear where you can set the appropriate coordinate information.

The above steps are illustrated in Figure B.44.

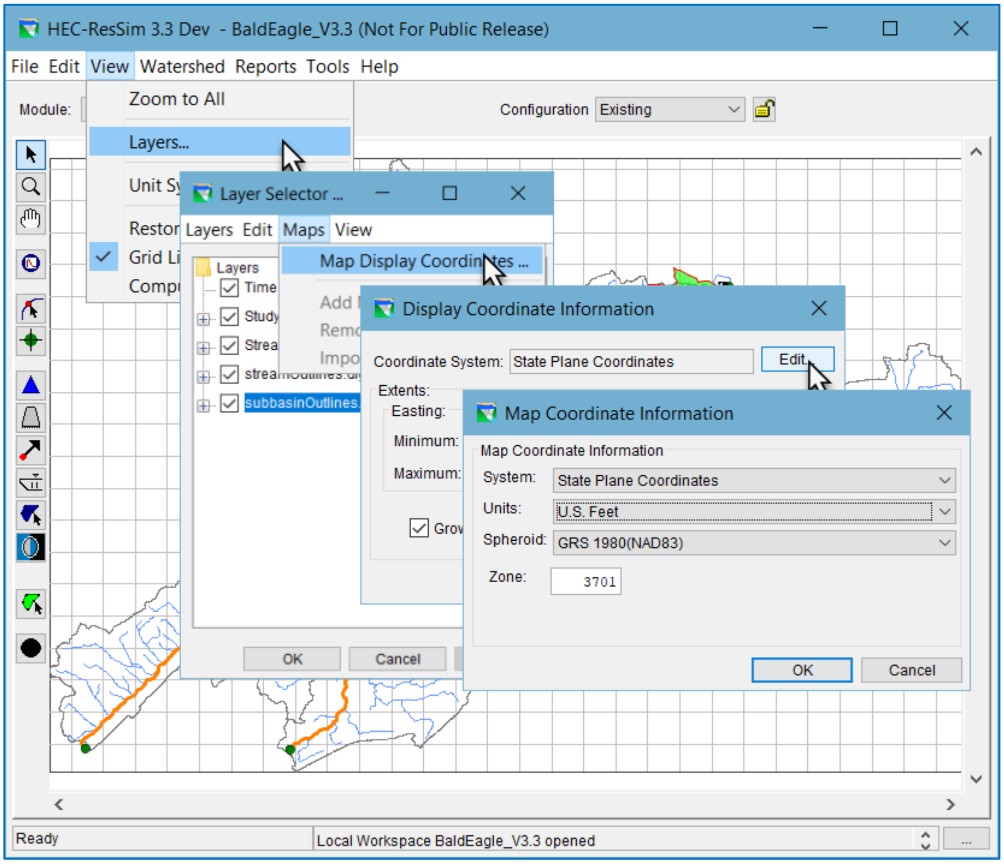


Figure B.44 Steps for Accessing the Coordinate System

Table B.2 shows the options available for **System**, **Units**, and **Spheroid**. When you choose specific coordinate systems, not all options are available from the **Spheroid** list. For example, choosing **X-Y** from the **System** list deactivates the **Spheroid** list. Additionally, when **State Plane Coordinates** is selected as the **System**, only **Clarke 1886** and **GRS 1980** are available choices from the **Spheroid** list.

Table B.2 Available Map Coordinate Systems, Units, and Spheroid Options

System	Units	Spheroid
X-Y	U.S. Feet Meters International Feet	*X-Y does not allow selection of Spheroid*
Geographic	Radians Seconds of Arc Degrees of Arc	• Clarke 1866 (NAD27) • WGS 72 • GRS1980 (NAD83) • WGS84 • Sphere of Radius 6371200 Meters
Universal Transverse Mercator(2)	U.S. Feet Meters International Feet	• Clarke 1866 (NAD27) • WGS 72 • GRS1980 (NAD83) • WGS84 • Sphere of Radius 6371200 Meters
State Plane Coordinates(3)	U.S. Feet Meters International Feet	• Clarke 1866 (NAD27) • GRS1980 (NAD83)
Albers Equal-Area Conic(4)	U.S. Feet Meters International Feet	• Clarke 1866 (NAD27) • WGS 72 • GRS1980 (NAD83) • WGS84 • Sphere of Radius 6371200 Meters
Lambert Conformal Conic(4)	U.S. Feet Meters International Feet	• Clarke 1866 (NAD27) • WGS 72 • GRS1980 (NAD83) • WGS84 • Sphere of Radius 6371200 Meters
Transverse Mercator (5)	U.S. Feet Meters International Feet	• Clarke 1866 (NAD27) • WGS 72 • GRS1980 (NAD83) • WGS84 • Sphere of Radius 6371200 Meters
Albers Equal-Area Conic (SHG)	U.S. Feet Meters International Feet	• Clarke 1866 (NAD27) • WGS 72 • GRS1980 (NAD83) • WGS84 • Sphere of Radius 6371200 Meters
Polar Stereographic (HRAP)	U.S. Feet Meters International Feet	• Clarke 1866 (NAD27) • WGS 72 • GRS1980 (NAD83) • WGS84 • Sphere of Radius 6371200 Meters

(2) Universal Transverse Mercator System also requires a UTM Zone to be entered

(3) State Plane Coordinates also requires a numeric Zone to be entered (e.g., 3701)

(4) Albers Equal-Area Conic and Lambert Conformal Conic also require the following: •Latitude for the first and second standard parallel, •Longitude of the central meridian, •Latitude of the projection origin, •False easting and northing

(5) Transverse Mercator also requires the following entries:

Scale factor and Longitude of the central meridian, Latitude of the projection origin, False easting and northing.

B.9.2 Coordinate Systems Options

The following sections describe each of the Coordinate Systems that are available from the **Map Coordinate Information** editor. With one exception, all require you to select distance **Units** and a **Spheroid** definition. Each spheroid definition inherently describes a prime meridian and a datum.

The options for distance **Units** include:

- (U.S. Feet
- Meters
- International Feet

The options for **Spheroid** definition include:

- Clark 1866(NAD27)
- WGS 72
- GRS 1980(NAD83)
- WGS 84
- Sphere of Radius 6371200 Meters

B.9.2.1 X-Y System

X-Y is a simple Cartesian coordinate system (Figure B.45).

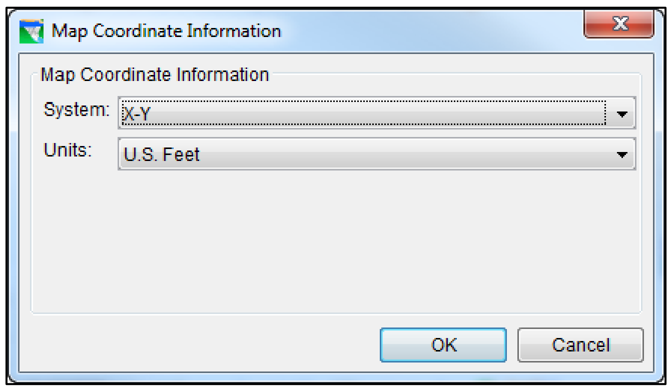


Figure B.45 X-Y Coordinate System

B.9.2.2 Geographic System

A **Geographic** coordinate system (Figure B.46) defines locations on the earth using a three-dimensional spherical surface. This coordinate system differs from the others in that the **Units** are not of distance but rather angle. The **Units** options include:

- Radians
- Seconds of Arc
- Degrees of Arc

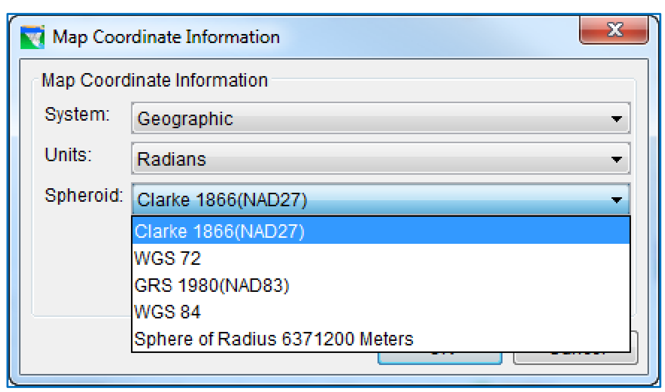


Figure B.46 Geographic Coordinate System—Spheroid List

B.9.2.3 Transverse Mercator System and Universal Transverse Mercator (UTM) System

The **Transverse Mercator** coordinate system represents the earth as a spheroid that has been flattened onto a transverse (sideways) cylinder such that each longitude line is equally spaced and parallel to one another. Small shapes are well represented because this projection preserves angular relationships, but area becomes increasingly distorted the further you get from the equator (toward the Polar Regions).

The **Universal Transverse Mercator** (**UTM**) coordinate system is a special implementation of the Transverse Mercator system that divides the globe into a specific set of 60 6° north-south segments or zones. The limits of each zone extent to 84 N and 80 S. Each zone is divided into a north and south half at the equator. The Polar Regions use the Universal Polar Stereographic coordinate system.

For the **Transverse Mercator** coordinate system (Figure B.47), you must choose both the **Units** (U.S. Feet, Meters, or International Feet) and **Spheroid**.

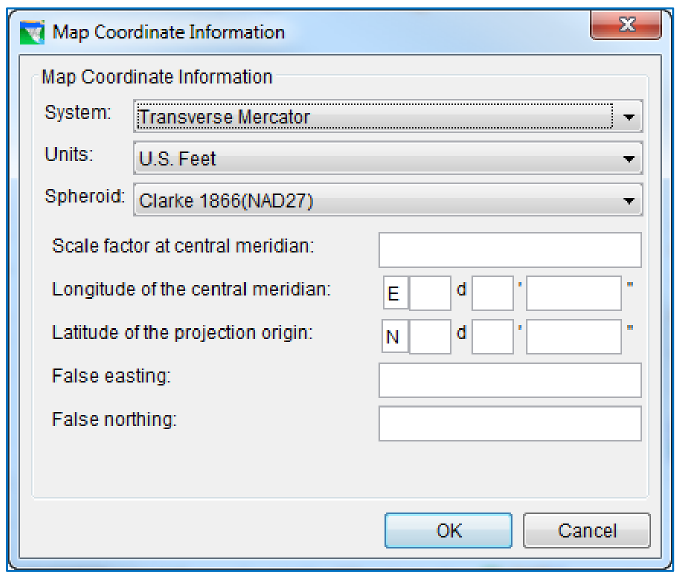


Figure B.47 Map Coordinate Information Transverse Mercator System

Additional required information includes: Scale factor at central meridian; Longitude of the central meridian; Latitude of the projection origin; False easting; and False northing.

When specifying latitudes, use **N** or **S**. For longitudes, use **E** or **W**. You may either type these into the fields or use the SPACEBAR on your keyboard to toggle between the acceptable entries (after placing your cursor in the first box of either the latitude or longitude field).

The **Universal Transverse Mercator** system (Figure B.48) is a specialized form of the Transverse Mercator projection. It uses a simple Cartesian coordinate system to represent locations on the earth. It represents the earth as a spheroid divided into 60 6° north-south segments or zones, flattens (transforms) each segment onto a transverse (sideways) cylinder, and applies an X-Y coordinate system to it. To use this system for your watershed, you must select:

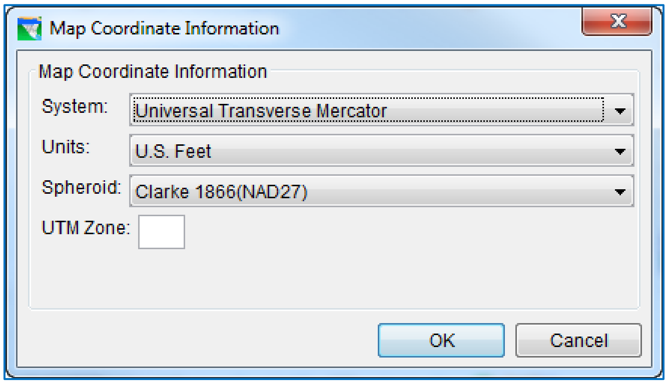


Figure B.48 UTM Coordinate System

- Distance **Units** (U.S.)
- **Spheroid** definition
- UTM Zone

B.9.2.4 State Plane Coordinates System

The **State Plane** coordinate system (Figure B.49) is a set of 124 geographic zones covering the 50 US states and US territories. It uses a simple Cartesian coordinate system to specify locations rather than a more complex spherical coordinate system. This system requires that you select:

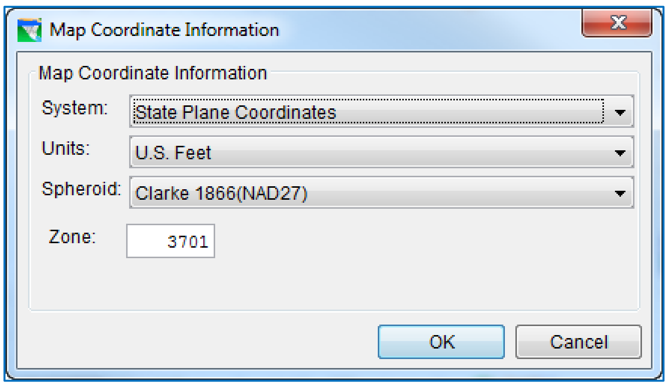


Figure B.49 State Plane Coordinate System

- Distance **Units**
- **Spheroid** definition:
  - Clarke 1866 (NAD27)
  - GRS 1980(NAD83)
- Zone

B.9.2.5 Albers Equal-Area Conic System and Lambert Conformal Conic System

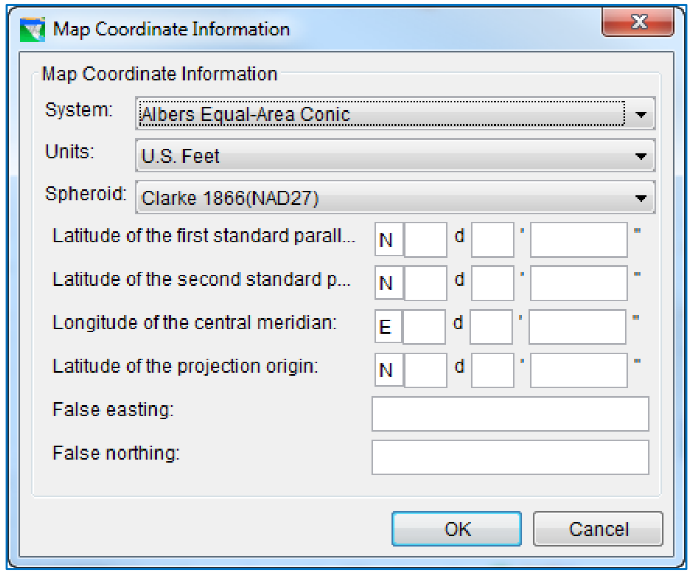


Figure B.51 Map Coordinate Information Lambert Conformal Conic System

The **Albers Equal-Area Conic** coordinate system (Figure B.50) is based on a projection of a horizontal section of the earth onto a cone placed directly above the pole. The section of the earth that is projected is defined by two standard parallels—circles around the earth parallel to the equator. The Albers projection preserves the area of each section at the expense of a small distortion of distance (shape).

Like Albers, the **Lambert Conformal Conic** coordinate system (Figure B.51) is based on a projection of a horizontal section of the earth onto a cone placed directly above the pole. The **Lambert Conformal Conic** differs from the **Albers Equal Area Conic** projection because it preserves distances (shape) within each section at the expense of a small error in area.

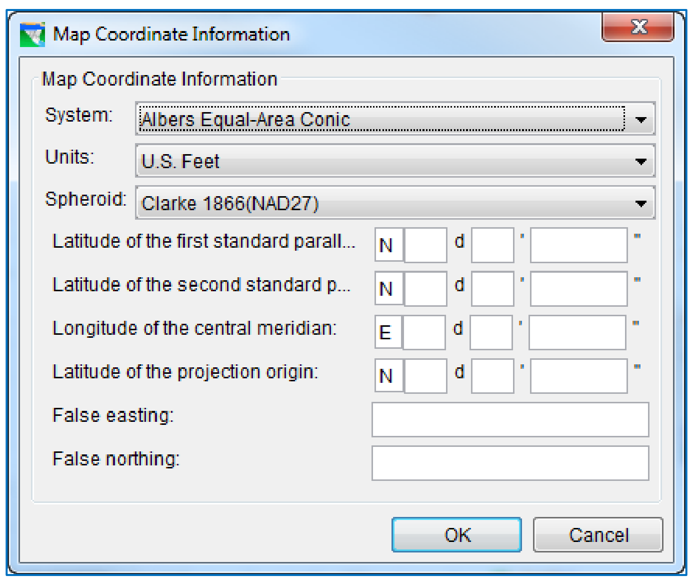


Figure B.50 Albers Equal-Area Conic Coordinate System

These projections are best for regions that are predominately east-west in orientation and are best applied in the middle latitudes (the zones nearer the equator). The difference between the first and second parallels should not exceed 30-35 degrees.

To use one of these coordinate systems for your watershed, you must specify quite a few parameters:

- Distance **Units**

- **Spheroid** definition
- Latitude of the two standard parallels
- Longitude of the central meridian
- Latitude of the projection origin
- **False easting**—a value for the origin of the x-coordinates
- **False northing**—a value for the origin of the Y-coordinates

When specifying latitudes, use **N** or **S**. For longitudes, use **E** or **W**. You may either type N, S, E, or W into the fields, or use the SPACEBAR on your keyboard to toggle through the acceptable entries (after placing your cursor in the first box of either the latitude or longitude field).

B.9.2.6 Albers Equal-Area Conic (SHG) System

The **SHG** version of the **Albers Equal-Area Conic (SHG)** coordinate system (Figure B.52) displays a set of default parameters. These parameters cannot be changed. If you must change them, use instead the **Albers Equal Area Conic** system described above.

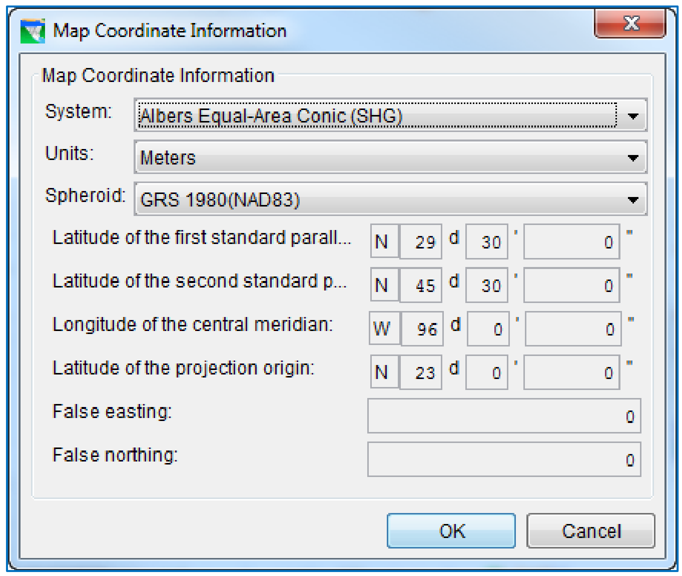


Figure B.52 Map Coordinate Information Albers Equal-Area Conic (SHG) System

B.9.2.7 Polar Stereographic (HRAP) System

The **Polar Stereographic (HRAP)** coordinate system (Figure B.53) is used by the National Weather Service for describing their NEXRAD precipitation grids. The editor for this coordinate system shows the default parameters that define the HRAP system. These parameters cannot be changed.

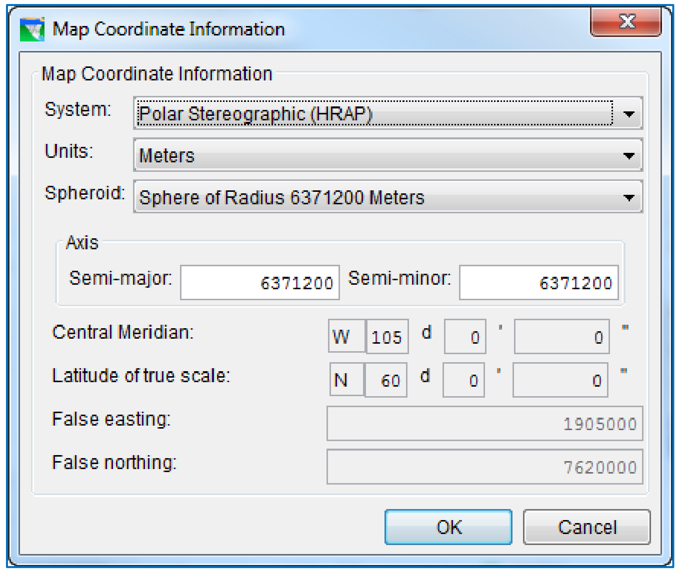


Figure B.53 Map Coordinate Information Polar Stereographic (HRAP) System

Appendix C
Commonly Used Editors and Dialogs

There are a few editors and dialogs that are used repeatedly throughout the ResSim user interface. These editors have the same purpose or provide the same functionality for a variety of model features. To minimize repetition in the manual, the details of these editors and dialogs are described here.

C.1 File Browser Dialog

A file browser dialog may be used whenever ResSim needs to know a filename or a directory or folder location.

The file browser can come in a variety of shapes, sizes, and views, but they all have essentially the same purpose—to help you identify and select a file (or folder) on your computer.

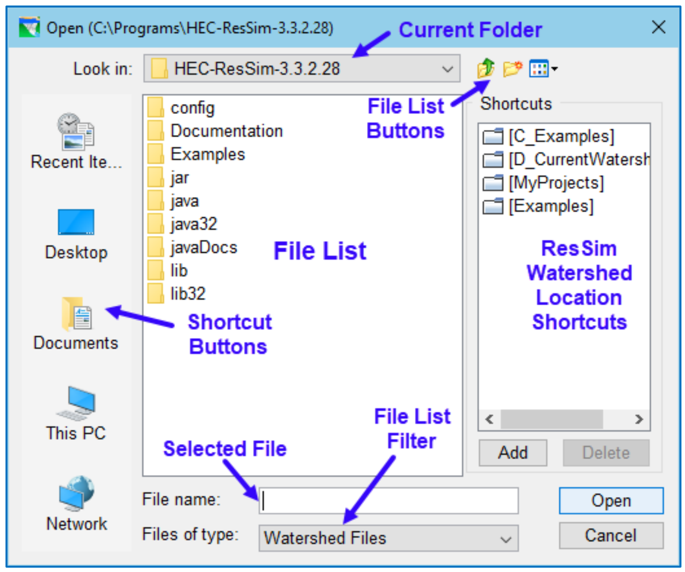


Figure C.1 File Browser Dialog—Open Watershed

Figure C.1 shows the **Open** file browser that is opened when you select **Open Watershed...** from the main **File** menu in ResSim. This browser includes a **Shortcuts** menu on the right. These shortcuts are ResSim watershed locations. You can create a shortcut right there in the browser (with the **Add** button) or in the **Options Editor** described in Appendix A, Section A.1.1.

Figure C.2 shows the **Save** file browser that is opened when you **Export** a report or table. This is probably the most generic of the file browsers.

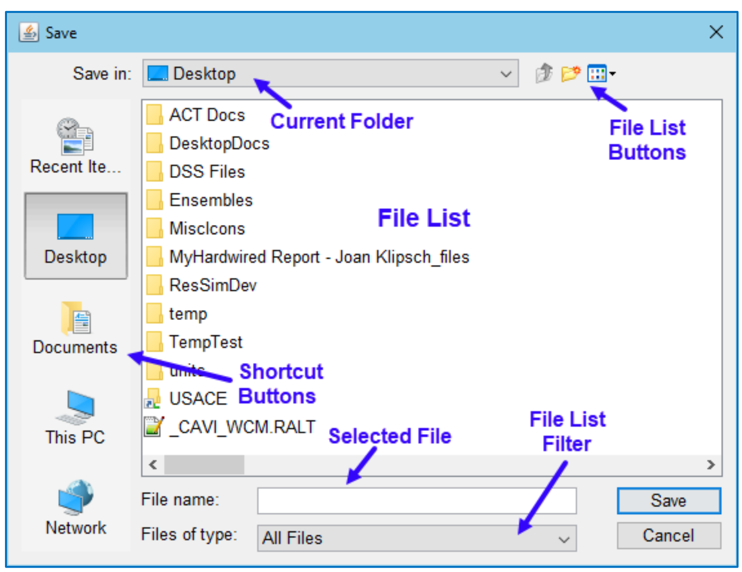


Figure C.2 File Browser Dialog—Export Report

Figure C.3 show the file browser that is opened when you select **Add Map…** from the **Maps** menu in the Layer Selector.

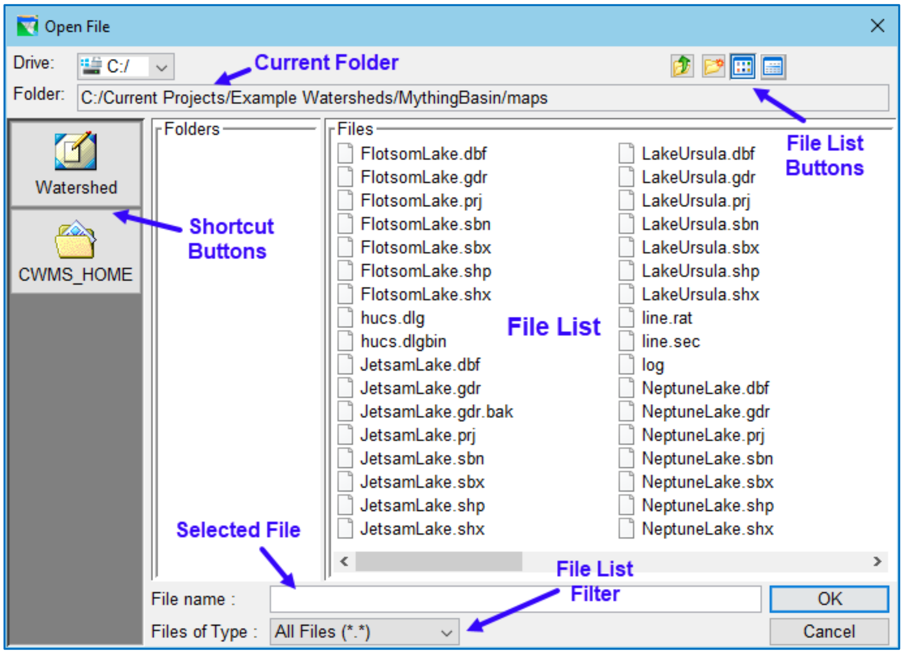


Figure C.3 File Browser Dialog—Open Map

The various file browsers share several common features:

C.1.1 Folder Shortcut Buttons

On the left side of each of these browser dialogs is a list of shortcut buttons (Icons). These buttons will change the current folder in the browser to the folder identified by that button. In the first two figures, these buttons reflect the same shortcuts you would find a Windows® file browser. In the third figure, the buttons reflect the two important folders to a CWMS watershed. This is an older version of the file browser that originated in the code that ResSim shares with CWMS.

C.1.2 Current Folder

The current folder is displayed in the field at the top of the dialog. Each browser that opens will open to its own default folder and that folder name will be displayed here.

C.1.3 File List

The center of the browser dialog is the list of files and folders in the current folder. In most of the browsers, the files and folders are listed together. In older file browsers, the folders are listed in a separate pane to the left of the list of files. Use the file (and/or folder) list to find the file or folder you need.

To change the current folder to a folder shown in the file (or folder) list, *double-click* on it.

To change the current folder to the parent folder of the current folder, use the **Move Up** file button.

To select a file, *click* on it in the file list; the filename will appear in the selected file field.

C.1.4 File List Buttons

To the right of the current folder field are a set of small (icons) buttons for managing the current folder and for how the list of files are displayed. These buttons access file and folder utility functions. The typical File List buttons you may find in a file browser include:

 Move Up—change to the parent of the current folder

 New Folder—create a new folder in the current folder

 File Names—the file list should show only the file names

 File Details—the file list should show the file names, size, and modified date.

C.1.5 File List Filter

Each file browser will have its own potential list of file type filters. The **All Files** option will be in the filter lists and equates to "no filter" so that all files in the current folder are lists. The default filter in each browser will be the file type that the specific browser was intended to help you locate and select. For example, the **Open Watershed** file browser defaults to **Watershed Files**, which means that it is filtering for files with the extension: *.wksp*.

C.1.6 Selected File

The field near the bottom of the browser labeled **File Name:** is the file that the browser thinks you selected (*click*ed on) in the File List. Verify the file name before you click **OK** to accept the file and close the Browser.

C.2 The Independent Variable Definition Dialog

The Independent Variable Definition dialog is used anywhere a model element allows a functional relationship (lookup table) to be defined as "a function of" one of a set of options including: Date, Date and Time, Model Variable, External Variable, State Variable, and sometimes even Two Variables, depending on the element and feature. The three most common forms of the **Function of:** option list are shown in Figure C.4.

Each option for the **Function of:** selector and its edit panel in the Independent Variable Definition dialog is described in the following sections.

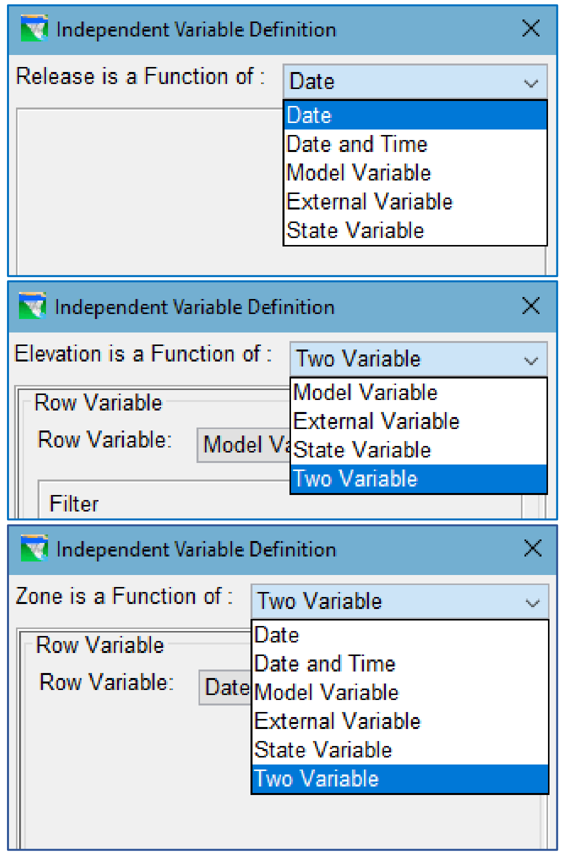


Figure C.4 Option Lists for the Independent Variable Definition Dialog

C.2.1 Function of: Date

Use **Date** if the lookup table or relationship you need to define is ***constant*** or ***varies as a function of the time of year*** **or season**.

The independent variable **Date** needs no further definition so its edit panel in the **Independent Variable Definition** dialog is blank.

After you select **Date** and *click* **OK** to accept that selection and close the **Independent Variable Definition** dialog, a seasonal function table will appear in the editor you came from. The seasonal function table will have **Date** in the header of the first column and the first cell below the **Date** header will contain the value **01Jan**. Any additional dates needed for your function should be entered in the form DDMMM, where DD is a two-digit day of month and MMM is a three-character abbreviation for the month. **Function-of-Date** relationships in ResSim repeat annually and always begin on 01 January, so no year specification is included in the date. If your relationship is constant, only one row of the table is required—just enter your constant value in the 01Jan row.

All seasonal tables in ResSim treat each date entered as the beginning of the day, i.e., as applying at 0000 hours. If you would prefer that your dates in the Function-of-Date table applied at the end of the day, use the *next* day in the table or define your relationship (table) as a Function-of-Date-and-Time.

C.2.2 Function of: Date and Time

A **Function of Date and Time** relationship (lookup table) is almost exactly the same as a **Function of Date** relationship—with one difference: the addition of a **Time** column in the seasonal function table. Use this function type when the demand varies with *both* date and time. When entering time values, use a 24–hour clock, e.g., 6 pm is 1800.

C.2.3 Function of: Model Variable

Model Variables are the variables that ResSim calculates for each element property that it needs or uses in its standard computations. A value for each model variable is computed for each timestep of the simulation and stored in a time-series object. For example, pool elevation is a property of a reservoir that ResSim uses to determine release capacity, but it must be computed each timestep to account for changes that result from the release decisions ResSim makes.

Use **Model Variable** if the lookup table or relationship you need to define ***varies as a function of a variable that is computed by ResSim***.

When **Model Variable** is selected, the edit panel of the **Independent Variable Definition** dialog (Figure C.5) will display the Model Variable pane; this pane contains the user interface widgets needed to define the model variable to be used as the independent variable for your function.

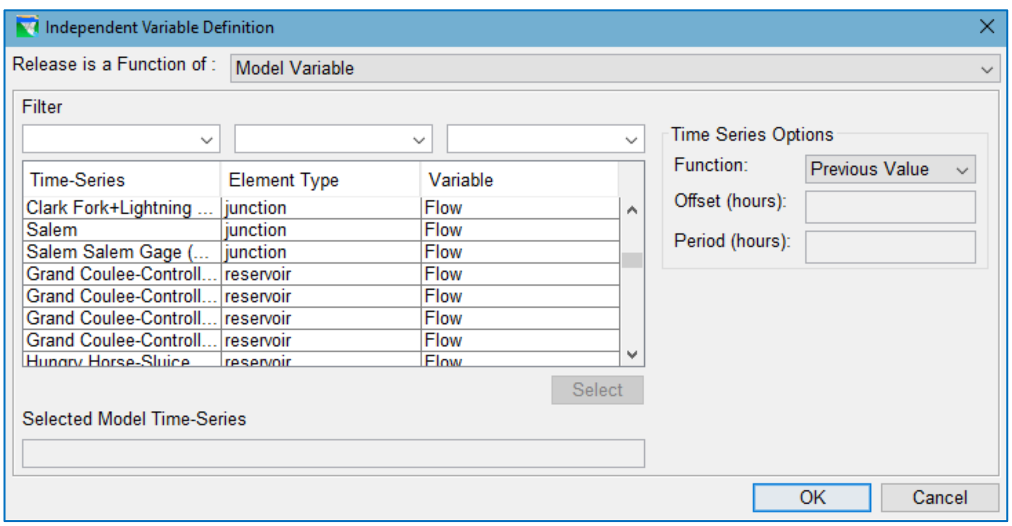


Figure C.5 Independent Variable Definition Dialog—Model Variable

The Model Variable pane includes:

A Model Variables table—a scrollable list of all the model variables computed by ResSim *for the current network*.

A set of three **Filters**—use one or more of these selectors, located above the Model Variables table, to thin out the list of model variables so that you can quickly find the variable you need. Each filter selector contains a list of all the possible values for its associated column in the variable table.

Tip

Although it is natural to use the first filter first, try using the second filter first. You'll probably find what you are looking for a lot faster.

When you select a value in one of the **Filter** selectors, the model variables table will show only those variables that have the selected value for that column of the table. For example, if you select **reservoir** in the second filter, then the table will show only those model variables that are computed for the reservoirs in your network. Also, when one (or more) **Filters** has a selected value, the list of values in the other **Filters** will change to reflect the new, shorted list of variables shown in the table.

A **Select** button—when you find the model variable you need, highlight it (*click* on it) in the table then click the **Select** button.

Note: *double-clicking* on the variable you want will NOT perform the selection; you must use the Select button.

A **Selected Model (Variable) Time-Series** textbox—The name of the model variable you selected from the table will appear here when you press the **Select** button. Be sure your selected model variable is shown in this field before you click the **OK** button to apply your selection.

A set of **Time Series Options** (Figure C.6)—Model variables are time-series data; during the compute, a value reflective of the state of that model variable is used to look up the dependent variables in the relationship you are defining. The Time Series options allow you to specify how the value of the model variable is to be selected or computed from the model variable time series. The time series options are specified as a combination of the Function and, if needed, the offset and or period used by the function. The Function are all used to identify or compute a value from the time series relative to the current timestep, so think of the time of the current timestep as the base time or *origin*. For example, if you selected **Previous Value** for the **Function**, the value returned from the model variable and used in the current timestep to lookup the value of the dependent variable from your function will be from the end of the previous timestep. The **Time Series Options** include:

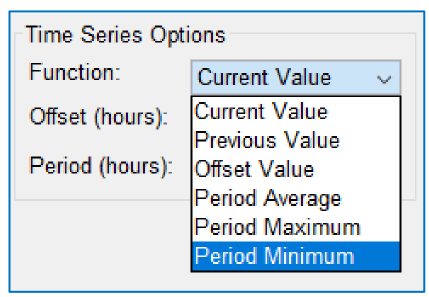


Figure C.6 Time Series Options

Function—this is how the value for the selected model variable is to be selected or computed from the model variable time series. Options include:

Current Value—returns the value stored in the model variable for the current timestep. Be careful, some model variables (like elevation) haven't been computed yet for the current timestep when the value is needed by the feature that uses the lookup table.

Previous Value—returns the value stored in the model variable at the end of the previous timestep.

Offset Value—returns the value stored in the model variable at a time offset from the current timestep's time. This function activates the **Offset** field; you may enter a positive or negative value, in HOURS, for the offset.

For example, if you enter an offset of 6 hours, the value obtained from the time series will be at time = current time + 6 hours.

Period Average, **Period Maximum**, **Period Minimum**—these functions will compute a value (average, maximum, or minimum) for the period of time specified in the time series relative to the current timestep's time. Both the **Period** and **Offset** fields are active for these functions.

For example, if you want a 6 hour average computed from the time series data offset 3 hours back from the current timestep, you should select **Period Average** for the Function and enter -3 for the Offset and 6 for the period. These entries will return the average value of the model variable from the period -9 hours to -3 hours relative to the current timestep.

Offset—The offset adjusts the origin time and may be entered as a positive or negative value. A positive offset will obtain a value from the time series at a time forward from the time of the current timestep. A negative offset will look backward from the current timestep. Note—the offset is entered in units of hours, not timesteps; take this and the compute interval (timestep) into account when specifying the offset.

Period—The period is used to specify a range of time *back* from the origin over which a value is to be computed. Only positive values can be used to specify the period. Note—the period is entered in units of hours, not timesteps; take this and the compute interval (timestep) into account when specifying the period.

C.2.4 Function of: External Variable

Use **External Variable** if the lookup table or relationship you need to define *varies as a function of a variable whose values are stored in a time series that you will provide to as input to the model*.

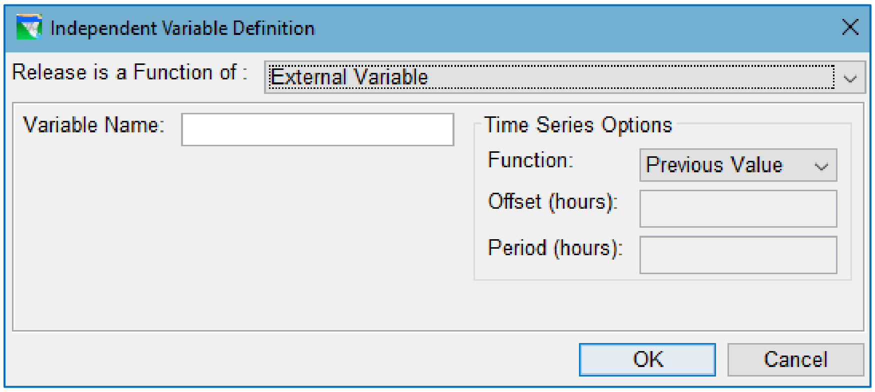


Figure C.7 Independent Variable Definition Dialog—External Variable

When **External Variable** is selected, the edit panel of the **Independent Variable Definition** dialog will display the **External Variable** pane; this pane contains:

A **Variable Name** text field—The name you enter will appear in the **Alternative Editor's Time Series** tab; this is where you will identify the HEC-DSS time-series dataset that represents the External Variable. Enter an appropriately descriptive name in the **Variable Name** field so that you will know what time-series to use for each External Variable you create in your model.

A set of **Time Series Options**—these options apply to the input time series representing the external variable and are described in detail in Section C.2.3 above.

C.2.5 Function of: State Variable

Use **State Variable** if the lookup table or relationship you need to define *varies as a function of a state variable.* State variables are variables that you create to represent the state of an abstract property or parameter that is not computed natively by ResSim. The value of the state variable is computed each timestep through execution of a Jython script that you write and save as part of the reservoir network.

When **State Variable** is selected, the edit panel of the **Independent Variable Definition** dialog will display the **State Variable** pane (Figure C.8). This pane looks and behaves like the **Model Variables** pane except it does not include **Filters** for the table; since the list of state variables is usually a lot smaller than the list of model variables, the filters are not needed.

The **State Variables** pane includes:

A State Variables table—a scrollable list of all the state variables defined in the current reservoir network.

A **Select** button—when you find the state variable you need in the table, highlight it (*click* on it) in the table then *click* the **Select** button.

Note: *double-clicking* on the variable you want will NOT perform the selection; you must use the Select button.

A **Selected Model Variable Time-Series** textbox—The name of the model variable you selected from the table will appear here when you press the **Select** button. Be sure your selected model variable is shown in this field before you *click* the **OK** button to apply your selection and close the **Independent Variable Definition** dialog.

A set of **Time Series Options**—these options apply to the input time series representing the external variable and are described in detail in Section C.2.3 above.

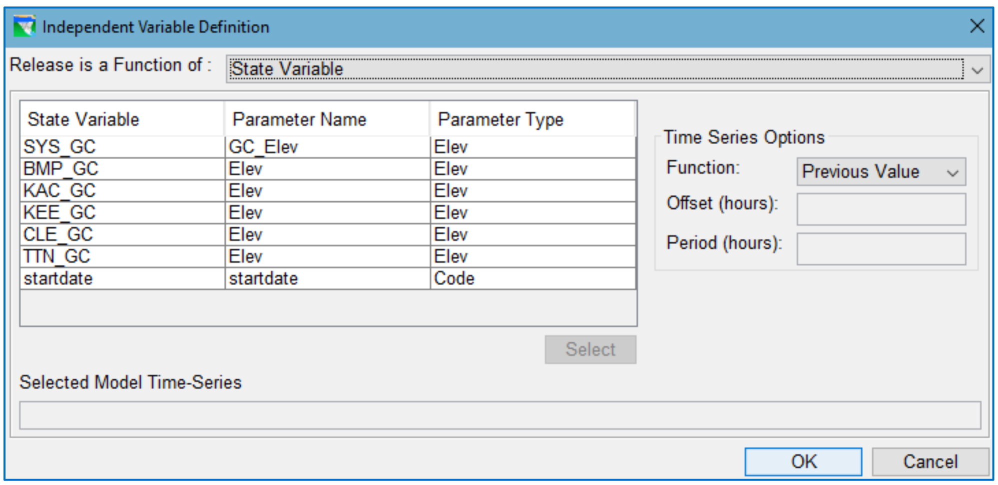


Figure C.8 Independent Variable Definition Dialog—State Variable

C.2.6 Function of: Two Variables

The Function of **Two Variables** option is available when creating a function that defines a zone, stage, or elevation. Use this option if the lookup table or relationship you need to define *varies as a function of two independent variables.* For example, stage at a location may be impacted by both the flow and the backwater of a downstream pool. In such a case, you may define the stage as a function of both flow and elevation. This will allow you to build a two-variable data table to represent your function, as shown in the example in Figure C.9. The independent variables are expressed as a **Row Variable** and a **Column Variable**. The Row Variable will be displayed in the left-most column of the data table and vary by row. The column variable varies per column of the data table.

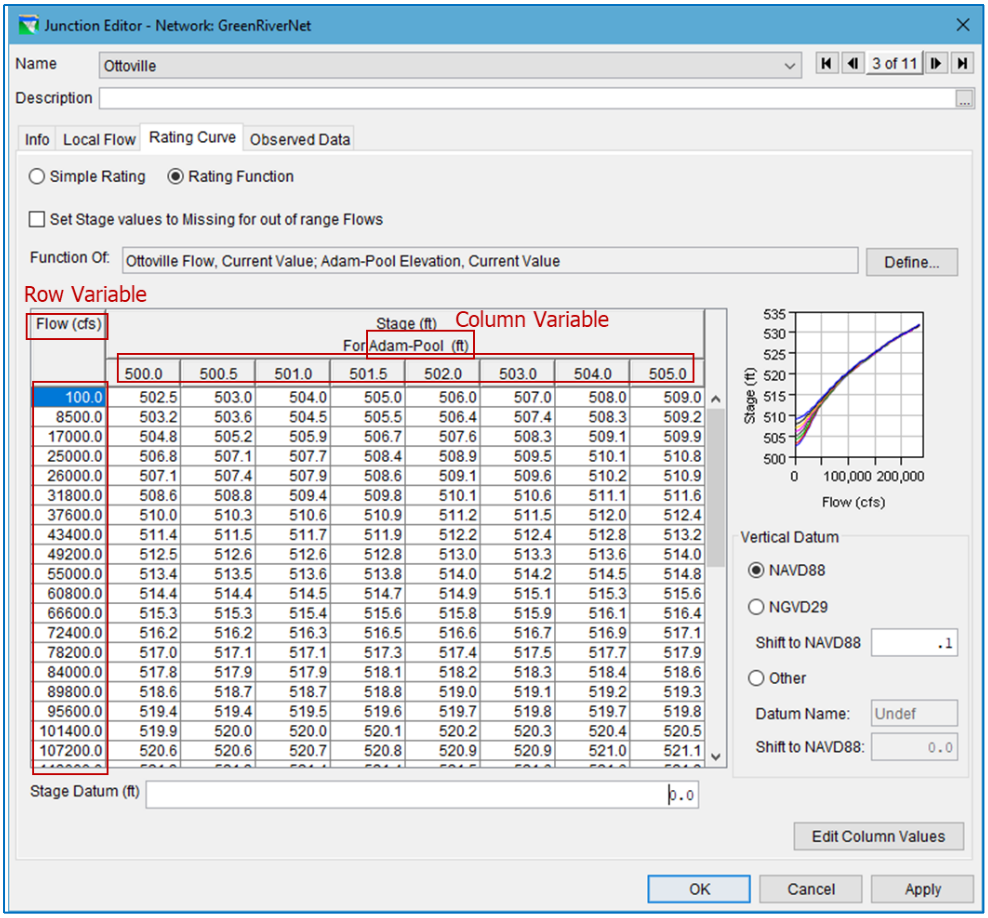


Figure C.9 Rating Curve as a Function of Two Variables

When the Function of **Two Variables** option is selected, the edit panel of the **Independent Variable Definition** dialog will display two panes: one for defining the **Row Variable** and one for defining the **Column Variable** (Figure C.10). Each of these two panes allow you to select either a Model, External or State Variable, and are otherwise similar to the other **Model Variables** panes.

Each of the two (row variable and column variable) **Two Variables** panes includes:

<u>A Variable Type selector</u>—a dropdown list for selecting Model Variable, External Variable, or State Variable as the independent variable.

<u>An Interpolation Type selector</u>—a dropdown list for selecting the type of interpolation to use between rows or columns of the independent variable. Interpolation types available are **Linear**, **Cubic**, and **Step**.

Once a variable type has been selected, the rest of the pane populates with the options inherent to either Model Variable, External Variable, or State Variable (as described in the previous sections).

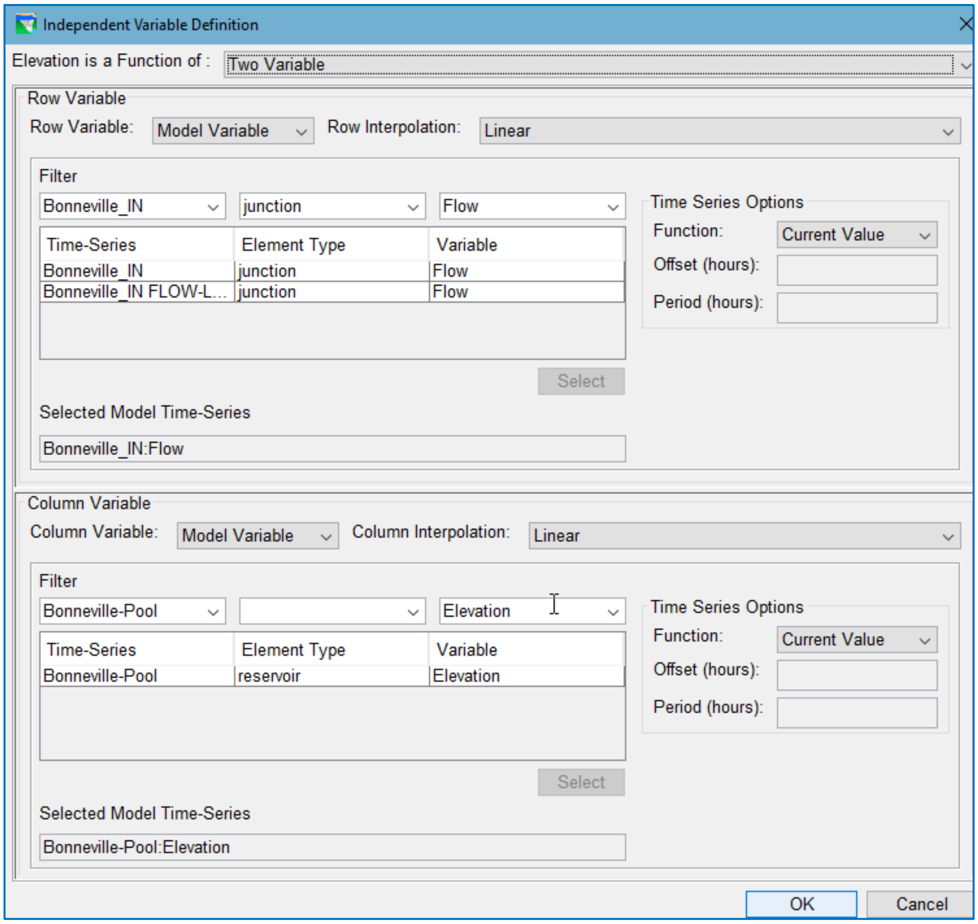


Figure C.10 Independent Variable Definition Dialog—Model Variable

C.3 Seasonal Variation Dialog

In the previous section, you learned about the **Independent Variable Definition** dialog. That dialog allowed you to specify one of the two variables in a functional relationship. Occasionally, the relationship between the two variables may vary as a function of time (season). The **Seasonal Variation** dialog is provided so that you can specify the seasons or days of the year when the relationship changes.

Usually included in the **Seasonal Variation** dialog is an **Interpolation Type** selector. The options in the selector are usually **Linear** and **Step**, although you may also find **Cubic** as an option. It is important to understand the interpolation type so that you know how ResSim will use the dates in the table with respect to the functional relationship you are defining.

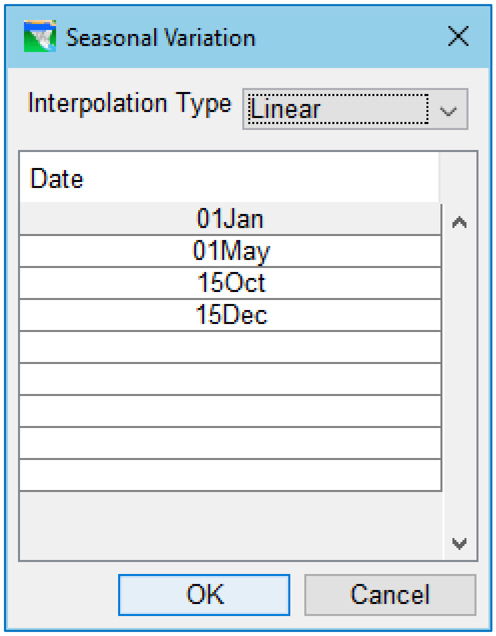


Figure C.11 Seasonal Variable Dialog

If the functional relationship does not vary (is constant) across a season, use **Step** as the **Interpolation Type** and enter dates in the **Seasonal Variation** table that represent the first day of each season.

If the functional relationship changes linearly across between two successive dates, then use Linear as the **Interpolation Type** and enter dates in the **Seasonal Variation** table that represent the inflection points of the linear relationship that the functional relationship has with time.

The dates in the Seasonal Variation table should be entered in DDMMM format. That means two digits to identify the day of the month and three 3 characters to identify the month. For days 1-9, use a preceding 0 (i.e., use 01-09). The 3-character month is the first three letters of the name of the month, in English. The month characters are not case sensitive, so Nov is the same as NOV and nov.

You may have noticed that the **Date** table in the Seasonal Variation dialog starts with **01Jan** and that entry is non-editable (greyed-out). This means that the list of seasons or dates always starts on **01Jan**. So, if you have a season that crosses the calendar year boundary, treat that season as two seasons. The start of the first half of the season will be the last date entry in the table and the start of the second half of the season will be the first date entry in the table (i.e., **01Jan**).

C.4 Selector Dialog

A selector (or mover) dialog is once in which two lists are provided side by side with some "move" buttons in between them. Typically, the list on the left is labeled **Available** and the list on the right is labeled **Selected** and you are expected to "move" entries from **Available** into **Selected**.. The **Selector** dialog (Figure C.12) gets its name from its primary purpose—to allow you select one or more items from the list of available items.

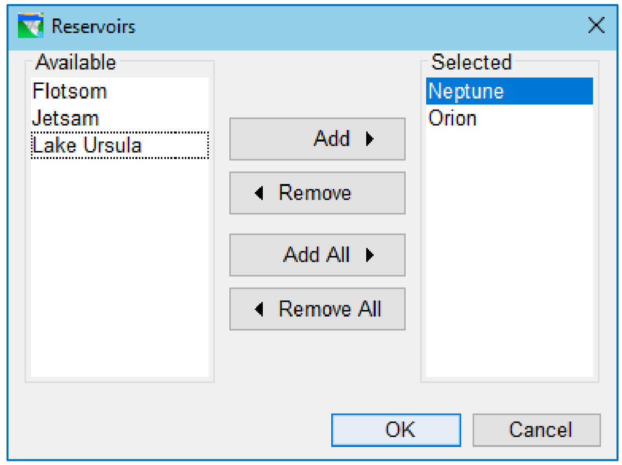


Figure C.12 Reservoirs Selector Dialog

To make a selection:

- *Double-click* on each item in the **Available** list that you want to select; this will move the item to the **Selected** list.

  —OR—

- *Click* on the item in the **Available** list, then use the **Add** button to move the selected item to the **Selected** list.

  NOTE: By holding the **Shift** or **Ctrl** keys while *clicking* on an item, you can highlight more than multiple items to be moved with the **Add** button.

- Click **OK** to accept your selections and close the **Reservoir** selector dialog. **Cancel** will close the **Selector** dialog without saving the changes that may have been made to the **Selected** list.

To select *all* the items in the **Available** list:

- Use the **Add All** button. This will move all the items in the **Available** list to the **Selected** list
- Click **OK** to accept your selections and close the **Reservoir** selector dialog.

To remove items from the **Selected** list:

- Double-click on each item in the **Selected** list that you want to deselect. This will move the item to the **Available** list

  —OR—

- *Click* on the item in the **Selected** list, then use the **Remove** button to move the selected item to the **Available** list.

  — OR—

- Use the **Remove All** button to move all the items from the **Selected** List to the **Available** list.
- Click **OK** to accept your changes and close the **Reservoir** selector dialog.

Appendix D
Using HEC-DSSVue

HEC-DSS is a data storage system developed by the US Army Corps of Engineers Hydrologic Engineering Center (HEC). HEC-DSSVue (or DSSVue) is a utility program, also developed by HEC, to create and manage the data stored in HEC-DSS database files. (DSS files have extension type *.dss*.) Since ResSim relies on HEC-DSS for storing and retrieving all of its input and output time-series and paired data, DSSVue is included as an integral part of ResSim. With DSSVue, data may be entered, edited, plotted, tabulated, and manipulated with over fifty mathematical functions. In addition to these functions, DSSVue provides several utility functions, such as entering data sets into a database, renaming data sets, copying data sets to other DSS database files, and deleting data sets.

HEC-DSS identifies, stores, and retrieves to data stored in DSS files by *pathnames*. Each pathname represents a record in the database. Pathnames are separated into six parts labeled "A" through "F." The six parts are delimited by slashes "/".

To identify *regular-interval time-series data*, the naming conventions for the six pathname parts are:

A	Project or watershed name
B	Location or gage identifier
C	Data variable or parameter, such as FLOW or PRECIP
D	Starting date of the time series data, using the date format DDMmmYYYY
E	Time interval of the data
F	Additional user-defined descriptive information

For each DSS file you open, DSSVue will generate a list of the pathnames associated with the data stored in the file. You can select one or more datasets from the list of pathnames (or *catalog*) and visualize the data in tabular or graphical form. The list of pathnames can be displayed in a variety of forms—from a list of simple strings, to a list of pathnames separated into parts. You can refine the list by searching for either a string anywhere in the in the pathnames or by filtering by one or more pathname parts.

One form of the pathname list is called the Condensed Catalog. This pathname list displays one row (condensed pathname) for each dataset. A *dataset* is a set of dss records with the same A, B, C, E, & F parts that spans the full time window of the data identified by those pathname parts stored in the database. The condensed pathname for the dataset displays the D part as the time window of the dataset rather than the start time of the data in a single record of the dataset.

This appendix provides a brief overview of DSSVue and describe some of the general functions you may need when working with DSS files in ResSim. Refer to the *HEC-DSSVue User's Manual* for a complete description of all the features and capabilities of DSSVue.

D.1 Launching DSSVue from ResSim

Although you can install and run HEC-DSSVue independently, it is also included as part of HEC-ResSim and is available from any ResSim module. To open DSSVue from within ResSim:

- Select **HEC-DSSVue** from the ResSim **Tools** menu.

The main window of HEC-DSSVue will appear (Figure D.1).

D.2 Exploring the DSSVue User Interface

The DSSVue user interface was designed to make working with DSS files easy and intuitive. The annotations in Figure D.1 identify the various components of the DSSVue interface that you will quickly become familiar with.

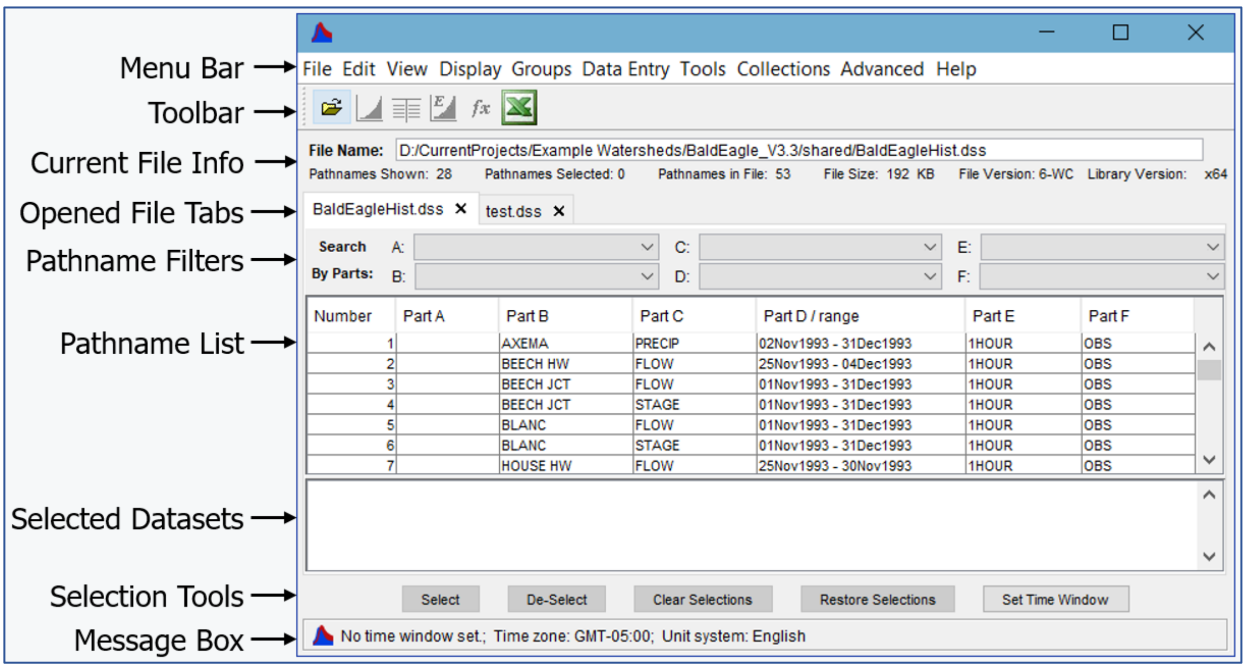


Figure D.1 The HEC-DSSVue User Interface

The **Menu Bar** includes menus that provide access to the various features for managing and viewing your DSS files and their data.

The **Toolbar** provides shortcuts to some menu options that are used most frequently. You can add buttons to the toolbar to execute scripts that you have written. And, some DSSVue Plugins may add buttons to the toolbar as well.

The **Current File Info** section displays the fully qualified path to the active DSS file as well as some properties of the file including its size and the version of HEC-DSS that rote it.

The **Opened File Tabs** cover the remainder of the DSSVue Main Window. The label on the tab is the name of the file whose contents are displayed on that tab. The components of the tab include:

The **Pathname Filters** are used to thin the pathname list, making it easier to find the data you are looking for.

The **Pathname List** which shows the list of records or data sets stored in the DSS file.

The **Selected Dataset**s list which shows the pathnames of the datasets you have chosen.

The **Selection Tools** are (with one exception) for managing the entries in the Selected Datasets list.

The **Message Box** shows status or informational messages generated by DSSVue.

D.3 The DSSVue Menus

The Menus in DSSVue provide access to a wide variety of features for managing your DSS files and the data stored in them. Some of the features available allow you to search for, select, and edit DSS datasets; control the display of pathnames; and access plots and tables. The HEC-DSSVue menus are as follows:

File—The **File** menu features are for managing dssfiles. Options include: **New…**, **Open…**, and **Close DSS File(s)**, **Print Catalog Preview…,** and **Print Catalog…**. The **File** menu also lists your six most recently opened files; these entries are shortcuts for opening the listed files.

Edit—The **Edit** menu options apply to the records in the Selected Datasets list (or the highlighted datasets if the Selected Datasets list is empty). The **Edit** functions include: **Tabular Edit…**, **Graphical Edit…**, **Select All**, **Rename Records**, **Delete Records**, **Undelete** (*All*, *Select…*, *Last Deleted*), **Duplicate**, **Copy To**, and **Merge (copy) into**….

View—The **View** menu options allow you to customize the display and filtering of DSS pathnames. Options include: **Pathname List**, **Pathname Parts**, **Condensed Catalog**, **No Pathnames**, **Unsorted List**, **Search pathnames by string**, **Search pathnames by parts**, **Refresh Catalog**, **Unit System** (***As Stored***, ***English***, ***SI***), and **Time Zone** (***As Stored***, ***Local***, ***UTC (GMT)***, ***Custom***).

Display—The **Display** menu features relate to the visualization of your selected datasets as well as provide accesses for setting some parameters that affect the visualization of the data. Menu options include: **Plot**, **Plot Individual Data Sets**, **Tabulate, Display Data Options** (***Normalize*** and ***Sync data set times to first***), **Supplemental Information**, and **Time Window…**.

Groups—The **Groups** menu features are for creating and managing groups of data sets. The menu options include: **Save Selected**, **Get**, **Plot**, **Plot Individual Sets**, **Tabulate**, **Math**, and **Manage…**

Data Entry—The **Data Entry** menu provides features for creating new datasets in DSS and entering data. The menu options include: **Manual Time Series…**, **Manual Paired Data…**, and **Manual Text…**. Also available are **Import** and **Export** options from a small list of formats although plugins exist that can expand the sources and data formats extensively.

Tools—The **Tools** menu features relate to the manipulation and analysis of the selected data sets as well as a couple of options for DSS file management. The **Tools** menu options include **Math Functions…**, **Compare** (***Files***, ***Data Sets***, ***Data Sets with Options***), **Search for Value**, **Check File Integrity**, and **Squeeze**.

Collections—The **Collections** menu is generated by a DSSVue plugin that is included in the ResSim package. ResSim includes the Collections plugin because DSS Collections are used to represent Ensemble datasets which are needed as input for ResSim Ensemble Alternatives. Appendix G provides details on **Ensemble Alternatives** and the DSSVue **Collections** features.

Advanced—The **Advanced** menu features are for the "advanced" user and relate to generating and managing catalog files and setting some program options. Menu options include **Condensed Disk Catalog** (***New***, ***View***, ***Print***), **Abbreviated Disk Catalog** (***New***, ***View***, ***Print***), **Full Disk Catalog** (***New***, ***View***, ***Print***), **Debug** (***Message Level…***, ***DSS File Header***, ***Record Addresses***, ***Pathname Bin***, ***Debug/Examine File…***, ***Memory Monitor…***, ***DSS Files Opened***, ***DSS ZSET***, ***DSS ZINQIR***), and **Program Options**.

Help—The **Help** menu contains the **About** command which will display version information about DSSVue.

D.4 DSSVue Toolbar Buttons

The buttons on the DSSVue **Toolbar** are shortcuts to some menu options that are used frequently. The standard set of buttons are:

Opens the file browser so that you can select a file to open.
This is a shortcut for **File -> Open**.

Plots the selected dataset(s).
This is a shortcut for **Display -> Plot**.

Displays a table listing the contents of the selected datasets.
This is a shortcut for **Display ->Tabulate**.

Opens the Graphical Editor with the selected datasets.
This is a shortcut for **Edit -> Graphical Edit**.

Opens the **Math Functions** editor with the selected datasets.
This is a shortcut for **Tools -> Math Functions…**.

Open the selected datasets in a Microsoft® Excel spreadsheet.
This is a shortcut for **Display -> Tabulate in Excel**

D.5 Managing DSS Files

The **File** menu (Figure D.2) provides most of the options you might need for managing DSS files—like **New** and **Open**. And, like most Windows® applications, the **File** menu includes a **Close** command for exiting DSSVue.

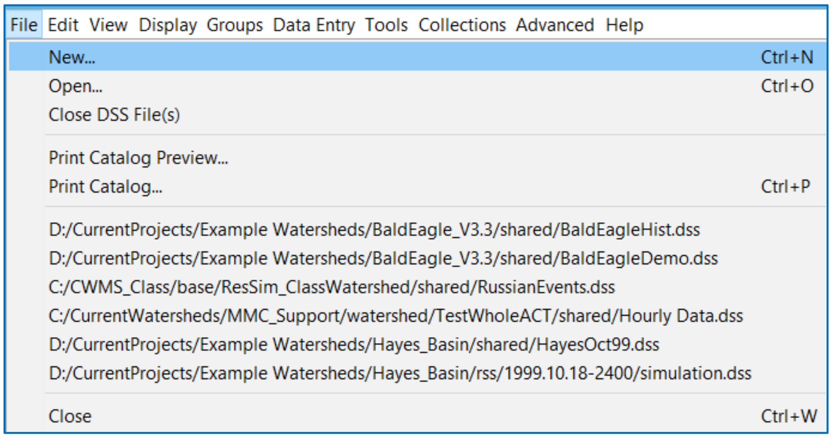


Figure D.2 HEC-DSSVue—File Menu

D.5.1 Opening DSS Files

To open an existing DSS file, you have three options:

- Click the **Open** toolbar button, —OR- Select **Open...** from the **File** menu.
- An **Open HEC-DSS File** file browser will appear (Figure D.3). Use the browser to navigate to and select the file you want to open, the click **Open**.

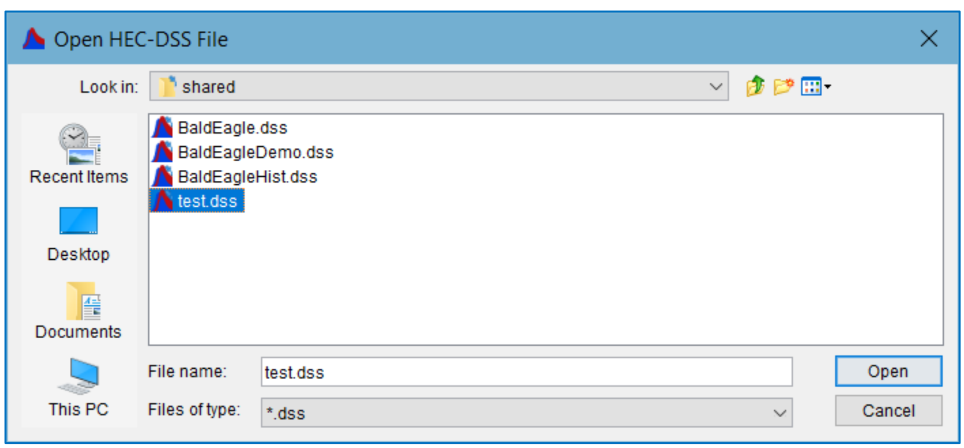


Figure D.3 Open HEC-DSS File File Browser

- Or, select one of the recently opened files listed near the bottom of the **File** menu.

NOTE: When DSSVue is opened from the Simulation module of ResSim, it automatically opens the current simulation's *simulation.dss* file.

D.5.2 Creating DSS Files

To create a new DSS file:

- Select **New…** from the **File** menu. A **Create new HEC-DSS File** file browser will appear (Figure D.4). Use the browser to navigate to the folder in which you want to store the new DSS file, then enter a filename and click **Create**.

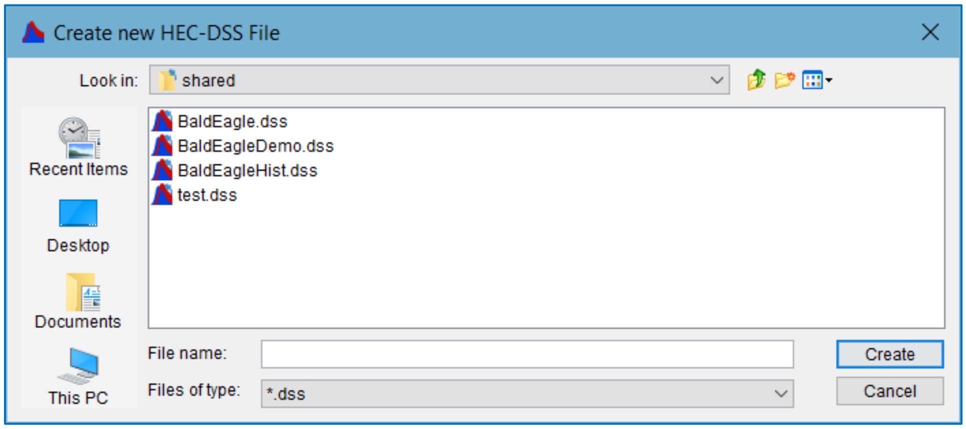


Figure D.4 HEC-DSSVue File Browser—Create new HEC-DSS File

You can also use the Open HEC-DSS File file browser to create a DSS file.

- Just *click* the Open toolbar button, .
- Browse to the folder where you want to store the file and enter a name in the File name field.
- *Click* Open.
- You'll get a message box telling you that the file doesn't exist and asking if you want to create it. *Click* OK to create the new DSS file.

D.6 The DSSVue Pathname List

When you open a DSS file, DSSVue will display the pathnames representing the datasets stored in the file. The appearance of this display of pathnames is controlled by the options in the **View** menu (Figure D.5).

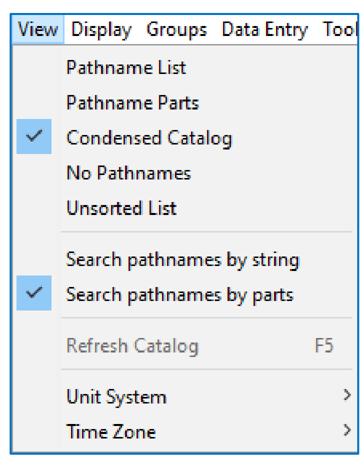


Figure D.5 HEC-DSSVue—View Menu

D.6.1 The Pathname List Views

The most commonly used **View** option is the:

Condensed Catalog (Figure D.6).—When in the **Condensed Catalog** view, DSSVue shortens the pathname list by combining pathnames that vary only by the D-part (start date); thus, the D-part of each "condensed" pathname shows the date range of all the records that share the rest of the pathname parts. Each condensed pathname identifies a DSS *dataset* (which can contain multiple DSS *records*).

Number	Part A	Part B	Part C	Part D / range	Part E	Part F
1		2ND ST BRIDGE	FLOW	18Oct1999 - 21Oct1999	1HOUR	DAY2DAY---0
2		2ND ST BRIDGE	FLOW	18Oct1999 - 21Oct1999	1HOUR	DAY2DAY---1
3		2ND ST BRIDGE	FLOW	18Oct1999 - 21Oct1999	1HOUR	DAY2DAY---2
4		2ND ST BRIDGE	FLOW	18Oct1999 - 21Oct1999	1HOUR	GC ONLY---0
5		2ND ST BRIDGE	FLOW-CUMLOC	18Oct1999 - 21Oct1999	1HOUR	DAY2DAY---0
6		2ND ST BRIDGE	FLOW-CUMLOC	18Oct1999 - 21Oct1999	1HOUR	DAY2DAY---1
7		2ND ST BRIDGE	FLOW-CUMLOC	18Oct1999 - 21Oct1999	1HOUR	DAY2DAY---2
8		2ND ST BRIDGE	FLOW-CUMLOC	18Oct1999 - 21Oct1999	1HOUR	GC ONLY---0
9		2ND ST BRIDGE	FLOW-IN	18Oct1999 - 21Oct1999	1HOUR	DAY2DAY---0
10		2ND ST BRIDGE	FLOW-IN	18Oct1999 - 21Oct1999	1HOUR	DAY2DAY---1

Figure D.6 Pathname List—Condensed Catalog View

Other Pathname List views available from the **View** menu include:

Pathname List (Figure D.7)—Displays the pathnames as concatenated strings.

Pathname Parts (Figure D.8)—Displays the pathnames in columns by their six parts.

No Pathnames—the **No Pathnames** view is useful when you need to open a very large DSS file but do not need to see any pathnames in the Pathname List.

Unsorted List—like the **No Pathnames** view, the **Unsorted List** view is most useful for very large DSS files that could otherwise take a long time to open simply because DSSVue must not only generate the pathname list, it must sort it. This view looks like an unsorted version of the **Pathname List** view.

Number	Pathname
1	//2ND ST BRIDGE/FLOW/01OCT1999/1HOUR/DAY2DAY---0/
2	//2ND ST BRIDGE/FLOW/01OCT1999/1HOUR/DAY2DAY---1/
3	//2ND ST BRIDGE/FLOW/01OCT1999/1HOUR/DAY2DAY---2/
4	//2ND ST BRIDGE/FLOW/01OCT1999/1HOUR/GC ONLY---0/
5	//2ND ST BRIDGE/FLOW-CUMLOC/01OCT1999/1HOUR/DAY2DAY---0/
6	//2ND ST BRIDGE/FLOW-CUMLOC/01OCT1999/1HOUR/DAY2DAY---1/
7	//2ND ST BRIDGE/FLOW-CUMLOC/01OCT1999/1HOUR/DAY2DAY---2/
8	//2ND ST BRIDGE/FLOW-CUMLOC/01OCT1999/1HOUR/GC ONLY---0/
9	//2ND ST BRIDGE/FLOW-IN/01OCT1999/1HOUR/DAY2DAY---0/
10	//2ND ST BRIDGE/FLOW-IN/01OCT1999/1HOUR/DAY2DAY---1/

Figure D.7 Pathname List—Pathname List View

Number	Part A	Part B	Part C	Part D	Part E	Part F
1		2ND ST BRIDGE	FLOW	01OCT1999	1HOUR	DAY2DAY---0
2		2ND ST BRIDGE	FLOW	01OCT1999	1HOUR	DAY2DAY---1
3		2ND ST BRIDGE	FLOW	01OCT1999	1HOUR	DAY2DAY---2
4		2ND ST BRIDGE	FLOW	01OCT1999	1HOUR	GC ONLY---0
5		2ND ST BRIDGE	FLOW-CUMLOC	01OCT1999	1HOUR	DAY2DAY---0
6		2ND ST BRIDGE	FLOW-CUMLOC	01OCT1999	1HOUR	DAY2DAY---1
7		2ND ST BRIDGE	FLOW-CUMLOC	01OCT1999	1HOUR	DAY2DAY---2
8		2ND ST BRIDGE	FLOW-CUMLOC	01OCT1999	1HOUR	GC ONLY---0
9		2ND ST BRIDGE	FLOW-IN	01OCT1999	1HOUR	DAY2DAY---0
10		2ND ST BRIDGE	FLOW-IN	01OCT1999	1HOUR	DAY2DAY---1

Figure D.8 Pathname List—Pathname Parts View

D.6.2 Filtering the Pathname List

There are two options for filtering the Pathname List. These options are:

Search by Parts provides six drop-down lists, one for each pathname part (Figure D.9). Each drop-down list contains a list of all the different values for that pathname part currently used in the pathnames show in the Pathname List. That means that if you select a value for one of the filters, not only does the Pathname List update to show only those pathnames that use the selected string, the other filter lists update to include only those pathname part strings for the pathnames shown in the filtered pathname list.

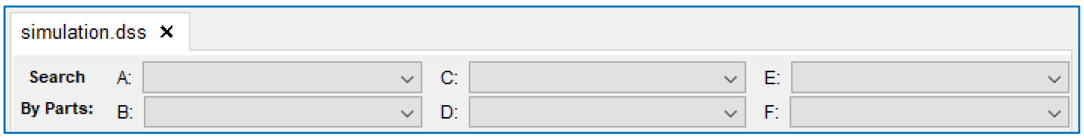


Figure D.9 DSSVue Pathname Filters—Search by Parts

Search by String provides a single field in which you can enter a string that you want to use to filter the list of pathnames (Figure D.10). In other words, the string will apply to all the pathname parts. For example, if you enter the string "flow", the Pathname List will show all pathnames that include the word flow *anywhere* in the pathname.

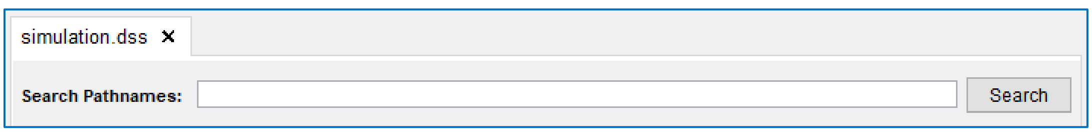


Figure D.10 DSSVue Pathname Filter—Search by String

D.6.3 Selecting DSS Pathnames

Selecting DSS Pathnames is no mystery. Just highlight one or more pathnames (rows) in the Pathname list then click the **Select** button. Or, double-click on each pathname you want to select. The pathnames you select will appear in the Selected Pathnames list.

Figure D.11 illustrates the row of buttons that are provided below the Selected Pathnames list in the DSSVue interface.

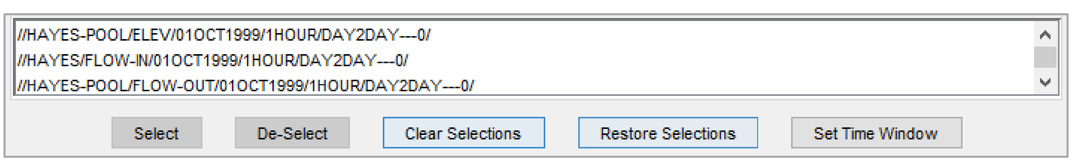


Figure D.11 Pathname Selection Tools

Each of the buttons in the set of **Selection Tools** is described below:

Select—To add pathnames to the Selected Pathnames list, highlight (click on) the pathnames to want to add in the Pathname List then click the **Select** button.

De-Select—To remove pathnames from the Selected Pathnames list, select the pathnames you want to remove the click the **De-Select** button.

Clear Selections—To remove all the pathnames from the Selected Pathnames list, click the **Clear Selections** button.

Restore Selections—This tool works as an UNDO button for the **Clear Selections** option; it restores the state of the Selected Pathnames list from right before the Clear Selections button was last pressed.

Set Time Window—This tool does not directly operate on the list of Selected Pathnames. Instead, it is a shortcut for the **Display -> Time Window…** command, which allows you to set a time window for viewing and editing your selected datasets.

D.7 Setting the Time Window

The setting of the DSSVue Time Window determines how much of each selected dataset will be used or impacted when viewing or editing. To set (or unset) the DSSVue Time Window you can either:

- Select **Time Window**...from the **Display** menu, or
- Click the **Set Time Window** button. The **Set Time Window** editor will be displayed (Figure D.12).

The **Set Time Window** editor has four radio button options to choose from for setting the time window:

No Time Window—this option clears the Time Window setting. When no Time Window is set, all the data in the selected datasets will be used or affected by the actions you take on them.

Specific Time Window—this option requires that you enter specific **Start** and **End** Dates and Times to define the time window. Dates are entered in DD MMM YYYY format. Times are entered using a military 24-hour clock (e.g., 3:00 pm is 1500).

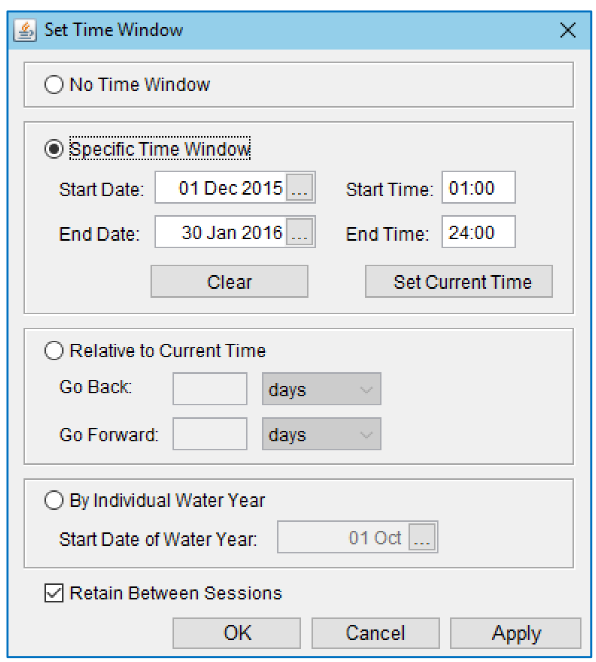


Figure D.12 Set Time Window Editor

The **... button**—in any date field can be used to open a Calendar tool for selecting a specific date.

Clear—clears the time window fields of any prior settings.

Set Current Time—uses the system time on your computer to fill in the Start and End dates and times.

Relative to Current Time—this option uses the system time of your compute to determine a time window. The parameters for this option are:

Go Back—a time interval from the drop-down list and then enter the number of intervals *backward* in time to set the *start* of the time window.

Go Forward—select a time interval from the drop-down list and then enter the number of intervals *forward* in time to set the *end* of the time window.

By Individual Water Year—this option will cause DSSVue to act on the selected datasets in Water Year chunks. This means that if data in a selected data set starts before the start of the Water Year or extends beyond the end of a water year, then the data outside a water year boundary will not be included in the "Time Window".

Start Date of Water Year—enter a date that defines the start of a water year for your data; this date is in the form DD MMM, a year is not required.

If you activate (check) the **Retain Between Sessions** checkbox, DSSVue will remember your Time Window settings between executions of DSSVue. If unchecked, the default of **No Time Window** will be used each time DSSVue is launched.

NOTE: When DSSVue is opened from the Simulation module of ResSim, ResSim sets the DSSVue Time Window to the time window of the active Simulation.

D.8 Visualizing Your DSS Data

Two methods for visualizing the data stored in your DSS files are available in DSSVue—plotting and tabulating. Below are some details you may find useful in working with DSS Plots and Tables

D.8.1 Plotting DSS Data

To tabulate the data stored in a DSS file:

- Select the datasets of interest, then…
- Click the **Plot** button on the Toolbar—OR—Select **Plot** from the **Display** menu.

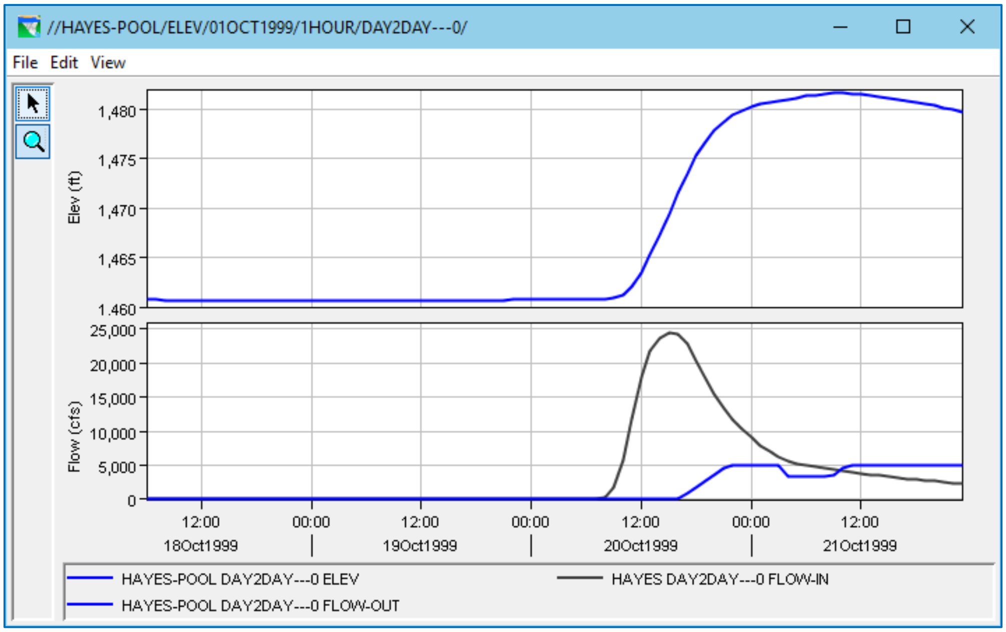


Figure D.13 HEC-DSSVue Time-Series Plot Window

Figure D.13 shows a Plot Window that displays the two time-series datasets selected in Figure D.11 with the time window set as shown in Figure D.12.

Each Plot Window that displays *time-series* data includes a…

Title Bar—showing the pathname of the first selected dataset
Menu Bar—containing **File**, **Edit**, and **View** menus.
Vertical **Toolbar**—containing a **Pointer** tool and a **Zoom** tool
Viewports (one or more)—showing a graph of the data
Legend—included directly below the viewports—it identifies each dataset and shows the line style used to display the data in its plot.
Y Axis—one for each viewport—it is labeled to show the parameter, units, and range of values of the plotted data
X Axis—the horizontal axis that is shared by all viewports since it represents the *time* axis; all the plotted data in the viewports are aligned in time.

You can resize the plot window by dragging any corner of the window to the desired size; the contents of the plot (especially the axes and the text of the legend) will rescale as you change the size of the plot window.

You can use the **Zoom** tool  to view only a subset of the data in the plot. To zoom in, use the **Zoom** tool to drag a box (within a viewport or along one axis or the other) around the portion of the plot you want to view; the plot content inside the zoom box will fill the viewport or the plot will adjust to show data relevant to the portion of the axis you zoomed in on. While zoomed in, the plot window will show scroll bars to allow you view more of your data without having to zoom back out. However, to zoom back out, just right click anywhere in the graph or along the axes.

Refer to the *HEC-DSSVue User's Manual* for more details on configuring a DSSVue plot window.

D.8.2 Tabulating DSS Data

To tabulate the data stored in a DSS file:

- Select the datasets of interest, then…
- Click the **Tabulate**  button on the Toolbar, or…
- Select **Tabulate** from the **Display** menu.

Figure D.14 shows a table window that displays the two time-series datasets selected in Figure D.11 with the time window set as shown in Figure D.12.

Like a plot window, each table window that displays *time-series* data includes a…

Title Bar—showing the pathname of the first selected dataset

Menu Bar—containing **File**, **Edit**, and **View** menus.

But it also includes a table showing the data stored in the selected datasets. The table includes a column for:

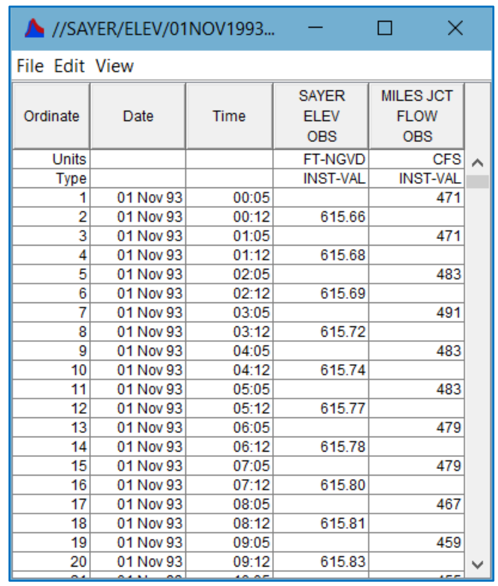
The **Ordinate** number (or counter) for each row of data in the table. Values in a time-series table are assigned an ordinate in increasing order of time.

The **Date** and **Time** associated with each value in the datasets

The **Values** for each selected dataset

A **Header** row that labels each column in the table.

A **Scrollbar** to allow you to scroll through the data since most tabulated time series data will not fit on your computer screen.



Ordinate	Date	Time	SAYER ELEV OBS	MILES JCT FLOW OBS
Units			FT-NGVD	CFS
Type			INST-VAL	INST-VAL
1	01 Nov 93	00:05		471
2	01 Nov 93	00:12	615.66	
3	01 Nov 93	01:05		471
4	01 Nov 93	01:12	615.68	
5	01 Nov 93	02:05		483
6	01 Nov 93	02:12	615.69	
7	01 Nov 93	03:05		491
8	01 Nov 93	03:12	615.72	
9	01 Nov 93	04:05		483
10	01 Nov 93	04:12	615.74	
11	01 Nov 93	05:05		483
12	01 Nov 93	05:12	615.77	
13	01 Nov 93	06:05		479
14	01 Nov 93	06:12	615.78	
15	01 Nov 93	07:05		479
16	01 Nov 93	07:12	615.80	
17	01 Nov 93	08:05		467
18	01 Nov 93	08:12	615.81	
19	01 Nov 93	09:05		459
20	01 Nov 93	09:12	615.83	

Figure D.14 HEC-DSSVue Table Window

The **View** menu of a DSSVue table window provides you with several options for displaying the tabulated data (Figure D.15):

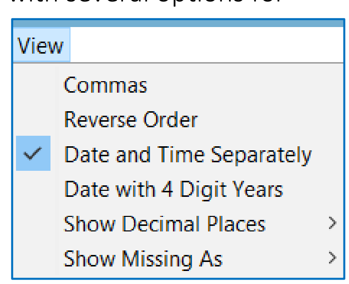


Figure D.15 HEC-DSSVue Table Window—View Menu

Commas—will include commas in the data values. For example, a value of one thousand, two hundred thirty-four and fifty-six hundredths will be displayed *without* commas as 1234.56 and *with* commas as 1,234.56. A checkmark will appear in front of this option in the menu when this option is ON. When unchecked (OFF), the data will display *without* commas.

Reverse Order —will display the data in the opposite order. Since the default order for time-series data is *oldest to newest* (oldest first), reversing the order will list the data from *newest to oldest* (newest first). The Ordinate values will be reassigned based on your selected order. A checkmark will appear in the menu when this option is ON.

Date and Time Separately—will split the date-time stamp into two columns. A checkmark will appear in the menu when this option is ON as shown in Figure D.15. When unchecked (OFF), the date and time appear together as a single column.

Date with 4 Digits —will display the year in the date with four digits instead of the two. A checkmark will appear in the menu when this option is ON.

Show Decimal Places—unlike the previous options, this is not a toggle switch. Instead, this menu option provides a submenu from which you can select how many digits after the decimal you want displayed.

Show Missing As—provides a submenu of options for how missing values are displayed in the table. Options include:

- **Blank** (the default)
- **-901.0** (an old HEC-DSS paradigm for missing values)
- **M**
- **-M-**

Refer to the *HEC-DSSVue User's Manual* for more details on configuring a DSSVue table window.

Tip

FYI—You can Tabulate from a Plot and Plot from a Table...
Just look in the File menu of your Plot or Table window to find the option to Tabulate or Plot.

D.8.3 Printing Plots and Tables in HEC-DSSVue

You can print plots and tables by selecting **Print** from the **File** menu of the plot and table windows. For more details on the printing capabilities, see Appendix E, Printing and Exporting ResSim Data.

D.9 Editing Your DSS Data

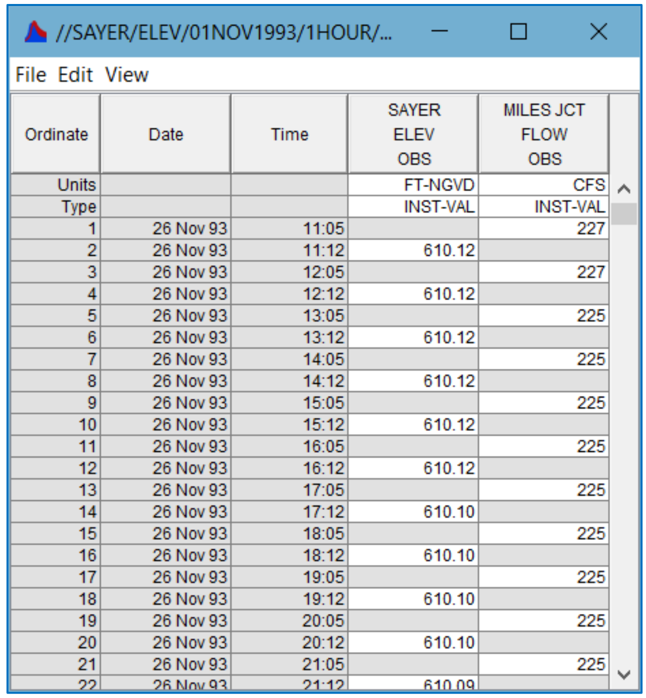
There are three ways to directly edit the values of the data stored in your *selected* DSS datasets:

- Select **Graphical Edit** from the **Edit** menu in DSSVue
- Select **Tabular Edit** from the **Edit** menu in DSSVue
- Select **Allow Editing** from the **Edit** menu of a DSSVue Table window

The third option is an easy way to make quick edits to data you are already viewing in a table window. Please refer to the *HEC-DSSVue User's Manual* for details on using the Graphical and Tabular Editors.

When you select **Allow Editing** from the **Edit** menu of a table window, the window display switches into *Edit* mode—see Figure D.16. The Ordinate and Date/Time columns grey out indicating that these columns are not editable. To edit a value in one of the data columns, just click in the cell and type in a new value.

You can also select a range of cells then right-click and select **Fill...** from the context menu. After making changes to the data, DSSVue will prompt you to save changes to the data set when you close the window or you can select **Save** or **Save As...** from the **File** menu before closing.



Ordinate	Date	Time	SAYER ELEV OBS	MILES JCT FLOW OBS
Units			FT-NGVD	CFS
Type			INST-VAL	INST-VAL
1	26 Nov 93	11:05		227
2	26 Nov 93	11:12	610.12	
3	26 Nov 93	12:05		227
4	26 Nov 93	12:12	610.12	
5	26 Nov 93	13:05		225
6	26 Nov 93	13:12	610.12	
7	26 Nov 93	14:05		225
8	26 Nov 93	14:12	610.12	
9	26 Nov 93	15:05		225
10	26 Nov 93	15:12	610.12	
11	26 Nov 93	16:05		225
12	26 Nov 93	16:12	610.12	
13	26 Nov 93	17:05		225
14	26 Nov 93	17:12	610.10	
15	26 Nov 93	18:05		225
16	26 Nov 93	18:12	610.10	
17	26 Nov 93	19:05		225
18	26 Nov 93	19:12	610.10	
19	26 Nov 93	20:05		225
20	26 Nov 93	20:12	610.10	
21	26 Nov 93	21:05		225
22	26 Nov 93	21:12	610.09	

Figure D.16 DSSVue Table Window—Edit Mode

D.9.1 Using the Math Functions Editor

To perform math functions on your selected datasets:

- Select the dataset(s) on which you want to perform a math function.
- Click the Math Functions tool or select **Math Functions** from the **Tools** menu. The **Math Functions** editor will open, as shown in Figure D.17.

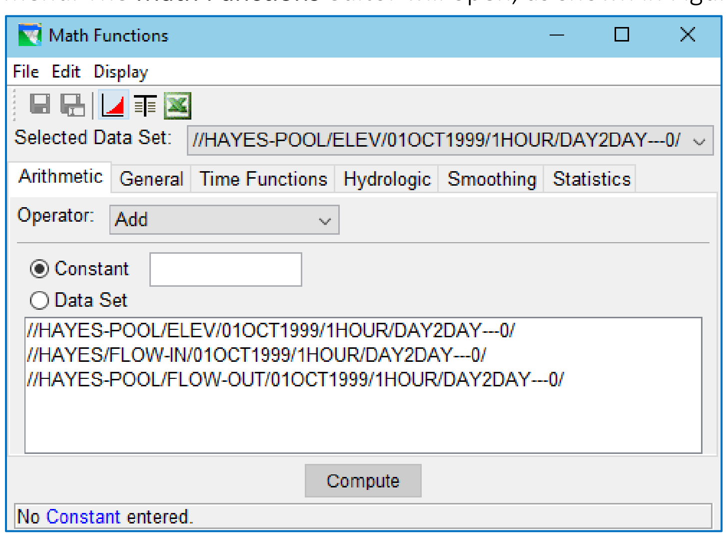


Figure D.17 HEC-DSSVue Math Functions Editor

- Select one of the six tabs (**Arithmetic**, **General**, **Time Functions**, **Hydrologic**, **Smoothing**, and **Statistics**). Figure D.18 shows the **Statistics** tab of the Math Functions Editor—these basic statistics on a dataset can be very useful when analyzing data and results.

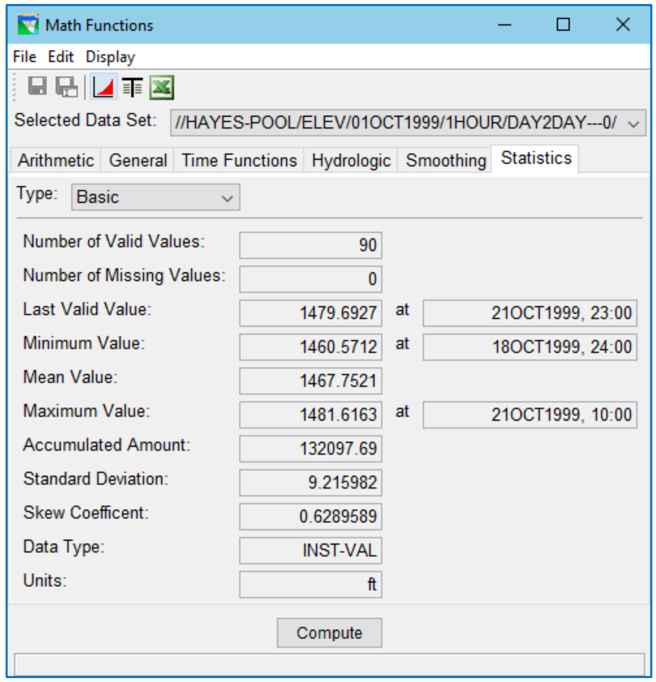
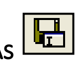


Figure D.18 HEC-DSSVue Math Functions—Statistics Tab

- After you have selected the appropriate math function (tab and **Operator**) and the required parameters for the function, click **Compute**. DSSVue will perform the selected math on the dataset shown in the **Selected Data Set** selector.
- To view the results, click the **Plot**  or **Tabulate**  button. The computed results from the selected math function will be displayed.
- To *save the new values in place of the original values*, click the **Save**  button (or select **Save** from the **File** menu). HEC-DSSVue will ask you to confirm that you wish to replace the original data.
- To *save the new values as a new record*, click the **SaveAs**  button or select **Save As** from the **File** menu. A **Save As** dialog will open (Figure D.19).
- Enter the new pathname information then click **OK** to save the record.

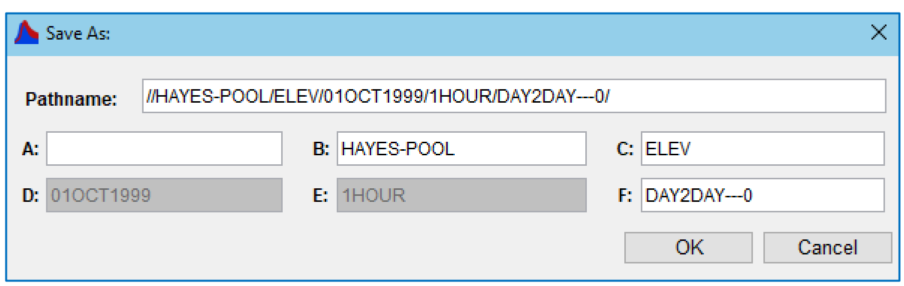


Figure D.19 HEC-DSSVue Save As Dialog

There are many very useful math functions available in HEC-DSSVue. This Appendix only covers the basic functionality for accessing the functions. The *HEC-DSSVue User's Manual* (HEC, 2009) is the primary reference for further information and details for using the **Math Functions** Editor.

D.9.2 Manual Data Entry

D.9.2.1 Time Series Data

To enter **Time Series** data manually:

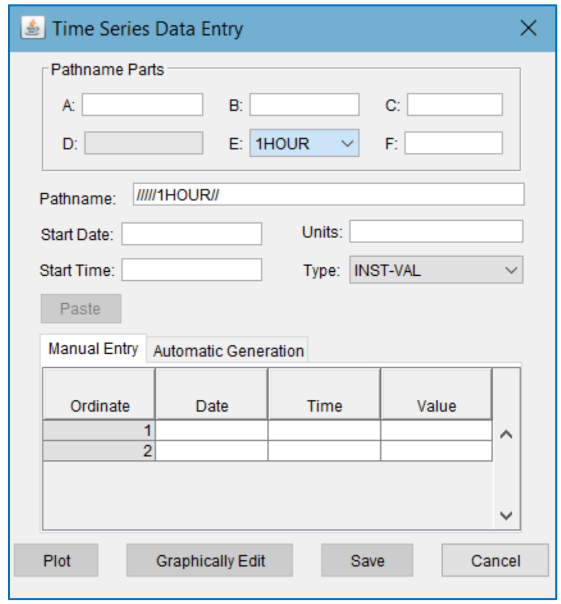


Figure D.20 HEC-DSSVue Manual Time Series Data Entry Editor

- From the **Data Entry** menu, select **Manual Time Series...**. The **Time Series Data Entry** editor (Figure D.20) will open.
- Enter the **Pathname Parts** into the A, B, C, and F fields. DSSVue will define the D-part of the pathname based on the dates/times you enter for the data.
- You must select an appropriate time interval (E-part) from the list. The list includes both *regular* and *irregular* interval options for the time interval. The behavior of the data entry tabs will change depending on the E-part you select.
- The complete pathname will automatically appear in the **Pathname** field. (Alternately, you can enter the pathname into the **Pathname** field and the parts will appear in the **Pathname Parts** fields.)
- Enter the **Start Date** (*e.g.,* 30 Mar 2002).
- Enter the **Start Time** (*e.g.,* 0800).
- Enter the **Units** (*e.g.,* cfs, feet, etc.).
- Select a data type from the **Type** list. Your options are PER-AVER, PER-CUM, INST-VAL, and INST-CUM.
- Using the **Manual Entry** tab—if you have selected a *regular* interval for the E-part of the pathname, the **Date** and **Time** fields will fill in automatically according to the start date and time you have entered. If you selected an *irregular* interval for the E-part of the pathname, you must enter the Date and Time values for each data value you wish to enter.
    - The first two cells of the table are provided automatically, but when you press **Enter** or **Tab** on your keyboard after entering a value in the last cell of the table, a new row will appear to allow you to continue entering data.
    - Or, using the **Automatic Generation** tab, enter the **End Date** and **End Time** and a Fill Value. This option is useful when you want to generate a dataset that contains a constant value.

- To view the data in plot form, click the **Plot** button at the bottom of the window.
- To graphically edit (or enter) the data, click the **Graphically Edit** button.
- To save the new time-series dataset, click **Save**.
- To close the **Time Series Data Entry** editor, click Cancel.

D.9.2.2 Paired Data

To enter Paired Data manually:

- From the Data Entry menu, select Manual Paired Data….
- The **Manual Paired Data Entry** editor (Figure D.21) will open.
- Type the **Pathname Parts** into the A, B, C, D, E, and F fields.
- The **C-part** name has two fields, separated by a dash. Enter the **X** parameter name in the first field and the **Y** parameter name in the second. For example, a rating curve for a gage would have a C-part of STAGE-FLOW.

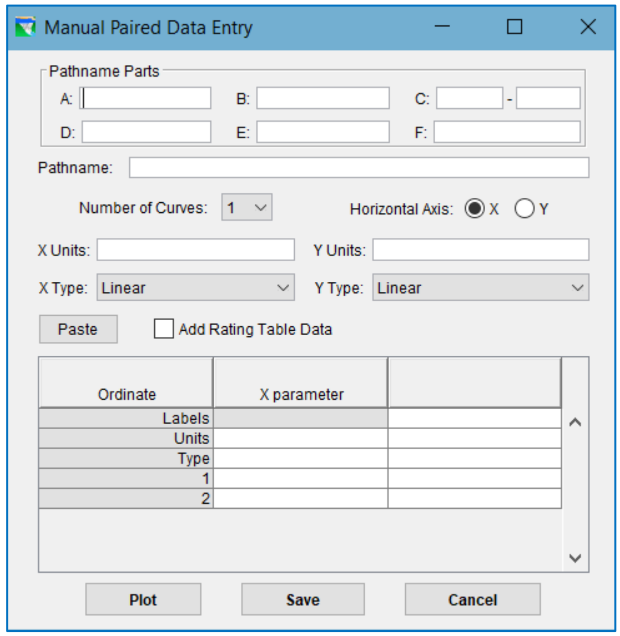


Figure D.21 HEC-DSSVue Manual Paired Data Entry Editor

- As you enter the various pathname parts, the full pathname will automatically appear in the **Pathname** field.
- Select the **Number of Curves** (Y curves) from the list (1-50). For example, if you are entering a gated outlet capacity table, each column of gate opening is a **Y** curve.
- Enter the **X Units** and the **Y Units**.
- Choose the **X Type** and **Y Type** scale from the lists. Available options are: **Linear**, **Log**, **Probability**, and **Percent**.
- The **Y ordinates** column of the table will split into individual columns according to the **Number of Curves** you have specified.
- Type the data values into the **X ordinates** and **Y ordinates** columns.
- To view the data in plot form, click the **Plot** button.
- To save the new time-series record, click **Save**.
- To close the Manual Paired Data Entry editor, click Cancel.

D.10 Managing Your DSS Datasets

D.10.1 Renaming Datasets

To **Rename** a DSS Record:

- Select the dataset(s) you want to rename.
- Select **Rename Records** from the **Edit** menu.
- The **Rename Records to** editor (Figure D.22) will open.
- Type the new pathname parts into the **A**, **B**, **C**, and/or **F** fields.
- Click **OK**.
- A confirmation message will appear, indicating that the record(s) has been renamed.

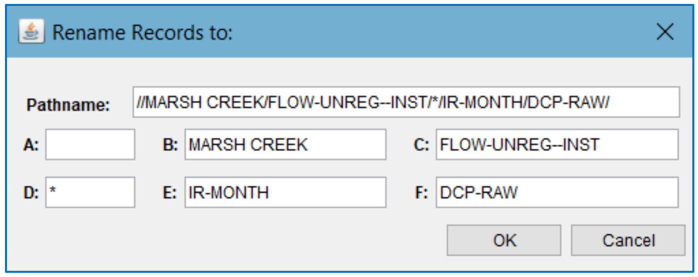


Figure D.22 Rename Records Editor

D.10.2 Copying Datasets to another DSS File

To **Copy** records into a DSS file:

- Select the dataset(s) you want to copy to another DSS file.
- Select **Copy To** from the **Edit** menu
- The **Copy Records into HEC-DSS File** Browser (Figure D.23) will open.
- Type in a new DSS **File Name** or select an existing DSS file into which you want to copy the record(s) and click **Open**. A message will appear, stating that the record has been copied to the DSS file you specified.

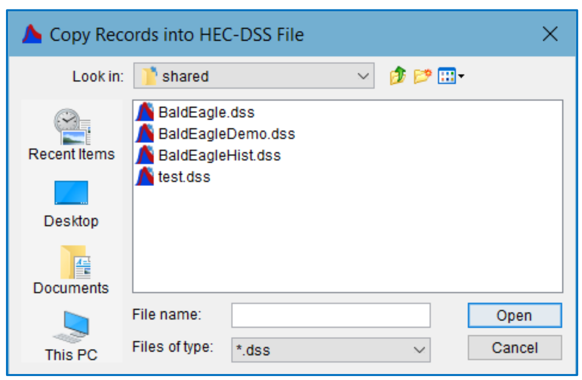


Figure D.23 Copy Records into HEC-DSS File Browser

D.10.3 Duplicating Datasets

To **Duplicate** one or more datasets:

- Select the dataset(s) you want to make a copy of in the current DSS file.
- Select **Duplicate** from the **Edit** menu.
- The **New pathname parts for duplicate records** dialog (Figure D.24) will open.

- Type the new pathname parts into the **A**, **B**, **C**, or **F** fields.
- Click **OK**.
- A message will appear, stating that the record(s) have been duplicated.

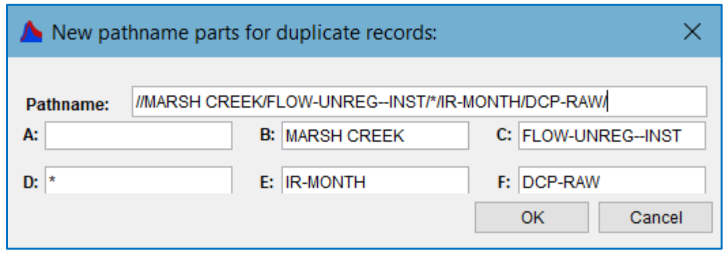


Figure D.24 New Pathname Parts for Duplicate Records Dialog

D.10.4 Deleting Datasets

To **Delete** records from a DSS file:

- Select the dataset(s) you want to delete.
- Select **Delete** from the **Edit** menu.
- A confirmation dialog will appear asking you to verify the datasets to be deleted (Figure D.25). If less than 5 datasets have been selected, they will be listed in the confirmation dialog, otherwise the number of datasets (and the number of records they represent) will be indicated.

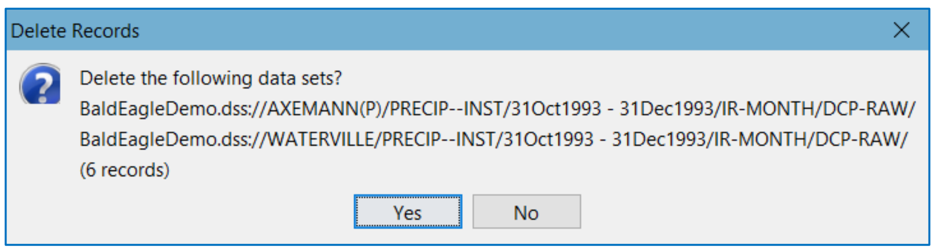


Figure D.25 Confirmation Message Box—List of Records to be Deleted

- If the list correctly identifies the record(s) to be deleted, select **Yes**.
- A message box will appear (Figure D.26) telling you the records have been deleted. Otherwise, select **No**, and the records will not be deleted.

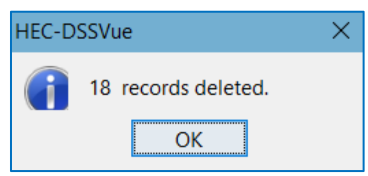


Figure D.26 Records Deleted Message

D.10.5 Undeleting Records

Deleted records can be recovered ***as long as the HEC-DSS file has NOT been squeezed*** (see Section E.14). When a file is squeezed, all deleted records are physically removed. You can undelete records in three ways:

- Undelete all records in a file
- Select the records to undelete from a list
- Undelete the records that you just deleted

To **Undelete** records from a HEC-DSS file:

- Select **Undelete** from the **Edit** menu. A submenu will appear with the options:

  All—To undelete *all* records, select **All**.

    - A confirmation message box will appear asking if you want to undelete *nn* records (where *nn* is the number of deleted records in the current file).
    - Click **OK.**
    - A final message box will appear stating that all available records were undeleted.

  Select…—To pick which records to undelete, select **Select…**

    - A window will appear containing a list of pathnames for all the records that have been deleted in the current file (Figure D.27) that are eligible to be undeleted. Check the checkboxes of the records that you want to undelete, or press **Select All** to undelete all records in the list. You can unselect the checked records by pressing **Unselect All**.

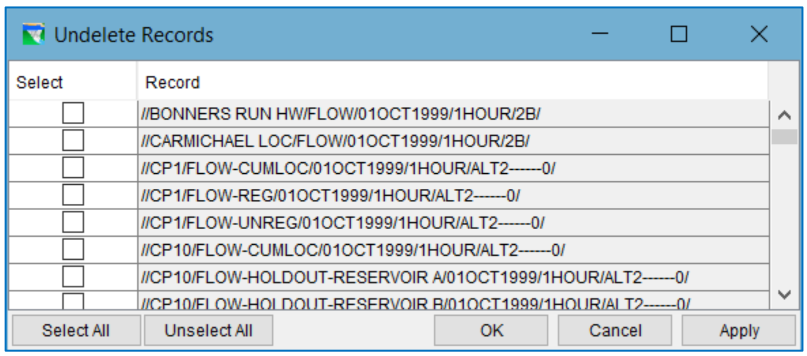


Figure D.27 Undelete Records Selection Window

    - Once you have indicated which records to undelete…
        - Click **OK** to undelete those records and close the window
        - Click **Apply** to undelete the records and leave the window open
        - Click **Cancel** to close the window without making further undeletes.

  Last Deleted—To undelete the records that you have just deleted, select **Last Deleted**.

    - A message box will appear asking if you want to undelete *nn* records (where *nn* is the number of records that were deleted in the most recent delete operation).
    - Click **OK.**

D.11 Managing Your DSS Files

D.11.1 Merging HEC-DSS Files

To Merge the current DSS file into another DSS file:

- Select **Merge (copy) …** from the **Edit** menu. The **Merge (copy all records) into HEC-DSS File** Browser (Figure D.28) will open.
- Enter in a new DSS **File name** or select an existing DSS file into which you want to copy all of the records from the current DSS file and click **Open**.
- A message will appear, stating "Files Merged." Click **OK** to acknowledge the message and close the message box.

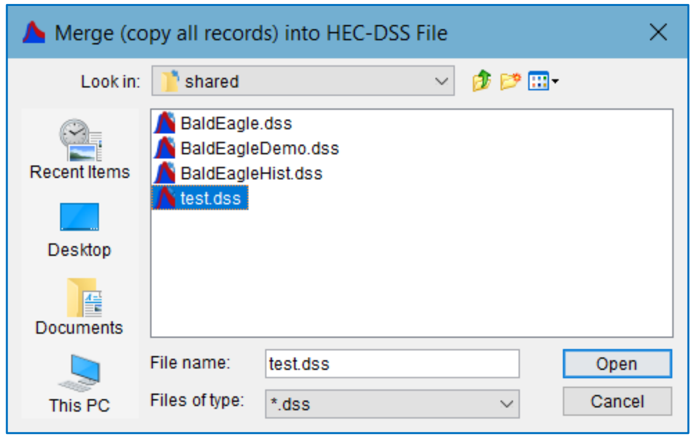


Figure D.28 Merge (Copy All Records) into HEC-DSS File Window

D.11.2 Squeezing DSS Files in HEC-DSSVue

Whenever you delete a DSS record, HEC-DSS marks the record as deleted but doesn't actually delete the record from the DSS file. Other actions, like rename and edit can cause HEC-DSS to create a new record and mark the old one as deleted. This dis-inclination to actually delete a record from the database is normal for database systems but it means that periodically, "maintenance" must be performed on the database to recover storage space by cleaning out the deleted records and to speed up data access by rebuilding index tables. The **Squeeze** command performs this maintenance by copying all valid data in the current DSS file to a new DSS file then deleting the original DSS file and renaming the new file to the old filename.

To perform maintenance on an HECDSS database:

- Open the DSS file in DSSVue
- Select **Squeeze** from the **Tools** menu.
- When the process is complete, a message box will appear (Figure D.29)

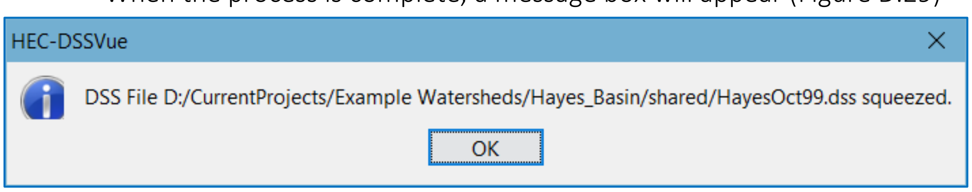


Figure D.29 HEC-DSSVue Squeeze Message (Example)

NOTE: Once a file has been squeezed, datasets that had been deleted CANNOT be undeleted.

D.11.3 Viewing Status of DSSVue and its DSS Files

DSSVue provides a variety of options to monitoring the status of DSSVue itself as well as the database(s) you have open. To access these options:

- Select **Debug** -> **OFF** from the **Advanced** menu in DSSVue.
- This will activate the **Debug** submenu which contains the following options:
  - **Message Level…—**this will temporarily change the message level for the messages that are written by HEC-DSS to its log file.
  - 
    DSS File Header—this should open a window showing a table of information that is stored by HEC-DSS in the DSS file header for the current file. *Unfortunately, in the version of DSSVue included in this version of ResSim, this option crashes ResSim.*
  - **Record Addresses—**this option opens a window that provides a variety of esoteric information about the records for the currently selected datasets. Most of this information is only useful to the programmers of DSSVue.
  - **Pathname Bin**—works only DSS 7 files. Shows information about the indexing of the Pathnames in the DSS file
  - **Debug/Examine File…**—Only useful to the programmers; this option allows them to look at what is stored at specific addresses in the DSS file
  - **Memory Monitor…**—this option displays the Memory Monitor (Figure D.30), a very small window which shows a graph of the quantity of real-time memory used by HEC-ResSim and HEC-DSSVue in relation to the memory allocated.

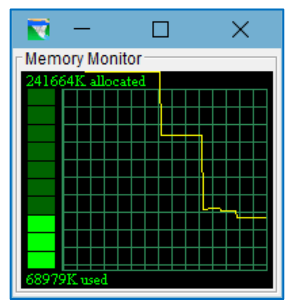


Figure D.30
Memory Monitor

  - **DSS Files Opened**—this opens the HEC-DSS File Manager Status window (Figure D.31). This window displays the number of HEC-DSS files you have accessed during the current session of DSSVue; and for each file it reports: the name and location of the file, whether the file is currently open, the first and last times you accessed the file, and the total number of accesses.
  - **DSS ZSET**—this option allows you to enter a ZSET parameter and value.
  - **DSS ZINQIR**—this option allows you to enter a ZINQIR parameter and will respond with information relevant to the parameter you provided.

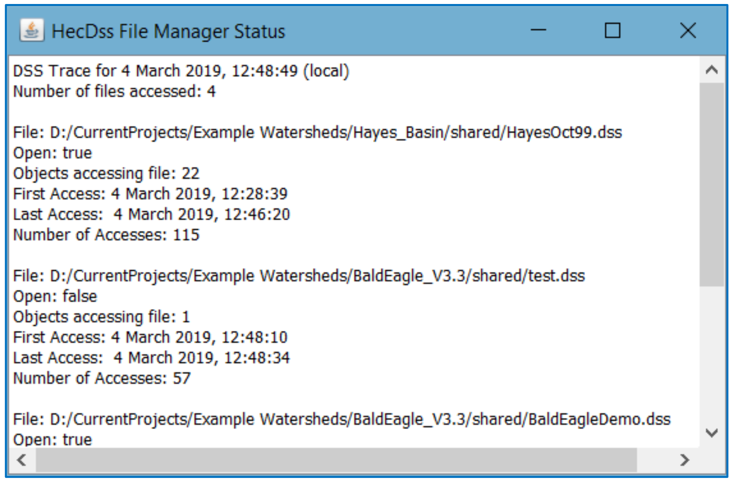


Figure D.31 HEC-DSS File Manager Status Window

Appendix E
Printing and Exporting ResSim Data

ResSim's plots and reports as well as HEC-DSSVue plots and tables offer detailed views of model data and results. Each of these data visualization features provide options for printing, as well as exporting and copying of the data for use in other applications.

E.1 Saving and Printing Plots

The **File** menu in ResSim and DSSVue plots (Figure E.1) contains options that allow you to:

- Save the plot
- Save a template of the plot
- Apply a template to the plot
- Define some page setup parameters for printing, and
- Print the plot.

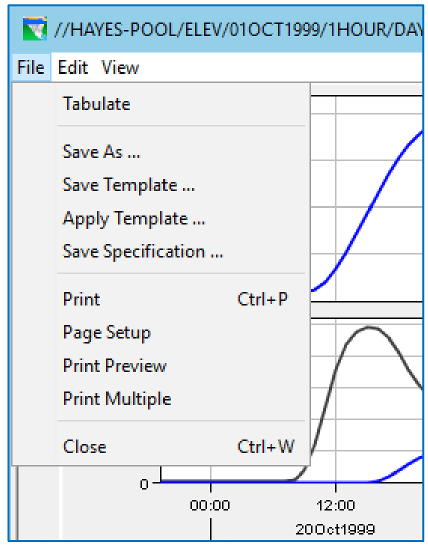


Figure E.1 Plot Window—File Menu

E.2 Saving a Plot to a File

To save a plot to a file:

- Select **Save As ...** from the plot's **File** menu.
- A **Save** file browser will appear (Figure E.2). Enter a **File name** and select the file format from the **Files of type** selector. The File format options are:
- Windows Metafile (*.wmf)
- Postscript (*.ps)
- JPEG (*.jpg, *.jpeg)
- Portable Network Graphics (*.png)

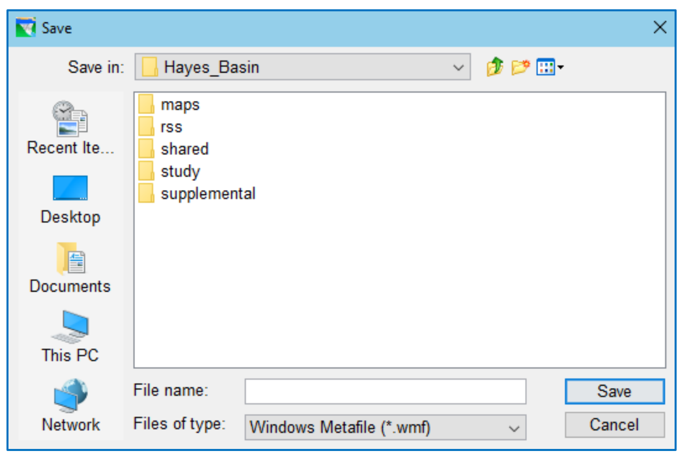


Figure E.2 Plot, Save As File Browser

E.2.1 Saving and Applying Plot Templates

Once you have customized an individual plot (by changing its line styles, adding marker lines and/or callouts, adjusting the axes, etc., you can save the settings you have made to the plot as a *Template* that you can apply to future plots.

E.2.1.1 Saving a Plot Template

To create a template based upon a customized plot:

- Select **Save Template...** from the plot's **File** menu.
- The **Export Plot Template** dialog appears (Figure E.3).
- Enter a **Name** for the new template and choose whether the template will be available for:
    - **All Applications**—this really means all watersheds.
    - **This Watershed only**
    - **All Users Watershed**
    - **All Users**.
- Click **OK** to save the template and close the dialog. Like user preferences (see Appendix A), the template is stored in your *AppData* area in a folder under this version of ResSim.

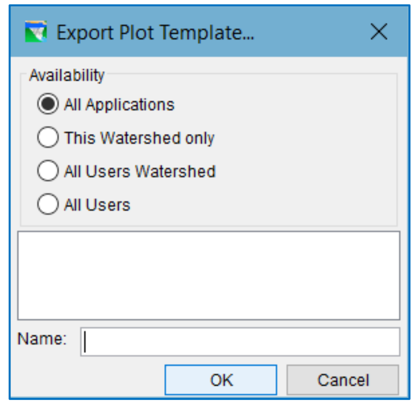


Figure E.3 Export Plot Template

Select from the first two options only. The second two options were designed for a multi-user, client-server version of ResSim.

E.2.1.2 Applying a Plot Template

To apply a template that you previously saved:

- Select **Apply Template** from a plot's **File** menu.
- Select the template from the **Import Plot Template** dialog that appears (Figure E.4).
- Click **OK** to apply the template to your current plot and close the dialog.

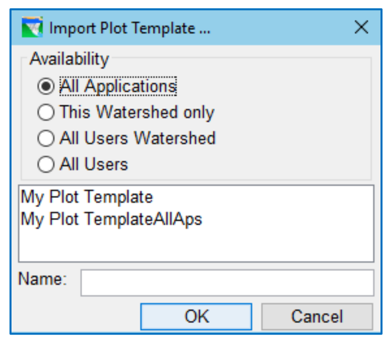


Figure E.4 Apply Plot Template

E.2.2 Preparing and Printing Plots

E.2.2.1 Page Setup for Printing Plots

The **Page Setup** option in a plot's **File** menu opens the **Page Setup** editor (Figure E.5), where you can:

- Set the page **Orientation**
- Add Page Numbers
- Set the **Printer Scale**, and

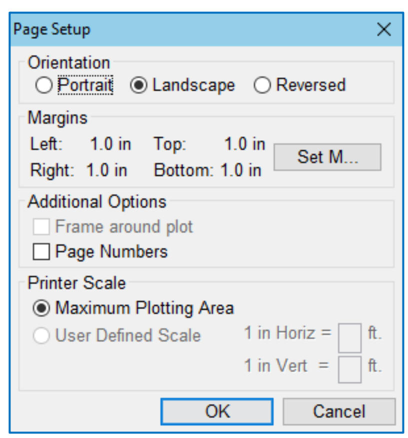


Figure E.5 Page Setup Edit

- Set the **Margins**. The **Set Margins** button opens the **Printer Margins** dialog (Figure E.6).

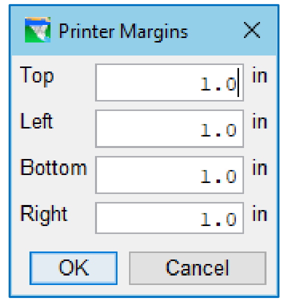


Figure E.6 Printer Margins

E.2.2.2 Previewing Printed Plots

The **Print Preview** command in the plot window's **File** menu allows you to view the plot as it will be printed. Figure E.7 shows an example of the **Print Preview** window showing the plot as it should look printed.

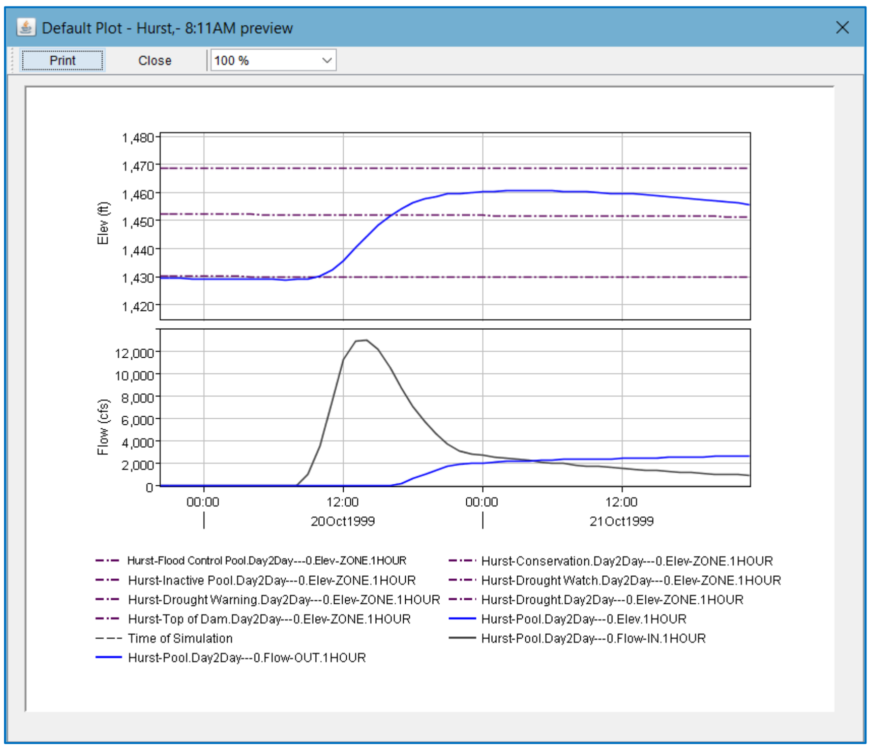


Figure E.7 Print Preview of a Plot

E.2.2.3 Printing Plots

The **Print** option in the plot window's **File** menu opens the **Print** dialog (Figure E.8), where you can choose your printer (by **Name**), set printer **Properties**, and specify the **Number of copies** to print. You can also select **Print to File** to print your plot to a **File** instead of to a printer.

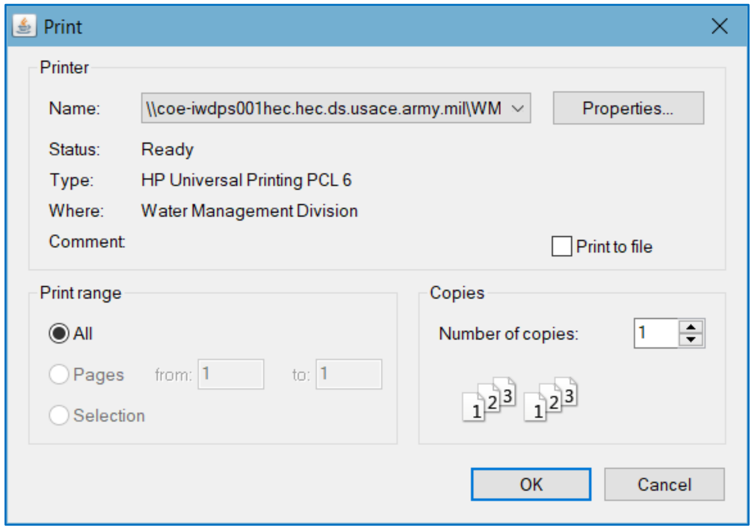


Figure E.8 Print Dialog

E.2.2.4 Printing Multiple Plots

The **File** menu of a Plot window also provides a **Print Multiple** option. With this feature, you can print several plots on one page.

The **Print Multiple** dialog (Figure E.9) shows all of the currently opened plots in the **Available Plots** pane. To select plots for printing, double-click on them and they will move to the **Selected Plots** pane.

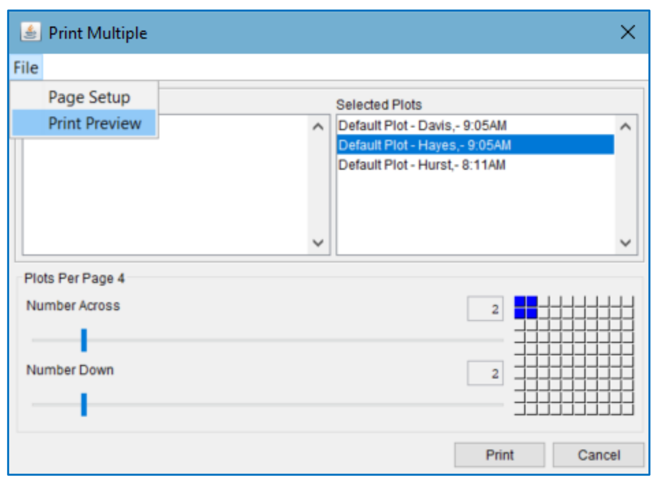


Figure E.9 Print Multiple (Plots) Dialog

Next, use the slider bars to specify the number of plots you wish to appear horizontally and vertically on the page. The grid to the right of the sliders reflects your choices.

Page Setup and **Print Preview** are available from the **File** menu of the **Print Multiple** dialog. Figure E.10 shows a print preview of multiple plots.

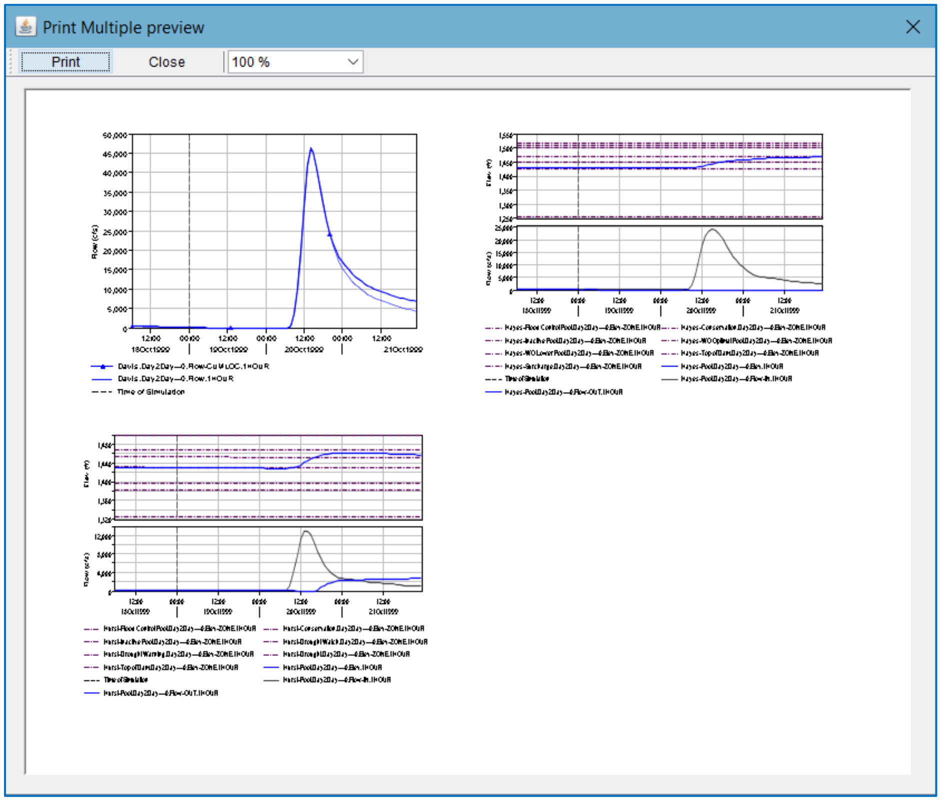


Figure E.10 Print Multiple Preview Window

E.3 Saving and Printing Tabulated Data

When tabulating data from a ResSim Plot or from DSSVue, the Table window's **File**, **Edit**, and context menus provide options for printing, exporting, and copying the contents of the table.

E.3.1 Copying Tabulated Data to the Clipboard

To copy tabulated data to the clipboard:

- Select the data you want to copy. To select the whole table:
    - Select **Select All** from the table window's Edit or context menus
    - Or, click anywhere in the table and type Ctrl-A.
- Then, to copy the selection to the clipboard:
    - Select **Copy** from the table window's **Edit** or context menu (Figure E.11), *or*
    - Or, type Ctrl-C.

The data you have copied to the clipboard is in *tab-separated* format. It can now be pasted into another application such as Microsoft® Excel or Word.

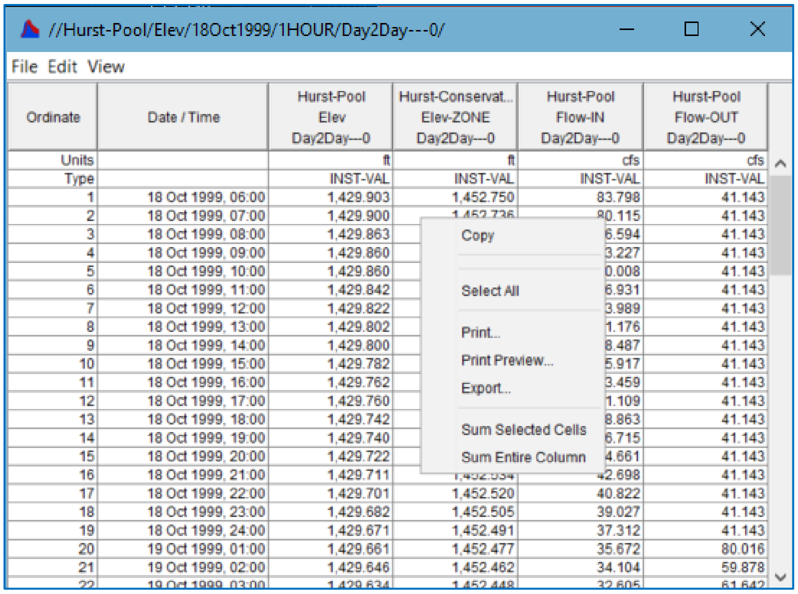
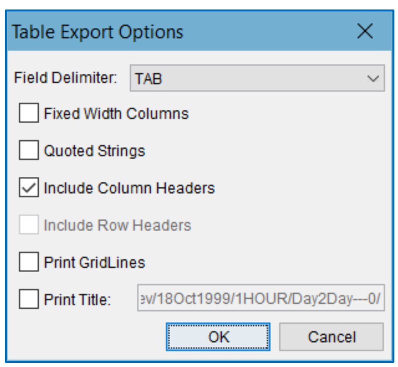


Figure E.11 Tabulated Data Window Showing Context Menu

E.3.2 Saving Tabulated Data to a File

To save (or export) tabulated data to a file:

- Select **Export** from the **File** or context menu of the table window. The **Table Export Options** dialog (Figure E.12) will open.



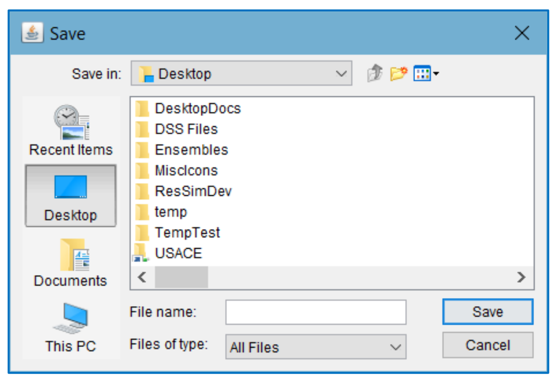


Figure E.12 Table Export Options *Figure E.13 Save File Browser*

- Select a **Field Delimiter** (Tab, Space, Comma, Colon, or Semi-Colon)
- Check or uncheck the various options for formatting the data, including:
  - o Fixed-Width Columns
  - o Quoted Strings
  - o Include Column and Row Headers
  - o Print Gridlines
  - o Print Title—you can override the default title by entering a new title in the text box.
- Click **OK** to accept your formatting selections and proceed. A **Save** file browser (Figure E.13) will open.

- Select the folder where you want to save the file and enter a **File name**.
- Click **Save** to complete the export, save the file, and close the browser.

E.3.3 Previewing Tabulated Data (before Printing)

To preview tabulated data before printing:

- Select **Print Preview** from the **File** or context menu of the table window (Figure E.14). The Print **Properties** editor will open.

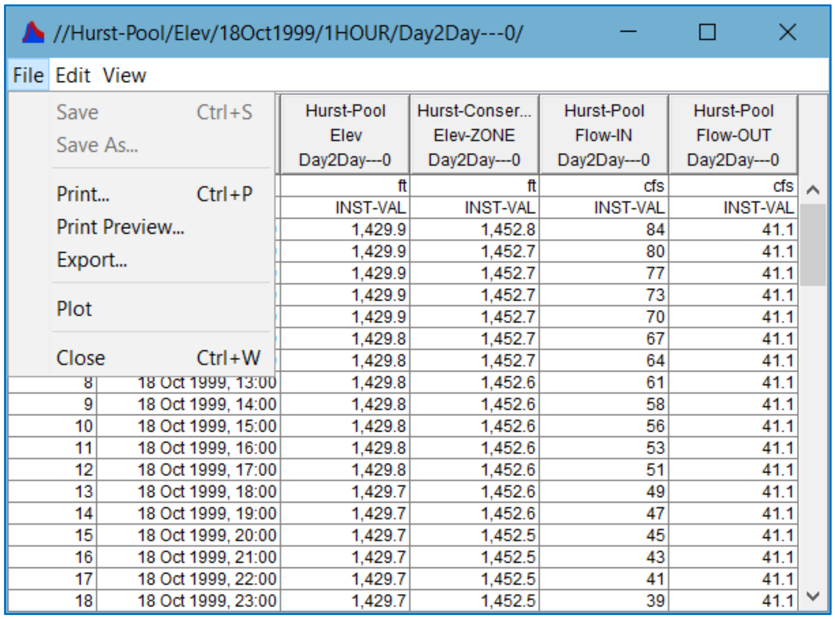


Figure E.14 Table Window—File Menu

- The Print **Properties** editor (Figure E.15) uses three tabs to organize the various properties you can configure:

  The **Page** tab allows you to specify the page **Orientation**, **Scaling**, and **Selection**. You can also choose to print the table as **ASCII**, to **Repeat Headers** on every page, and print the **Gridlines**.

  The **Header/Footer** tab provides text boxes in which you can type the header and footer you want to appear on your printed pages.

  The **Table Title** tab offers a default title for the table based on the data source. You may edit this title.

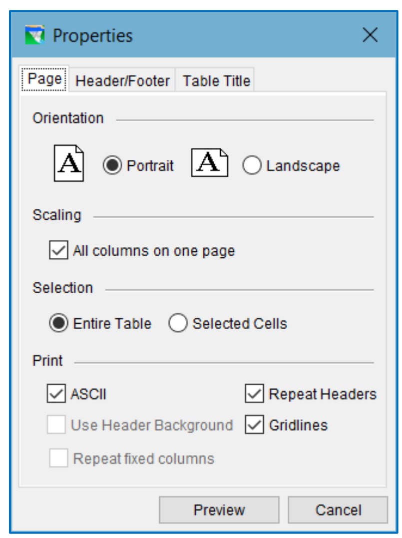
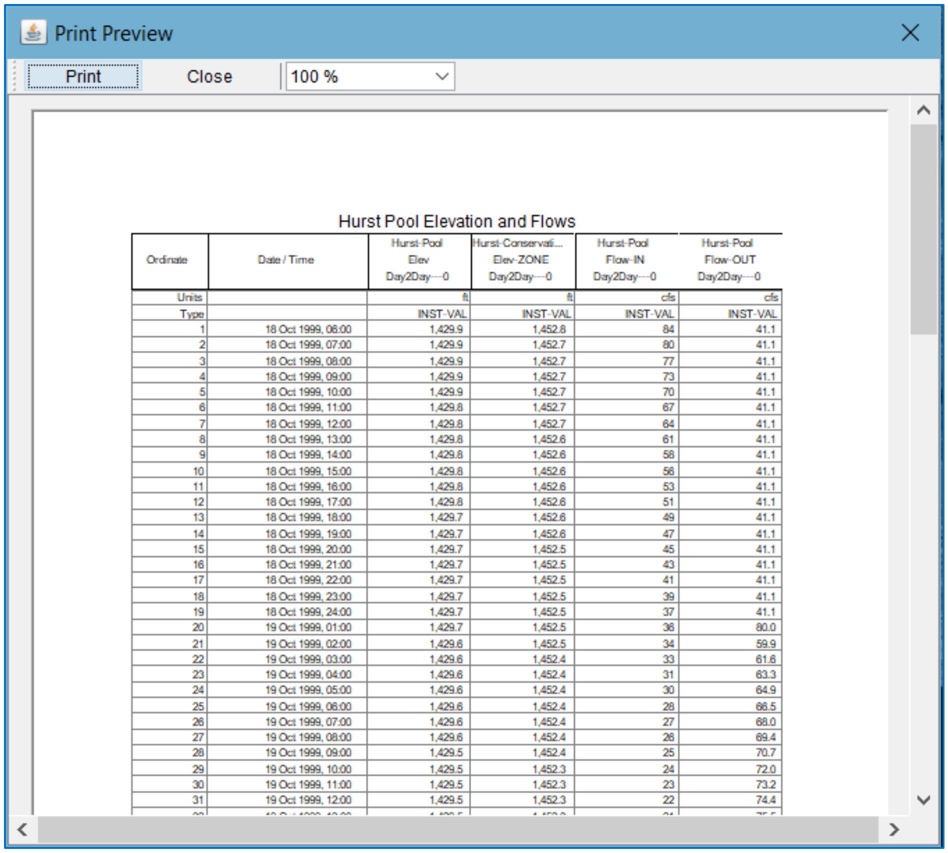


Figure E.15 Print Properties Editor from Print Preview

- After you have configured the Print **Properties**, click the **Preview** button.
- A **Print Preview** window (Figure E.16) will open, allowing you to view the tabulated data as it will be printed.



Hurst Pool Elevation and Flows

Ordinate	Date / Time	Hurst-Pool Elev Day2Day---0	Hurst-Conservati... Elev-ZONE Day2Day---0	Hurst-Pool Flow-IN Day2Day---0	Hurst-Pool Flow-OUT Day2Day---0
Units		ft	ft	cfs	cfs
Type		INST-VAL	INST-VAL	INST-VAL	INST-VAL
1	18 Oct 1999, 06:00	1,429.9	1,452.8	84	41.1
2	18 Oct 1999, 07:00	1,429.9	1,452.7	80	41.1
3	18 Oct 1999, 08:00	1,429.9	1,452.7	77	41.1
4	18 Oct 1999, 09:00	1,429.9	1,452.7	73	41.1
5	18 Oct 1999, 10:00	1,429.9	1,452.7	70	41.1
6	18 Oct 1999, 11:00	1,429.8	1,452.7	67	41.1
7	18 Oct 1999, 12:00	1,429.8	1,452.7	64	41.1
8	18 Oct 1999, 13:00	1,429.8	1,452.6	61	41.1
9	18 Oct 1999, 14:00	1,429.8	1,452.6	58	41.1
10	18 Oct 1999, 15:00	1,429.8	1,452.6	56	41.1
11	18 Oct 1999, 16:00	1,429.8	1,452.6	53	41.1
12	18 Oct 1999, 17:00	1,429.8	1,452.6	51	41.1
13	18 Oct 1999, 18:00	1,429.7	1,452.6	49	41.1
14	18 Oct 1999, 19:00	1,429.7	1,452.6	47	41.1
15	18 Oct 1999, 20:00	1,429.7	1,452.5	45	41.1
16	18 Oct 1999, 21:00	1,429.7	1,452.5	43	41.1
17	18 Oct 1999, 22:00	1,429.7	1,452.5	41	41.1
18	18 Oct 1999, 23:00	1,429.7	1,452.5	39	41.1
19	18 Oct 1999, 24:00	1,429.7	1,452.5	37	41.1
20	19 Oct 1999, 01:00	1,429.7	1,452.5	36	80.0
21	19 Oct 1999, 02:00	1,429.6	1,452.5	34	59.9
22	19 Oct 1999, 03:00	1,429.6	1,452.4	33	61.6
23	19 Oct 1999, 04:00	1,429.6	1,452.4	31	63.3
24	19 Oct 1999, 05:00	1,429.6	1,452.4	30	64.9
25	19 Oct 1999, 06:00	1,429.6	1,452.4	28	66.5
26	19 Oct 1999, 07:00	1,429.6	1,452.4	27	68.0
27	19 Oct 1999, 08:00	1,429.6	1,452.4	26	69.4
28	19 Oct 1999, 09:00	1,429.5	1,452.4	25	70.7
29	19 Oct 1999, 10:00	1,429.5	1,452.3	24	72.0
30	19 Oct 1999, 11:00	1,429.5	1,452.3	23	73.2
31	19 Oct 1999, 12:00	1,429.5	1,452.3	22	74.4

Figure E.16 Print Preview of a Table (Example)

- Click the **Print** button at the top of the **Print Preview** window to proceed with printing your data. The **Print** Dialog will open.
- Here you can:
  - o Choose your Printer (by Name)
  - o Set *Printer* Properties
  - o Specify the Number of copies to print
  - o Decide to Print to file rather than to a printer.
- Click **OK** to print your tabulated data.

How you configure (size) the columns of your table in the table window *will* affect how your tabulated data looks when printed. Be sure to use Print Preview to be sure things look right before printing.

E.3.4 Printing Tabulated Data

To print tabulated data:

- Select **Print** from the **File** or context menu of the table window (Figure E.17). The Print **Properties** editor will open.
- See Section E.3.3 above for details in setting the print properties.
- When you have configured your print properties, click the **Print** button.
- The **Print Dialog** will open.
- Choose your printer then click **OK** to print your data.

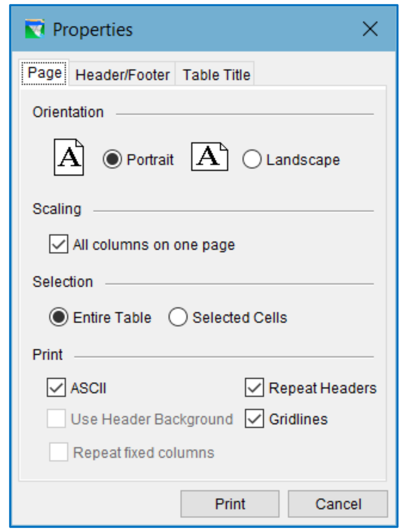


Figure E.17 Print Properties Editor

E.4 Printing and Exporting ResSim Reports

Each module in ResSim includes a Reports menu from which you can open a variety of reports that you can use to view your ResSim data and results. And, just like DSSVue's tabulated data windows, ResSim's reports all provides options for printing and exporting.

All ResSim reports have either a **File** menu or a **Report** menu. In all these menus, you will find a **Print** option. In most, you will also find a **Save As...** or **Export** option. In addition, if you right-click in the table area of any ResSim report, the context menu will include options for **Print**, **Print Preview**, and **Export**.

To **Preview** a ResSim report before printing:

- Select **Print Preview** from the report's context menu.
- The Print **Properties** editor will open. Proceed as you would for previewing and printing tabulated data. See Section E.3.3 for full details.

To **Print** a ResSim report:

- Select **Print** from the report's **File** or **Report** menu or from its context menu.
- The Print **Properties** editor will open. Proceed as you would for printing tabulated data. See Section E.3.4 for further details.

To **Export** a ResSim report:

- Select **Save As...** or **Export...** from the report's **File** or **Report** menu or **Export...** from its context menu.
- The **Table Export Options** dialog will open. Proceed as you would for saving tabulated data to a file. See Section E.3.2 for full details.

Appendix F
Yield Analysis

In the water resources field, the concept of yield can be understood a few different ways. USACE defines yield as follows:

> "Yield is the amount of water that can be supplied from the reservoir to a specified location and in a specified time pattern" (EM 1110-2-1420) across a period of streamflow.

Yield includes all water supplied, possibly at differing rates. Reservoir or system *firm yield* is an additional important calculation used for water supply studies. USACE defines firm yield as follows:

> "Firm yield is the largest consistent flow rate (demand) that can be provided [without fail] throughout a period of historic stream-flow" (EM 1110-2-1420).

Firm yield estimates the amount of water that can *always* be supplied. The ability to store water increases the firm yield by allowing water to be saved and used when streamflow is less than demand. Thus, the process of identifying firm yield also identifies the **critical period** as the driest in the record that identifies the firm yield for a given storage volume. Greater storage volume produces greater firm yield by providing the ability to supply more water to meet demand during a drier period or a longer dry period. A different available storage volume might have a different critical period for a given period of streamflow.

A modeler can manually perform a yield analysis in ResSim by iteratively computing a period-of-record simulation until the yield has been identified. With each iteration, the modeler makes a small change to the demand. This process continues until a run is made that just barely empties the water supply storage in the reservoir. That level of demand is the yield for the period-of-record conditions.

The **Yield Analysis** alternative type, added in ResSim 3.3, automates the iteration process and makes calculation of yield a more easily reproducible. There are two types of yield analysis currently implemented in ResSim: **Reservoir Yield** and **Storage Account Yield**. Reservoir Yield uses the entire conservation pool of the reservoir to meet the demand and can be calculated for an individual reservoir or for a system of reservoirs. Storage Account Yield calculates the yield of a single water storage account among multiple accounts at a reservoir. The ResSim **Water Accounts** feature allows you to divide the total conservation storage into multiple storage accounts based on either volume or percent of pool. ResSim can then track water use for each account and account holder based on the rules attached to it.

The ResSim yield feature can be used to calculate firm yield for water supply or hydropower, or it can be used to calculate a specific Exceedance Drought Yield by using a particular time series of inflows. This chapter covers the use of the ResSim Yield Analysis feature for the purpose of calculating water supply firm yield.

F.1 ResSim Yield Analysis Feature

The ResSim Yield Analysis feature automates the iterative work necessary to determine firm yield. A single minimum or specified flow rule (or downstream control rule) is used to represent the demand at the location of interest; the demand can be constant or seasonally-varying (though repeating each year). The yield is computed by repeatedly adjusting the value of the demand rule and re-simulating until the conservation pool is emptied once during the period-of-record; seasonally-varying demands are scaled proportionally during the search process.

After each simulation, the demand is increased or decreased to get closer to the goal of *exactly* emptying the pool *once*, with no failure to meet the minimum rule. The bisection search method is one method used to determine the next demand to try, and there is also an option to use a heuristic search based on mass balance to attempt to converge in fewer iterations. The bisection method is more reliable for finding the firm yield, but typically requires more iterations. The heuristic approach looks at the remaining storage and demand volumes and computes an estimate of exactly-meetable demand for the next iteration. While it requires fewer iterations than the Bisection method, the Heuristic method is currently successful only in fairly simple, single-reservoir watershed analyses.

The ResSim Yield Analysis Tool also allows the user to set flow rate and storage tolerances for defining convergence, to limit the number of iterations needed. The maximum number of iterations can also be directly limited.

A Yield Analysis is performed as part of a special ResSim alternative Type, a Yield Analysis Alternative. To create a Yield Analysis Alternative, you must set the Alternative Type to Yield Analysis on the Run Control tab of the Alternative Editor as illustrated in Figure F.1. When the alternative type is **Yield Analysis**, the content of the **Yield Analysis** tab will be enabled as illustrated in Figure F.2.

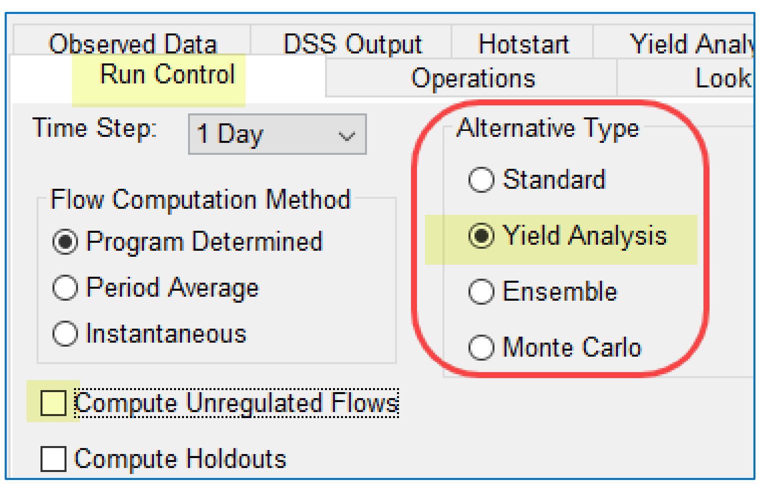


Figure F.1 Alternative Editor—Run Control Tab—Yield Analysis Alternative Type

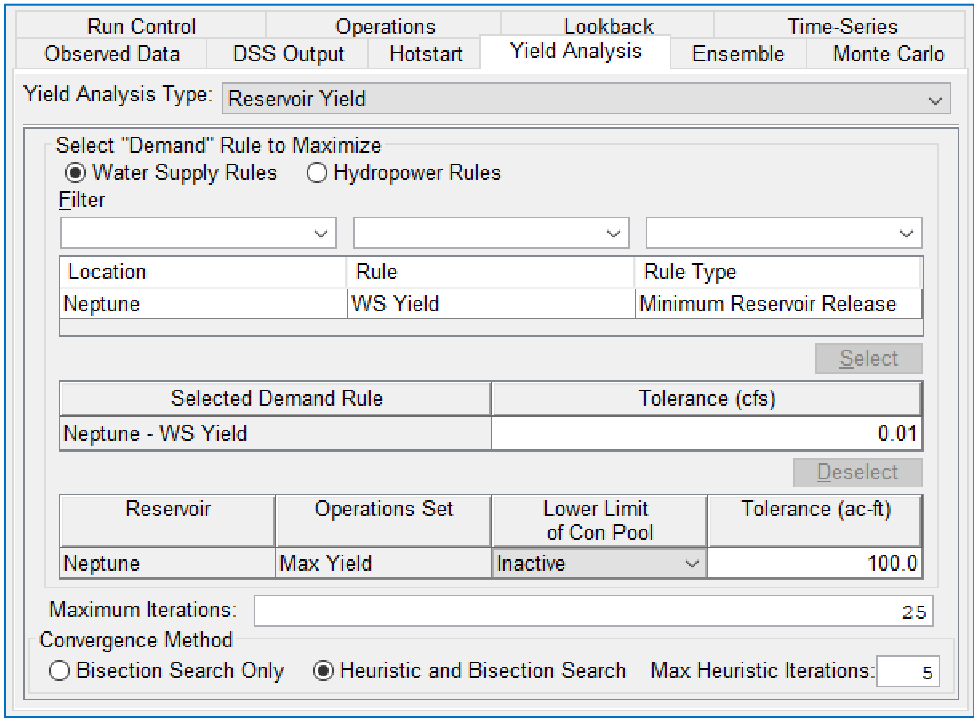


Figure F.2 Alternative Editor—Yield Analysis Tab with Reservoir Yield selected

The Yield Analysis tab has two views depending on the selection of the **Yield Analysis Type** at the top of the tab. As illustrated in Figure F.3, the **Yield Analysis Type** options include:

Reservoir Yield for the total yield from a reservoir pool or

Water Account Yield for yield based on a specified water storage account within a reservoir.

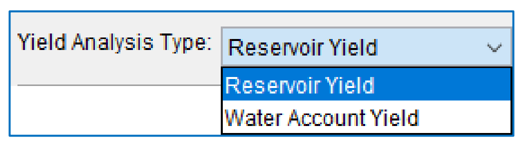


Figure F.3 Yield Analysis Types

The edit panel for each **Yield Analysis Type** is described in the next two sections.

F.1.1 Reservoir Yield Analysis

The edit panel for the **Reservoir Yield** analysis type is shown in Figure F.2. The process for setting up a **Reservoir Yield Analysis** proceeds from the top down in the edit panel. Most of the panel involves selecting the Demand Rule. The remainder of the panel provides options for setting the tolerances, maximum number of iterations, and the convergence method.

Select "Demand" Rule to Maximize—You must select one minimum limit or specified limit type rule to represent demand from the pool. The value of the rule will be maximized in order to determine the pool's firm yield. This rule may be a Release Function rule, Downstream Control Function rule or Hydropower rule.

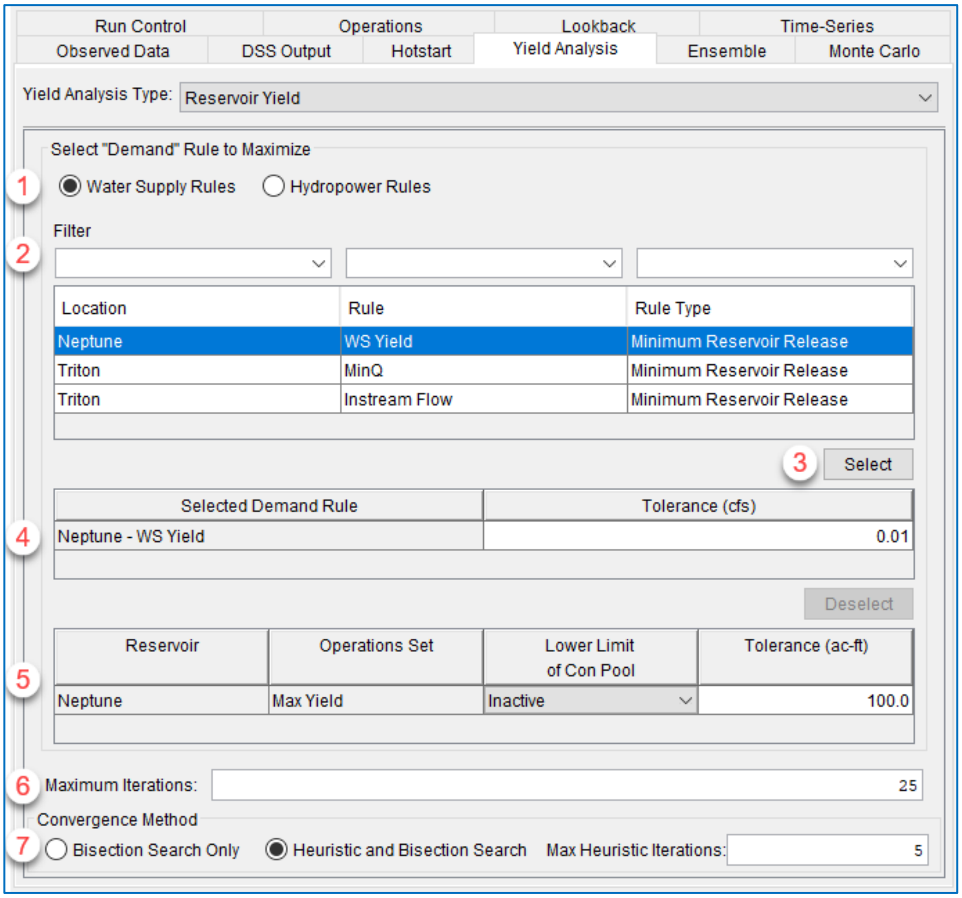


Figure F.4 Yield Analysis Tab—Reservoir Yield View

1. **Rule Category Radio Buttons**—select a rule category and the table below will be filled with all available rules of the selected category. The list of rules will be drawn from the active operation sets for the reservoirs in the current alternative.

    Water Supply Rules—minimum or specified Release Function and Downstream Control Function rules

    Hydropower Rules—system and local Hydropower rules

2. **Filter**—(optional) use the filter dropdown boxes to filter the list of rules based on **Location** (reservoir), **Rule** name, or **Rule Type**.

3. Highlight the desired rule in the rule list and *click* **Select.** This will place your selected rule into the **Selected Demand Rule** table. It will also place the **Reservoir** and **Operation Set** into the last table on the edit panel, the Storage Parameters table.

4. **Selected Demand Rule**—set the tolerance for the rule.

    Tolerance—this is the *flow* **Tolerance** used for the determination of the yield analysis convergence. The flow tolerance is the maximum tolerable limit for the demand being shorted. If the computed demand delivery in a given yield iteration is below the requirement by more than the tolerance, the iteration is considered "failed." Both the flow and storage tolerance criteria must pass in the same iteration for the yield analysis to reach final convergence.

5. **Reservoir** Storage Table**—**displays reservoir and active operations set. You set the pool lower limit and storage tolerance. Note that this table refers to the "Con Pool" but the storage pool used for the yield calculation is actually defined by the Guide Curve, which may or may not be the top of your conservation pool.

   Lower Limit of Con Pool—Use the selector to choose the top of zone curve that marks the bottom of the storage pool used to determine yield. The top of this storage pool (usually referred to as the Conservation pool) is by default the reservoir's Guide Curve. Note that the storage pool defined here may encompass more than one zone below the Guide Curve.

   Tolerance—this is the *storage* **Tolerance** used for the determination of the yield analysis convergence. The storage tolerance defines how close above the bottom of the conservation pool (defined as "Lower Limit of Con Pool") the computed minimum storage may reach to be acceptable for the analysis. If the computed minimum storage in an iteration is greater than the bottom plus tolerance, it is a failed iteration. If the reservoir empties below the lower limit of the conservation pool, it is also considered to be a failed iteration. Both the flow and storage tolerance criteria must pass in the same iteration for the yield analysis to reach final convergence.

6. **Maximum Iterations**—this is maximum number of iterations that the yield analysis may perform in its attempt to converge on a solution. The default value is 25. If the simulation is unable to converge within the specified flow and storage tolerances you entered but has reached the maximum iterations, it will save the answer from the last iteration and give a warning in the Compute Log that the convergence was not achieved after completing the maximum number of iterations.

7. **Convergence Method**—a radio button provides the choice of either **Bisection Search Only** or **Heuristic and Bisection Search**, and the second option allows you to specify a value for the **Maximum Heuristic Iterations** that will be performed before the convergence method automatically switches to the Bisection search. The heuristic method uses a mass balance approach; it calculates the storage remaining (or the volume of water shorted, if storage went to zero) from the prior iteration to estimate the next demand to use while attempting convergence. The combined heuristic and bisection search may narrow in on the final result faster than with Bisection Search only. The heuristic search can diverge in multiple reservoir cases, so it is used for a limited number of iterations before switching to the bisection method for the remaining iterations. Typically, the heuristic search can approach the final result within a few iterations, and the bisection method adds to the precision of the final answer if the maximum number of iterations for the heuristic method is exceeded. The combined

Heuristic and Bisection Search is currently meant for single-reservoir watersheds and is less successful for more complex reservoir systems and operations.

F.1.2 Water Account Yield Analysis

The edit panel for the **Water Account** yield analysis type is shown in Figure F.5. The process for setting up a **Water Account Yield Analysis** proceeds from the top down in the edit panel.

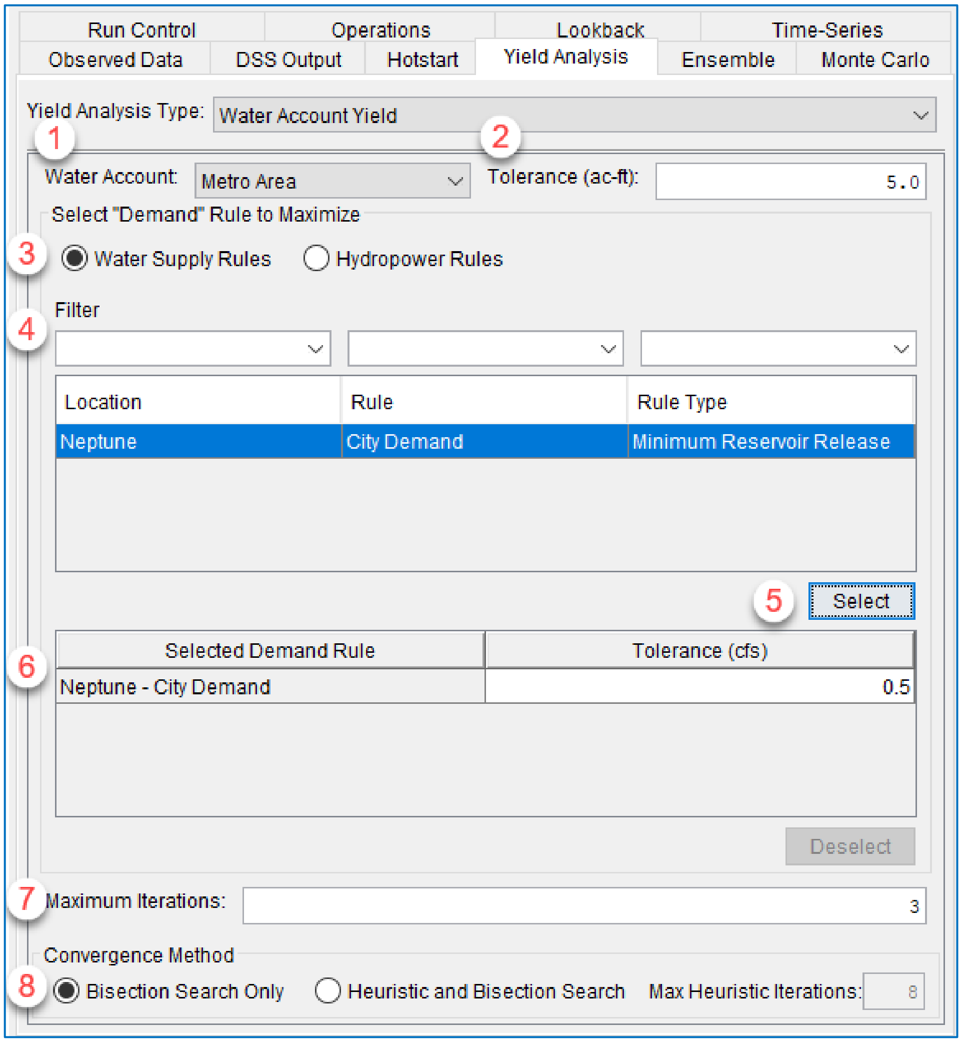


Figure F.5 Yield Analysis Tab—Water Supply Yield View

1. **Water Account—**Select the water account for which you want to compute the maximum yield. In order for there to be anything in the list to choose from, you must have—1) created a **Water Account Set** with at least one water account in it, and 2) selected that water account set on the operations tab of this yield alternative. See Section 14.8 of the main User's Manual.

2. **Tolerance**—this is the water account *storage* **Tolerance** used for the determination of the water account yield analysis convergence. The storage tolerance defines how close to zero the water account's storage must reach to be acceptable for the analysis. If the computed minimum water account storage in a given iteration is above the tolerance, it is a failed iteration. Both the flow and storage tolerance criteria must pass in the same iteration for the water account yield analysis to reach final convergence.

Select "Demand" Rule to Maximize—You must select a minimum limit or specified limit type rule attached to the water account to maximize its firm yield. This rule may be a Release Function rule, Downstream Control Function rule, or Hydropower rule. Like Steps 1-3 in the process of setting up a Reservoir Yield calculation, Steps 3-5 are for selecting a rule category, filtering, and selecting the demand rule. However, since each water account can only have one rule attached to it, there will only be one rule available to choose.

3. **Rule Category Radio Buttons**—select the rule category that matches the rule attached to the water account (most likely Water Supply Rules).

   Water Supply Rules—minimum or specified Release Function and Downstream Control Function rules.

   Hydropower Rules—system and local Hydropower rules

4. **Filter**—these boxes allow you to filter the list of rules based on **Location** (reservoir), **Rule** name, or **Rule Type**. This step is not needed for the Water Account Yield option.

5. Highlight the desired rule in the rule list and *click* **Select.** This will place your selected rule into the **Selected Demand Rule** table.

6. **Selected Demand Rule**—set the flow tolerance for the rule.

   Tolerance—this is the *flow* **Tolerance** used for the determination of the water account yield analysis convergence. The flow tolerance is the maximum tolerable limit for the water account's demand rule being shorted. If the computed demand delivery in a given yield iteration is below the rule requirement by more than the tolerance, the iteration is considered "failed." Both the flow and storage tolerance criteria must pass in the same iteration for the water account yield analysis to reach final convergence.

7. **Maximum Iterations**—this is maximum number of iterations that the yield analysis may perform in its attempt to converge on a solution. The default value is 25.

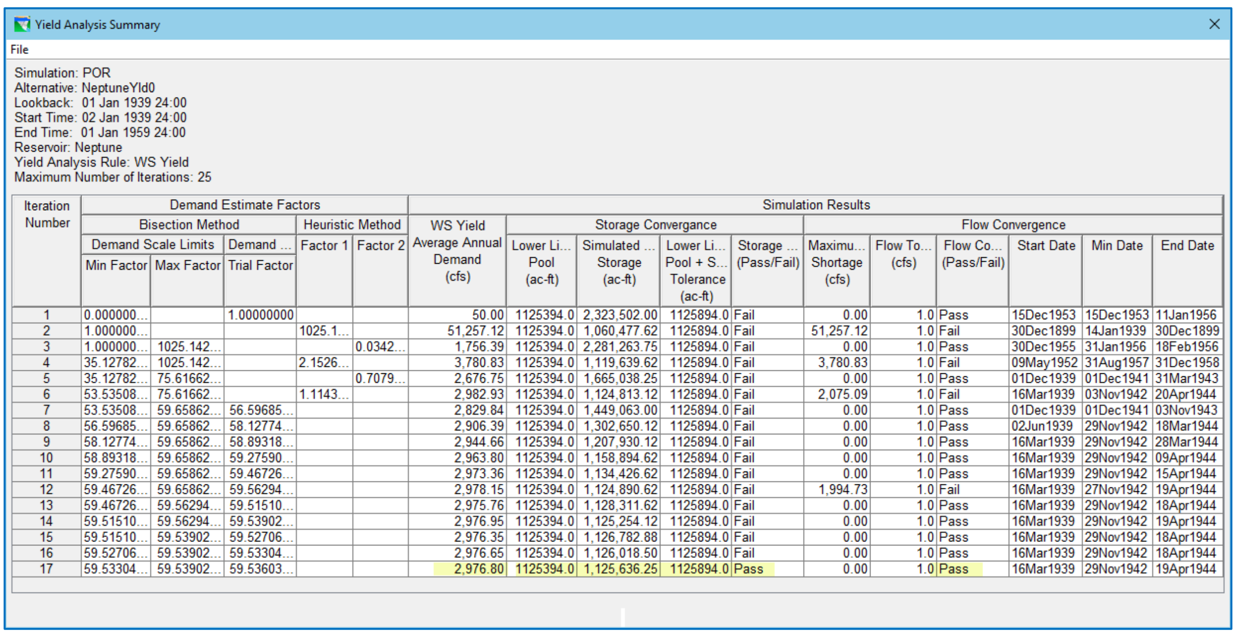
8. **Convergence Method**—this radio button provides the choice of either **Bisection Search Only** or **Heuristic and Bisection Search**. See the previous section for a detailed description.

F.2 Yield Analysis Summary Report

While the reservoir plot showing storage levels and releases is the first and most important result to study (Section F.3.4), ResSim also creates a **Yield Analysis Summary** report when a Yield Alternative type has been computed. The Yield Analysis Summary can be accessed from the **Reports** menu of the **Simulation** module. From the **Reports** menu, select **Storage Yield Analysis** > [yield alternative name] > **Output Summary Report** (Figure F.6).

Figure F.6 Reports menu—Yield Analysis Report Option

The resulting Yield Analysis Summary report (Figure F.7) includes **Simulation Run** identification information at the top followed by an **Iteration Summary** table with demand estimate scaling factors and simulation convergence results per iteration.



Yield Analysis Summary

File

Simulation: POR
Alternative: NeptuneYld0
Lookback: 01 Jan 1939 24:00
Start Time: 02 Jan 1939 24:00
End Time: 01 Jan 1959 24:00
Reservoir: Neptune
Yield Analysis Rule: WS Yield
Maximum Number of Iterations: 25

Iteration Number	Demand Estimate Factors					Simulation Results										
	Bisection Method			Heuristic Method		WS Yield Average Annual Demand (cfs)	Storage Convergance				Flow Convergence					
	Demand Scale Limits		Demand ...	Factor 1	Factor 2		Lower Li... Pool (ac-ft)	Simulated ... Storage (ac-ft)	Lower Li... Pool + S... Tolerance (ac-ft)	Storage ... (Pass/Fail)	Maximu... Shortage (cfs)	Flow To... (cfs)	Flow Co... (Pass/Fail)	Start Date	Min Date	End Date
	Min Factor	Max Factor	Trial Factor													
1	0.000000...		1.00000000			50.00	1125394.0	2,323,502.00	1125894.0	Fail	0.00	1.0	Pass	15Dec1953	15Dec1953	11Jan1956
2	1.000000...			1025.1...		51,257.12	1125394.0	1,060,477.62	1125894.0	Fail	51,257.12	1.0	Fail	30Dec1899	14Jan1939	30Dec1899
3	1.000000...	1025.142...			0.0342...	1,756.39	1125394.0	2,281,263.75	1125894.0	Fail	0.00	1.0	Pass	30Dec1955	31Jan1956	18Feb1956
4	35.12782...	1025.142...		2.1526...		3,780.83	1125394.0	1,119,639.62	1125894.0	Fail	3,780.83	1.0	Fail	09May1952	31Aug1957	31Dec1958
5	35.12782...	75.61662...			0.7079...	2,676.75	1125394.0	1,665,038.25	1125894.0	Fail	0.00	1.0	Pass	01Dec1939	01Dec1941	31Mar1943
6	53.53508...	75.61662...		1.1143...		2,982.93	1125394.0	1,124,813.12	1125894.0	Fail	2,075.09	1.0	Fail	16Mar1939	03Nov1942	20Apr1944
7	53.53508...	59.65862...	56.59685...			2,829.84	1125394.0	1,449,063.00	1125894.0	Fail	0.00	1.0	Pass	01Dec1939	01Dec1941	03Nov1943
8	56.59685...	59.65862...	58.12774...			2,906.39	1125394.0	1,302,650.12	1125894.0	Fail	0.00	1.0	Pass	02Jun1939	29Nov1942	18Mar1944
9	58.12774...	59.65862...	58.89318...			2,944.66	1125394.0	1,207,930.12	1125894.0	Fail	0.00	1.0	Pass	16Mar1939	29Nov1942	28Mar1944
10	58.89318...	59.65862...	59.27590...			2,963.80	1125394.0	1,158,894.62	1125894.0	Fail	0.00	1.0	Pass	16Mar1939	29Nov1942	09Apr1944
11	59.27590...	59.65862...	59.46726...			2,973.36	1125394.0	1,134,426.62	1125894.0	Fail	0.00	1.0	Pass	16Mar1939	29Nov1942	15Apr1944
12	59.46726...	59.65862...	59.56294...			2,978.15	1125394.0	1,124,890.62	1125894.0	Fail	1,994.73	1.0	Fail	16Mar1939	27Nov1942	19Apr1944
13	59.46726...	59.56294...	59.51510...			2,975.76	1125394.0	1,128,311.62	1125894.0	Fail	0.00	1.0	Pass	16Mar1939	29Nov1942	18Apr1944
14	59.51510...	59.56294...	59.53902...			2,976.95	1125394.0	1,125,254.12	1125894.0	Fail	0.00	1.0	Pass	16Mar1939	29Nov1942	19Apr1944
15	59.51510...	59.53902...	59.52706...			2,976.35	1125394.0	1,126,782.88	1125894.0	Fail	0.00	1.0	Pass	16Mar1939	29Nov1942	18Apr1944
16	59.52706...	59.53902...	59.53304...			2,976.65	1125394.0	1,126,018.50	1125894.0	Fail	0.00	1.0	Pass	16Mar1939	29Nov1942	18Apr1944
17	59.53304...	59.53902...	59.53603...			2,976.80	1125394.0	1,125,636.25	1125894.0	Pass	0.00	1.0	Pass	16Mar1939	29Nov1942	19Apr1944

Figure F.7 Yield Analysis Summary Report

The **Simulation Run** identification information includes:

- **The Simulation** name,
- **The Alternative** name,
- The Simulation Time Window: **Lookback**, **Start**, and **End** Dates and Times,
- The **Reservoir** name for which yield was calculated,
- **Yield Analysis Rule** name, and
- The **Maximum Number of Iterations** setting.

The **Iteration Summary** table has a row for each compute iteration describing the progress toward convergence of flow and storage. The results describe the scaling factors for the demand rule, resulting storage pools, flow and storage shortages or excesses, critical period dates, and whether the iteration passed or failed the flow and storage tolerances. There are two sets of columns:

Demand Estimate Factors—this set of columns is broken into two subsections that detail the two search methods and their search limits and/or scaling factors used to adjust the demand estimates.

Bisection Method—the data list includes **Demand Scale Limits** (**Min** and **Max Factors**) and the resulting **Trial Factor**. The **Trial Factor** is the bisection (average) of the **Min Factor** and **Max Factor**, and it is the value used to scale the *initial* demand value of the rule. (The first iteration uses the initial demand value with a trial factor of 1.0.)

Heuristic Method—these columns list the scaling factors used by this method. The heuristic method scaling factors are applied to the *previous* demand estimate, rather than the *initial* demand estimate. If the previous demand estimate was too small, the new scaling factor used is expressed under the **Factor 1** column, and if the previous demand estimate was too large, the new scaling factor is listed in the **Factor 2** column.

Simulation Results—this set of columns is broken into three subsets of columns:

WS Yield Average Annual Demand—the demand value selected for that iteration. The demand value represents the constant flow rate of the rule (or the average value if the rule criteria is time-varying). It is calculated using either the initial (bisection method) or previous (heuristic method) demand value multiplied by the current scaling factor.

Storage Convergence—these columns include:

Lower Limit of Con Pool—the value of the lower storage limit of the conservation pool, as defined in the Yield Analysis tab of the Alternative Editor. This value does not change across iterations. If the lower limit zone varies seasonally, the lowest value of the seasonal pool is considered the lower limit.

Simulated Minimum Storage—the minimum storage reached during that iteration.

Lower Limit of Con Pool + Storage Tolerance—the maximum volume of water that can be left in the reservoir that will satisfy the convergence tolerance when the simulated minimum storage is reached. This value defines the upper limit of the storage tolerance and does not vary across iterations.

Storage Convergence Test (Pass/Fail)—The storage convergence test passes when the minimum simulated storage falls within the storage tolerance. The storage convergence test fails when the minimum simulated storage falls below the **Lower Limit of the Con Pool**, or when the minimum simulated storage is greater than the **Lower Limit of the Con Pool + Storage Tolerance**.

Flow Convergence—these columns include:

Maximum Demand Shortage—the greatest shortage to the demand over the period of record. If demand was always met during an iteration, the maximum demand shortage will be zero.

Flow Tolerance—Maximum allowable shortage. This value does not change across iterations.

Flow Convergence Test (Pass/Fail)—The flow convergence test passes when the maximum demand shortage is zero or below the flow tolerance.

Start Date—the date on which the critical period began**—**when the pool was last at the top of the guide curve before reaching the minimum storage.

Min Date—the date when the pool was at its minimum value.

End Date—the end of the critical period—when the pool has fully recovered and is again at the top of the guide curve.

F.3 Example Firm Yield Analysis for a Reservoir Pool

The firm yield is the demand that can be *just barely* satisfied by inflow and storage through the driest period experienced (period of record hydrology) or expected (synthetic hydrology). Firm Yield is limited by a critical period which can vary based on the demand and storage capacity.

This example covers one way to conduct a firm yield analysis for a reservoir pool. Future documentation will describe other approaches and offer examples for doing water account yield analysis. To begin a firm yield analysis for a reservoir, an initial network and base alternative are needed, along with inflows for the period of record, or a comparable synthetic record. Typically, it is best to start with an existing alternative that has been shown to compute correctly through the period of record. These instructions assume that you are starting with an existing, well-reviewed watershed that contains an alternative that represents "current conditions".

F.3.1 Create a Base Network and Alternative

Begin by identifying an existing network and alternative that represent the current conditions. This "base network" will begin as a copy of the current conditions network. Later, you will simplify this network and then add some physical and/or operational features to it. The "base alternative" is a new alternative created for the base network but using the current conditions alternative as a template. You will use the base alternative to test that the model is running as expected before adding the yield operation.

F.3.1.1 Create a Base Network for the Yield Analysis

- In the **Reservoir Network** module, open the current conditions network.
- From the **Network** menu, select **Save As...** to create a copy of the current conditions network (Figure F.8). Give the new network an appropriate name.

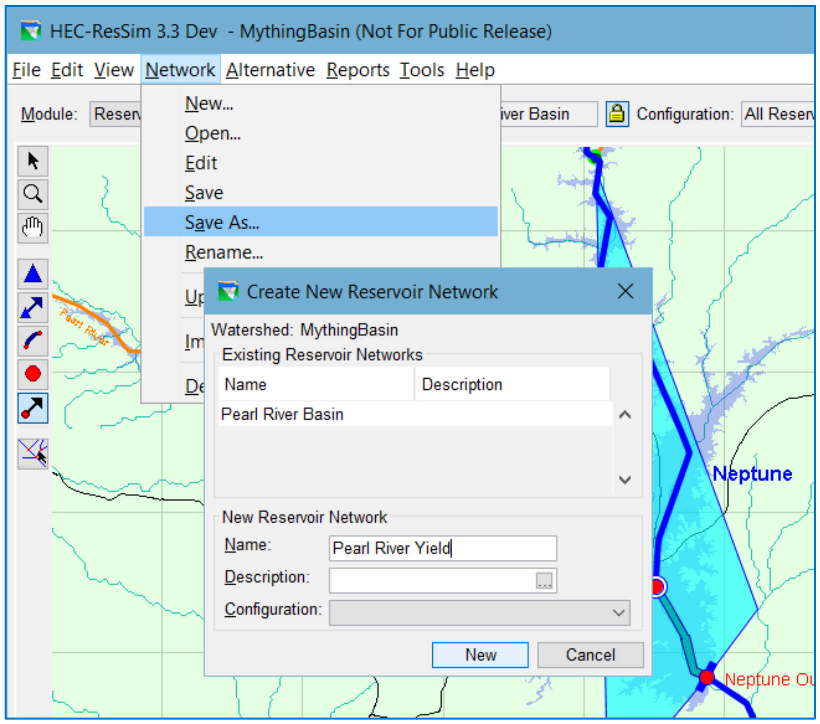


Figure F.8 New Network

F.3.1.2 Create a Base Alternative for the Yield Analysis

- Next, create a *new* alternative based on the *new* network (Figure F.9). This alternative will be the base alternative for all the yield alternatives you may create.
- Copy the setup of the existing current conditions alternative into the new alternative—this is easier to do if you haven't made any changes to the network yet.
- Start with the **Run Control** tab. Set the **Time Step** and the **Flow Computation Method** to the same settings as in the template (current conditions) alternative. Set the **Alternative Type** to **Standard**. Also, consider the settings for **Compute Unregulated Flow** and **Compute Holdouts**. You obviously will *not* need **Holdouts** for this alternative and you probably won't need **Unregulated Flows** either (unless there's something in the operations that uses them); if you can, set both

- of these options to OFF (unchecked) so that their computes do not add unnecessarily to the compute time of each iteration of your alternative.
- Next, on the **Operations** tab, set the **Operation Set** for each reservoir in the network using the same settings as those in the template alternative.

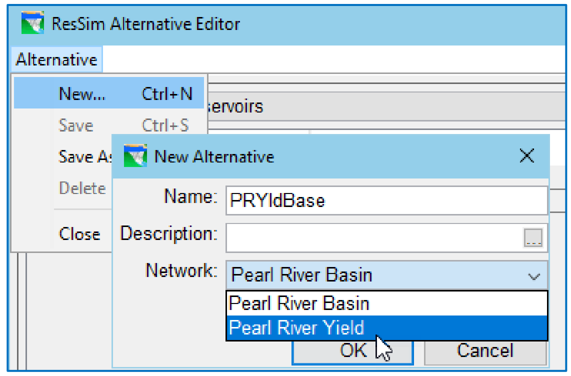


Figure F.9 Alternative Editor—New Alternative

- On the **Lookback** tab, copy the settings for each entry in the table from your template alternative to your new alternative, then **Save** the alternative before moving on to the **Time Series** tab**.** You do not need the **Observed** data in your yield alternatives, so don't copy that information. However, if you limited the output generated by ResSim using the DSS Output tab in your template alternative, consider copying those settings, too.
- When you are done copying the data from the template alternative to the new alternative, save the new alternative and close the **Alternative Editor**.
- Change to the simulation module and create a new period of record simulation with the same time window as a simulation in which your current conditions alternative was used. Include only the new alternative in the new simulation.
- Compute the new alternative. Be sure to use the **Save to Base Directory…** option if you have to make any changes to get the alternative working and computing properly.

Tip

When you need to make a copy of an alternative but Save As is not an option (*usually when the new alternative uses a* different *network than the original alternative*), use the following steps to simplify the process of copying the table data from one alternative to another:

- In the Alternative Editor, select the old alternative, then select the tab with the table you want to copy from.
- *Click* in the first cell of the table (the upper-left-most cell), then press…
  Ctrl+A—to select all the rows and columns, then press
  Ctrl+C—to copy your selection to the Windows® clipboard
- Now, select the new alternative
- Click in the first cell of the table you want to copy the data to, then press…
  Ctrl+V—to paste the data you copied into the table.

Repeat these steps for each tab and table of the alternative you want to copy.

**When using this shortcut, be sure the tables you are* copying from *and* pasting to *match row by row. When the new network is an* unchanged *copy of the original network, you can ensure matching fields by first copying the data from/to the Operations tab and the Lookback tab and then saving the alternative, before copying the data from/to the Time-Series tab.*

F.3.1.3 Add Physical Elements Needed for the Yield Analysis

- Return to the **Reservoir Network** module.
- Consider adding a diverted outlet for water supply (Figure F.10) to the yield reservoir(s). See Chapter 9 for instructions on adding a diverted outlet to a reservoir. Note that depending on the nature of your system and analysis, a diverted outlet may or may not be useful to conducting a firm yield analysis. You can alternatively apply the water supply specified release rule to the existing outlets. Considerations for whether or not to use a diverted outlet include: whether demand is taken directly from the pool or downstream, how many reservoirs have water supply demands, and how other operations interact with the water supply yield.

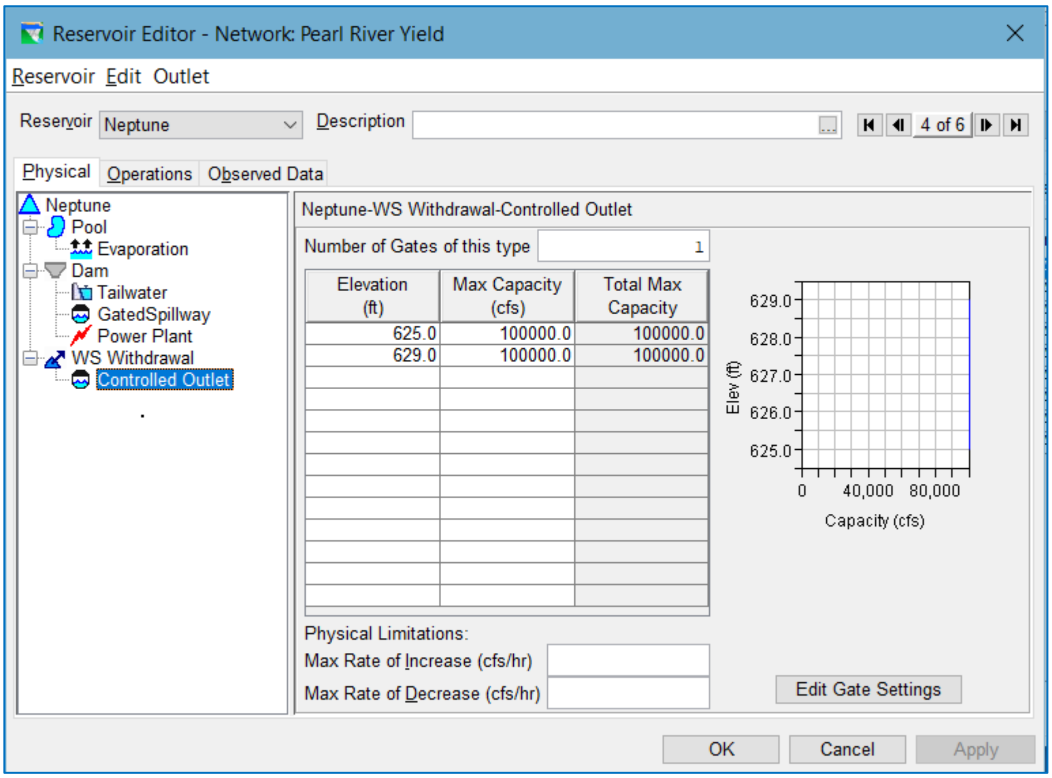


Figure F.10 Define the Diverted Outlet

F.3.1.4 Create a Base Operation Set for the Yield Analysis

- In each yield reservoir, make a base operation set for use in your yield alternative(s) by *duplicating* the operation set used to represent current conditions (Figure F.11).
- Next, remove rules in the yield operation set to reflect only the constraints that must exist at the same time as the demand. It may be appropriate to remove most, if not all, rules from the reservoirs. Note that a complete analysis may then involve adding the rules back into the operations to examine all system interactions.

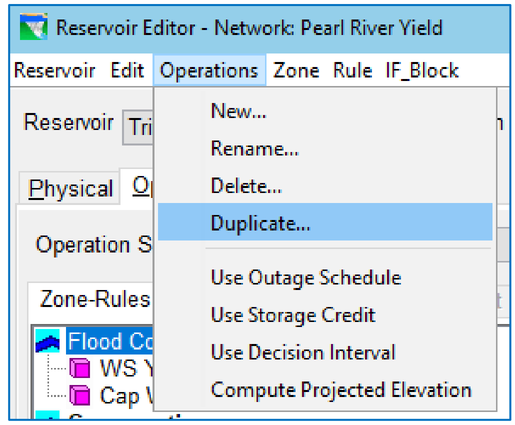


Figure F.11 Duplicate Reservoir Operation Set

- If you created any diverted outlets to represent the withdrawals that you are going to maximize, add a rule to keep the diverted outlet closed when not needed to meet demand. This can be accomplished with a low priority maximum release rule of zero flow applied to the diverted outlet (Figure F.12). When the water supply demand rule is added, it will be placed above the max of zero rule so that the diverted outlet will only release to meet demand.

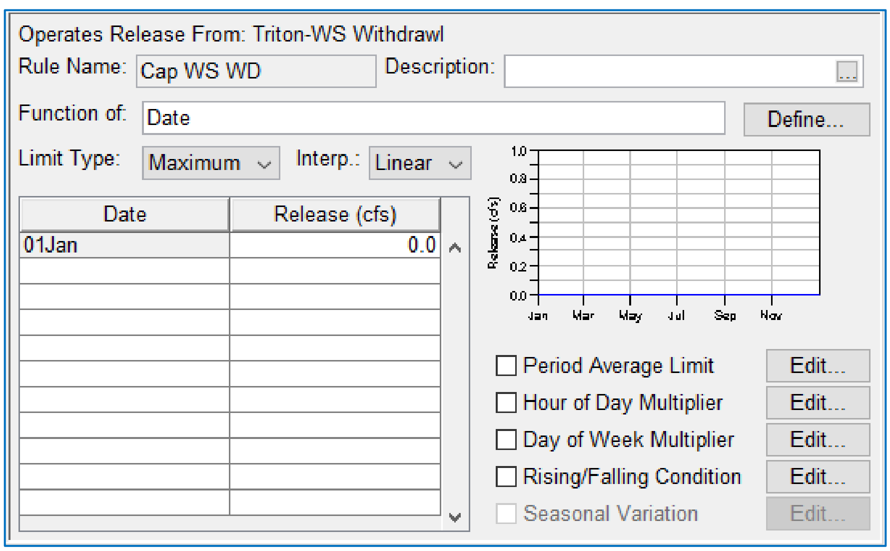


Figure F.12 Max of Zero Rule on Diverted Outlet

- Close the **Reservoir Editor** and **Save** the network.

F.3.1.5 Update and Verify the Base Alternative

- Open the **Alternative Editor** and select the base alternative
- On the **Operations** tab, select the new base yield operation sets for each yield reservoir.

- On the **Lookback** tab, set the **Lookback Release** of the diverted outlet(s) to a constant of zero flow. Set the **Lookback Storage** to the top of the guide curve; this assumes that the critical period is not during the start of the period of record. Make sure your alternative is using the correct inflows.
- **Save** the alternative and close the **Reservoir Editor**.
- Change back to the Simulation module and the period of record simulation you created.
- *Right-click* on your alternative in the **Simulation Control Panel** and select **Replace from Base** from the context menu to update the alternative with the changes you just made in the **Network** module.
- Compute the alternative and verify that it is still computing correctly.

F.3.2 Create a Yield Analysis Alternative

Once a base alternative has been created and is running correctly, a Yield Analysis alternative can be developed. The following instructions assume that you are going to perform a Firm Yield analysis on the existing conservation pool of a reservoir in your network. To do so:

- Return to the **Reservoir Network** module. Make sure your yield network is the active network.
- Open the **Reservoir Editor** and select the reservoir for which you will be computing the yield.
- **Duplicate** your base operation set. In the example shown in Figure F.13, the new operation set has been labeled *Max Yield*.
- Create a specified Release Function rule and apply it to the diverted outlet (or dam or other outlet, as necessary). This rule will represent the water supply demand to be maximized.
- Give the new demand rule a starting value or seasonal pattern. This value will be replaced during each iteration of the Yield Analysis run.

Add the demand rule to the zone(s) that make up the conservation pool. Also, add this rule to the adjacent zones above and below the conservation pool. If the lower adjacent zone is the **Inactive** zone, then the rule only needs to be added to the zone immediately above the guide curve (typically the Flood Control zone). In the example shown in Figure F.13, the new rule is called "WS Yield" and has a starting value of 50 cfs.

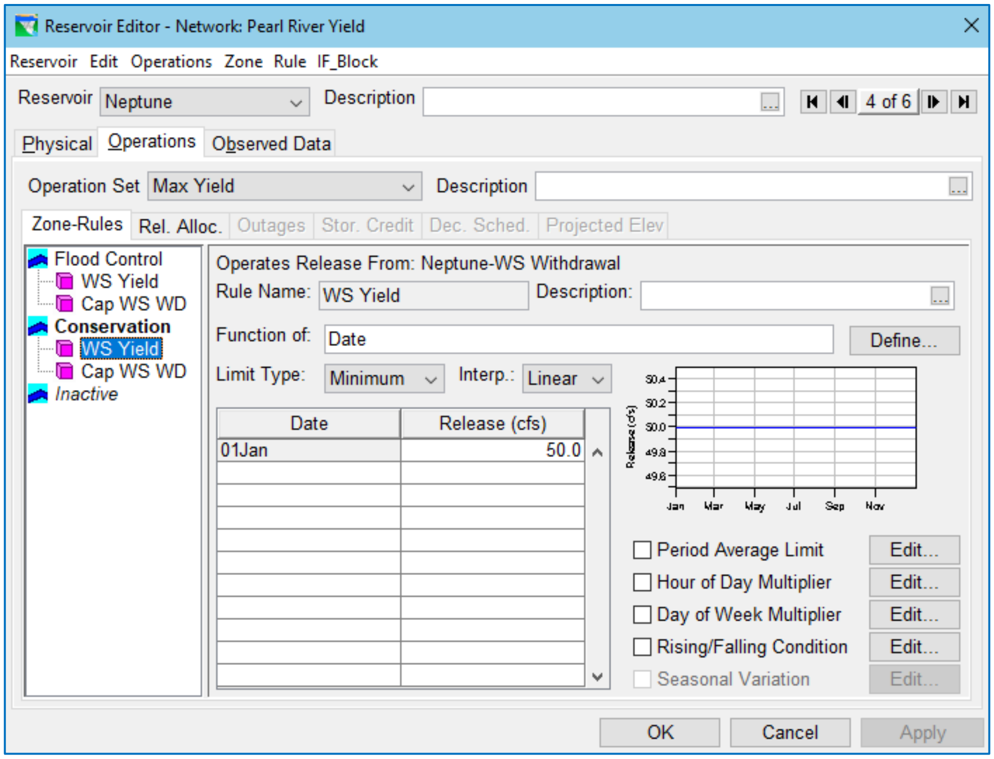


Figure F.13 Reservoir Editor—New Operation Set and New Yield rule

- In the **Alternative Editor**, use Save As… to make a copy of the base alternative (Figure F.14), then perform the following changes to create a yield alternative.

- On the **Run Control** tab, change the **Alternative Type** to **Yield Analysis** (Figure F.15). This will activate the **Yield Analysis** tab of the Alternative Editor.

- On the **Operations** tab, select the yield operation set you just created with your new yield rule (Figure F.16). Save your alternative.

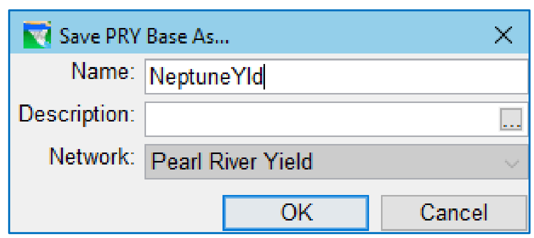


Figure F.14 Alternative Editor—Save As…

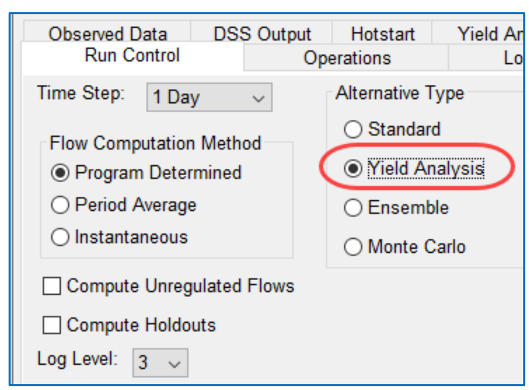


Figure F.15 Set Alternative Type to Yield Analysis

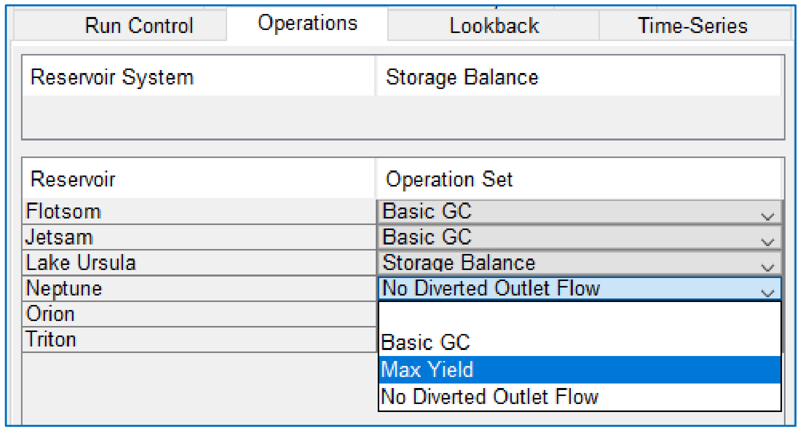


Figure F.16 Alternative Editor—Operations Tab—
Select the Yield Operation Set(s)

- On the **Yield Analysis** tab (Figure F.17), set the **Yield Analysis Type** to **Reservoir Yield**.
- Use the radio buttons to select the **Type of Demand Rule** you are going to Maximize. Choose **Water Supply Rules**.
- Select the Rule. The filters are provided to help you find the rule quickly. The list of available rules will only be long if the selected operation sets for your reservoirs are complex. To select the rule, you must highlight it and press the **Select** button.
- Your selected rule will appear in the **Selected Demand Rule** table. Enter a flow **Tolerance** value for the rule. The flow tolerance is the maximum amount that the demand can be shorted and still pass the flow convergence test.
- Storage information about the selected rule and its reservoir will appear in the storage table. Select the top-of-zone curve that marks the bottom of the pool from which the withdrawal defining the firm yield may be made.
- Specify a storage **Tolerance**. The storage tolerance is the maximum amount of storage that may remain the reservoir conservation pool and still pass the storage convergence storage test.
- Next, enter a **Maximum** number of **Iterations**. The default value is 25. Very few models require 25 iterations. If the compute has not converged within 25 iterations, you may have a problem with your model; review the Storage Yield report to determine if the iterations are oscillating away from solution or if one or both of tolerances are too small or too large.
- The last parameter to set is the **Convergence Method**. Your options are **Bisection Search Only** and **Heuristic and Bisection Search**. The latter option allows you to specify how many Heuristic Iterations should be performed before automatically switching to Bisection Search Iterations.
- **Save** your alternative.

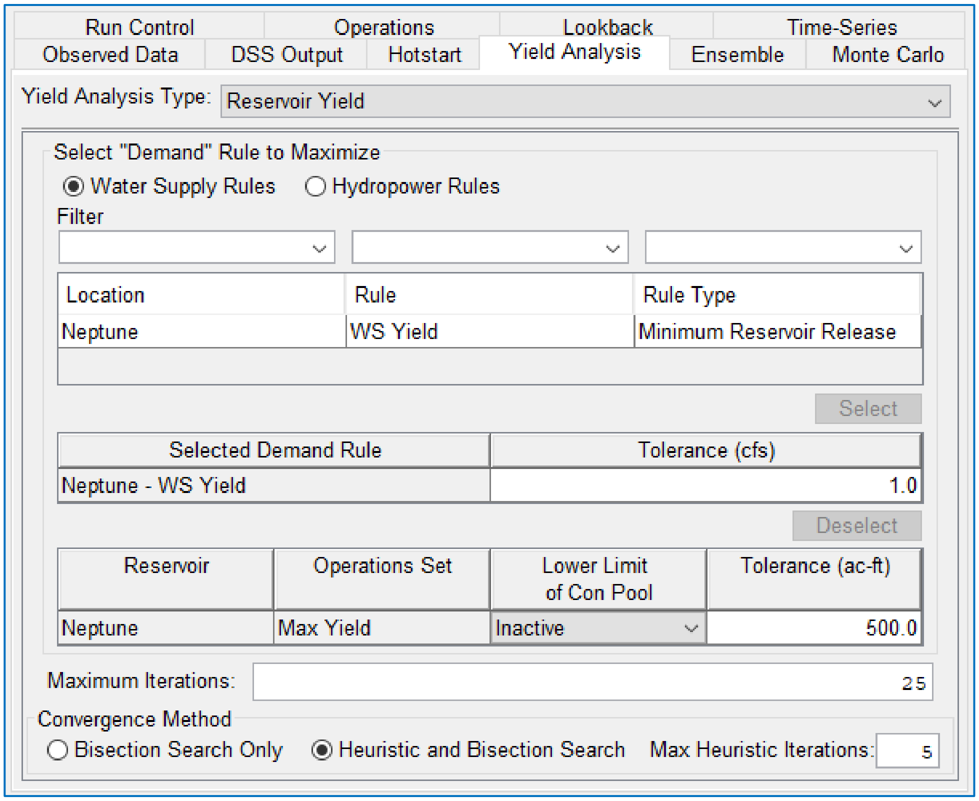


Figure F.17 Alternative Editor—Yield Analysis—Select the Rule and Set the Tolerances

F.3.3 Compute a Yield Alternative

Now that a Firm Yield alternative has been created, you can run it for the period of record.

- In the Simulation Module, add your Firm Yield alternative to the period of record simulation that currently holds your base alternative.
- Make your Firm Yield alternative the active alternative and compute it.

  As each iteration completes, it reports how well it did with respect to the tolerances in the Compute Window and in the Compute Log. You can review these reported results while it runs or after it computes.

  After the last iteration's status output, the final yield value will be reported. The final value for the maximized yield rule will also be stored in the demand rule itself. Figure F.18 shows the calculated yield value in the Compute Window.

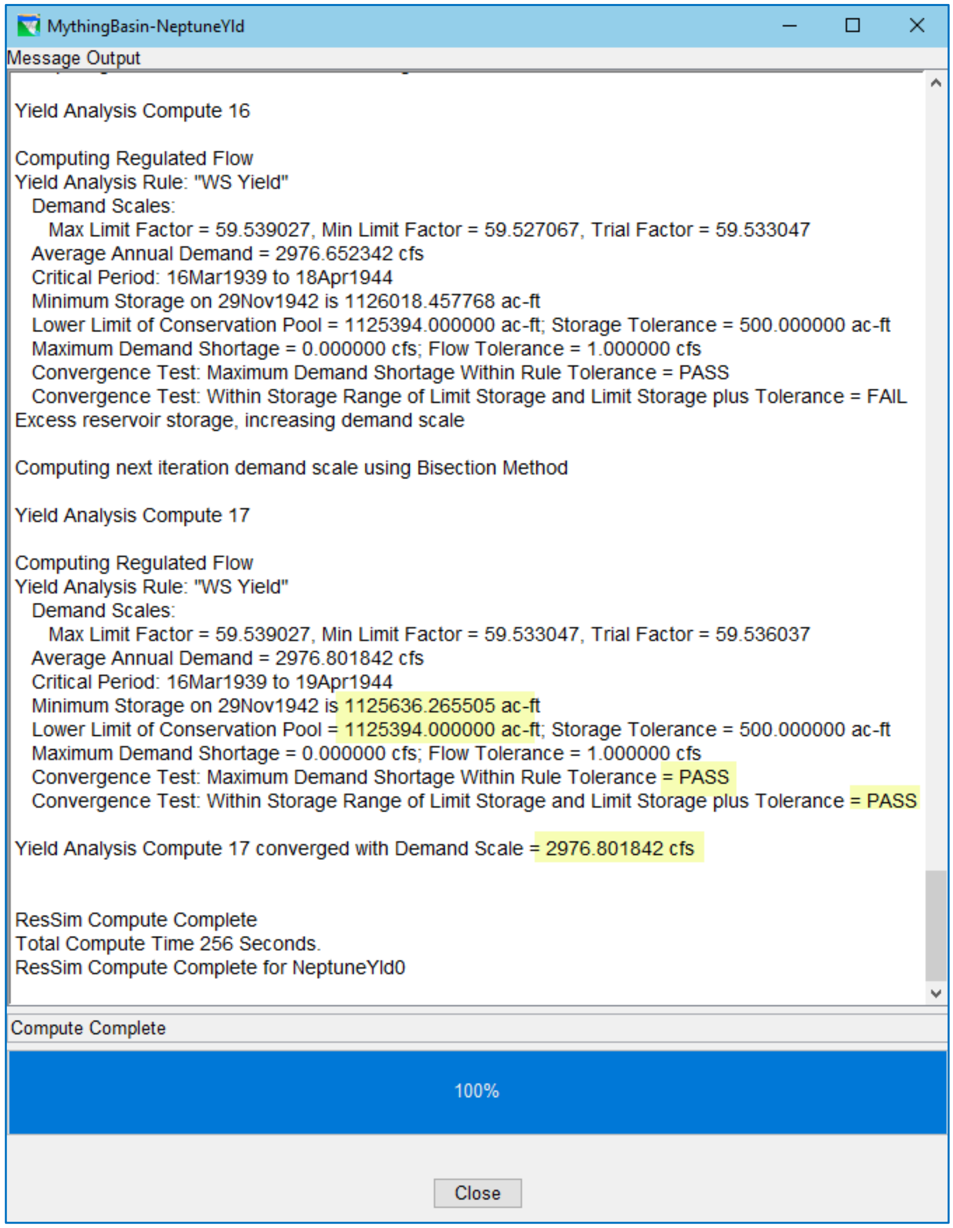


Figure F.18 Compute Window—Final Yield Value

- Open the **Reservoir Editor** to verify that the yield rule has been updated. Figure F.19 shows that the simulation's copy of the yield alternative has the "WS Yield" rule populated with the same value.

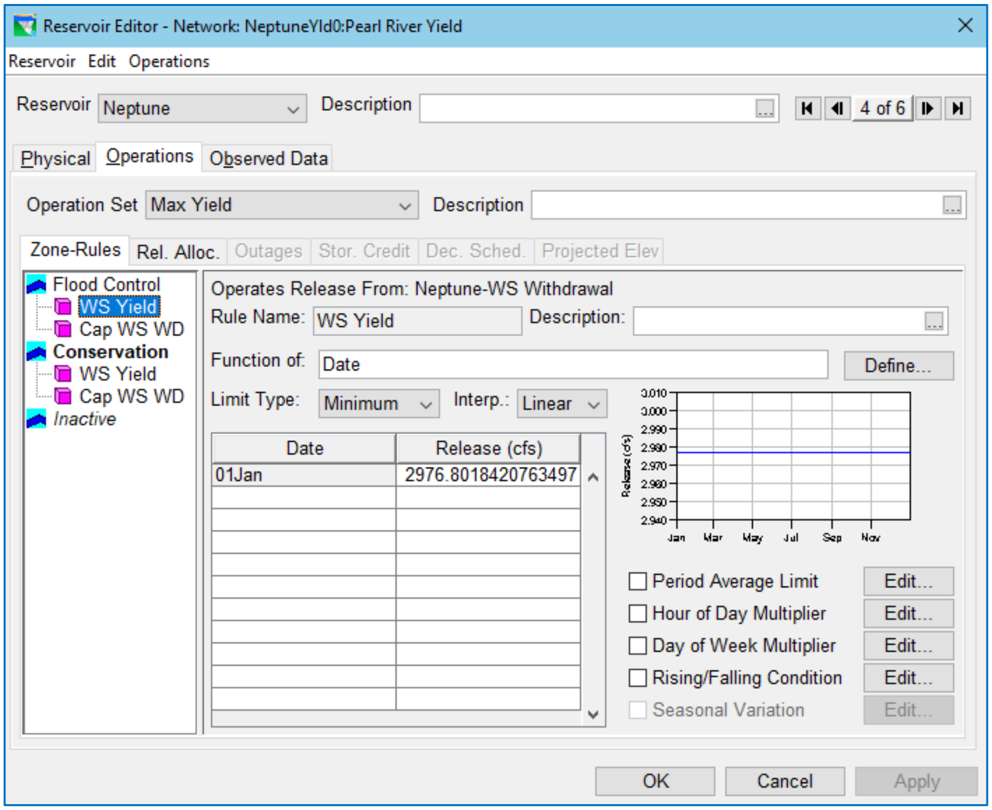


Figure F.19 Reservoir Editor—Yield Rule—After Last Yield Iteration

F.3.4 Analyze a Yield Alternative

Once ResSim has completed the Firm Yield simulation and reported a yield value in the demand rule, you may examine the results using the reservoir plot and the yield report.

The default reservoir plot (Figure F.20) can be used to visualize the critical period and the point at which the reservoir was drained to the bottom of the conservation pool. For this example, adding the "release" variable to the plot allows for the viewing of the controlled outlet release, which represents the demand. The yield value is constantly released over the period of record.

The Yield Analysis Summary report (Figure F.7) shows the information used in each iteration to achieve convergence. You can see the demand value used in each iteration and how close it was to the required storage and flow tolerances. The report also identifies the critical period.

Note: This is a cursory example of a firm yield analysis, provided only to guide you through some of the basics for setting up a one type of yield alternative. Setting up other types of yield analysis (such as hydropower and water account) use similar steps. A full summary of the available output and the process for fully examining a yield simulation is not described in this manual.

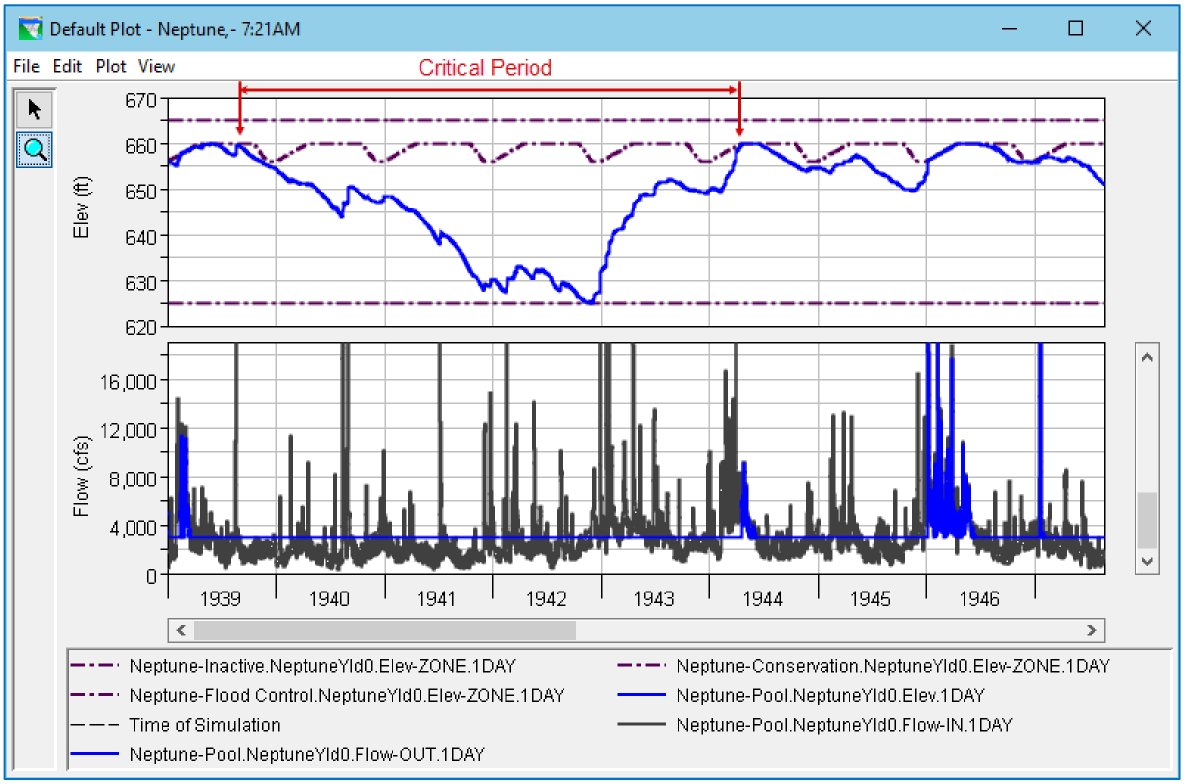


Figure F.20 Default Reservoir Plot with Releases

Appendix G
Ensemble Computing

Ensemble Computing was added to HEC-ResSim to support reservoir operations modeling using ensemble streamflow forecasts produced by NOAA's Extended Streamflow Prediction (ESP) of the National Weather Service River Forecast System (NWSRFS) and/or its successor, the Community Hydrologic Prediction System (CHPS). The ensemble computing capability can, however, be used with data that comes from other sources.

An ensemble streamflow forecast for a single location represents a set of possible streamflow hydrographs that could occur at that location over the forecast period. A full ensemble forecast for a watershed should be a correlated set of ensemble inflow forecasts for all inflow locations in the basin. This means that the first hydrograph in the ensemble forecast for a headwater location in the basin was generated with the same set of model parameters and boundary conditions as for the first hydrograph at the downstream-most location (and all points in between).

A *standard* ResSim alternative can be defined to simulate only one member (or trace) of an ensemble forecast set. Thus, if an ensemble forecast contained 30 members, then you would need to define 30 standard ResSim alternatives to simulate a given operation, one for each member of the ensemble forecast. This is very time consuming to setup and cumbersome to manage and analyze. However, with a new **Ensemble Alternative**, you only need to create one alternative to simulate through one or more members in an ensemble forecast. And, ensemble results from an Ensemble Alternative can be visualized together. Section G.3 of this Appendix will describe how to define and use **Ensemble Alternatives** in ResSim.

But, before you can jump into creating and running **Ensemble Alternatives**, you will need to "collect" your ensemble hydrographs into a new HEC-DSS data representation called a **Collection**. Collections are defined using an F-part naming convention that identifies each member of the collection with a numeric string, followed by a separator after which the normal F-part conventions apply for identifying the dataset. The collection member number (or ID) are how you can reference specific ensemble members to be computed in the **Alternative Editor**. Section G.1 below describes HEC-DSS **Collections** and how to create and edit them for use in ResSim **Ensemble Alternatives**.

G.1 HEC-DSS Collections

A **Collection** is a group or set of HEC-DSS time-series datasets that share a common location (A & B Parts), parameter (C Part), and time-step (E-Part). *Any* set of DSS time-series records for a given location, parameter, and time-step can be put into a collection, however, when applied to an *ensemble*, the records grouped into a collection must span a common time window.

The "collection" concept is implemented through an *F-part naming convention*. The F-part of each member of a collection has a collection ID string prepended to a common version label (F Part). The collection ID string is made up of: "C:" followed by a 6-digit collection member number (ID) and ending with a pipe symbol "|". The | is used as a delimiter; it separates the collection ID string from the standard F-part string that specifies the "version" or uniqueness of the data. The ID number must be 6 digits long and may be simple integers or alpha-numeric strings, but must be unique within the collection. Like the other pathname parts, the "version" string following the | will be the same for all members in the collection.

Below is an example pathname of a collection member:
/YUBA/SMARTSVILLE/FLOW/01JAN1997-30SEP1997/1HOUR/C:000042|ESP/

G.2 The DSSVue Collection Utilities

The software for creating and managing DSS collections is provided in a DSSVue Collection Utilities Plug-In. This plug-in may be used in HEC-DSSVue Version is 2.2.1 or later. The plugin is included with HEC-ResSim 3.3 (and later), along with ResSim's Ensemble plug-in, so you do not need to do anything to activate either plug-in in ResSim. However, if you would like to use the Collection Utilities in a standalone version of DSSVue, the instructions for installing the plug-in are below.

To install the *Collection Utilities Plug-In* to a standalone version of DSSVue:

- Make sure that DSSVue is not currently running.
- Find the installation folders of this version of ResSim and the version of DSSVue you want to add the plug-in to.
- Copy the file: CollectionUtilities.jar
  from: ResSim's *jar/ext* folder
  to: DSSVue's *jar/ext* folder
- When installed correctly, you'll see a new **Collections** menu added to DSSVue's menu bar the next time you launch DSSVue.

With the *Collection Utilities Plug-In* installed, a **Collections** menu is added to the menu bar of DSSVue providing access to several options for creating and managing collections. Many of these options represent different ways to create a collection. At present, collections can only be formed from individual records that already exist in DSS. Other options in the **Collections** menu include features to copy, re-number, rename, and perform some simple statistics on the collection members.

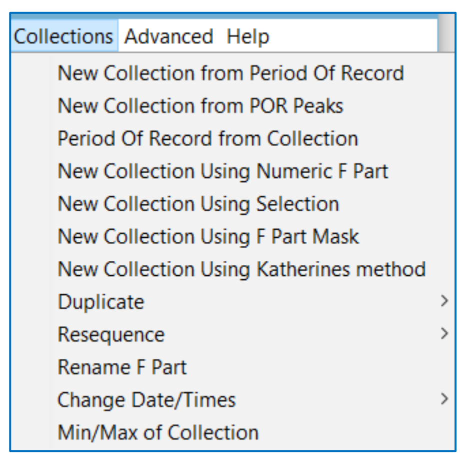


Figure G.1 Collections Menu Options

The Collections Utilities Plug-In also adds a new Condensed—Group Collections entry to the **View** menu in DSSVue. A description of this view option as well as descriptions of most of the **Collection** menu options are provided in the following sections.

G.2.1 Create a Collection from a Period of Record Dataset

Collections can be created within DSSVue in a number of ways, but you must start with existing DSS records. Two of the create-collection utilities start with a period of record (POR) data set; one searches for data based on a date range, for example December 15 through January 25; the other uses annual peaks.

G.2.1.1 New Collection from Period of Record (Date Range)

To create a collection from a period of record dataset using a date range:

- From the DSSVue **View** menu, select **Condensed Catalog**.
- Select a period of record dataset. A period of record dataset is usually one that spans several years.
- From the **Collections** menu, select **New Collection from Period of Record**.
- You will be prompted to enter the **starting** (Figure G.2) and **ending** (Figure G.3) **date** and **time** of the data that will l be copied from the period of record for each member of the collection.

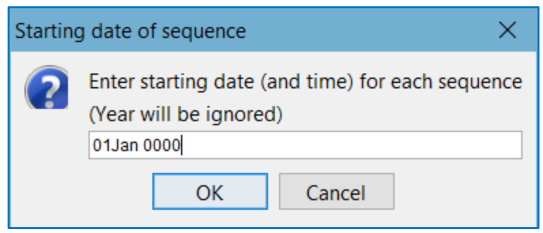


Figure G.2 Prompt for Starting Date and Time

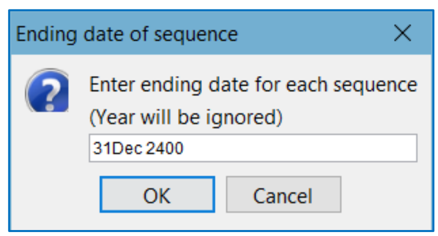


Figure G.3 Prompt for Ending Date and Time

  - ↳ Dates should be entered in the form DDMMM or DDMMMYYYY. And times should entered in hours and minutes of a 24-hour clock: hhmm or hh:mm. For values less than 10 for days (DD), hours (hh), or minutes (mm), leading zeros *are* required.
  - ↳ As indicated in the prompt, you do not need to enter a year, since the date range is year-independent. *However*, if the time window of data you want to copy for each member is longer than 1 year, the year must be entered and will *not* be ignored. For example. If you want 15 months of data, spanning October to December, copied from the POR dataset into each collection member, then if you entered 01Oct 0000 and 31Dec 2400, you would get only 3 months of data; the correct entries are 01Oct1901 0000 and 31Dec1902 2400.
  - ↳ If the timestep (interval) of the data is less than 1DAY, you must enter start & end times with the start & end dates.
  - ↳ If your date range extends beyond a single year, include the year *and* time in your start & end date/time specification.

- Next, you will be prompted (Figure G.4) to enter a **starting sequence number** (first member ID number). You can leave it blank if you want to use the data's original year as the sequence number (most common).

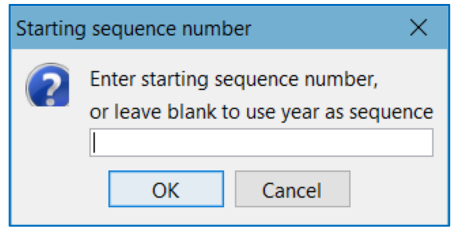


Figure G.4 Prompt for Starting Sequence Number

- Lastly, you will be prompted to enter the date and time for the first data point of the collection (Figure G.5).
- The prompt will provide a suggested date and time based on the start date and time you entered; it will include the year 3000, which is often used for generated data that is intended to be year independent.

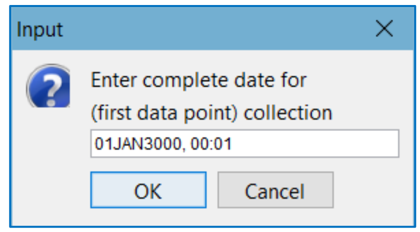


Figure G.5 Prompt for Date and Time of the First Value in each Collection Time Series

- ***Review the suggested data and time carefully***. The time suggested is sometimes a poor choice—for example, it may suggest 0001 as a start time for 1DAY data when a more appropriate value would be 2400. In addition, you may want to use a different base year for your collection
- *Click* **OK** to create the collection.

The plug-in will process its way through the time window of the selected POR dataset and copy (from each year) the data within the date range you specified into a new dataset. Each new dataset will be assigned a collection member ID number starting with the sequence number you provided or with the year the new dataset was copied from. And, the time of the first value of each new dataset will be the collection start date and time you entered.

G.2.1.2 New Collection from POR Peaks (Annual Peaks)

To create a collection from a period of record dataset using a time window around the annual peak value in the dataset:

- Select one or more period of record datasets.
- From the **Collections** menu, select **New Collection from POR Peaks**.
- You will be prompted to enter the number of days **before** and **after** (Figure G.6) the **peak** in order to define the date range.
- Then, just like in the previous section (POR date range), you will be prompted to enter a **starting sequence number**

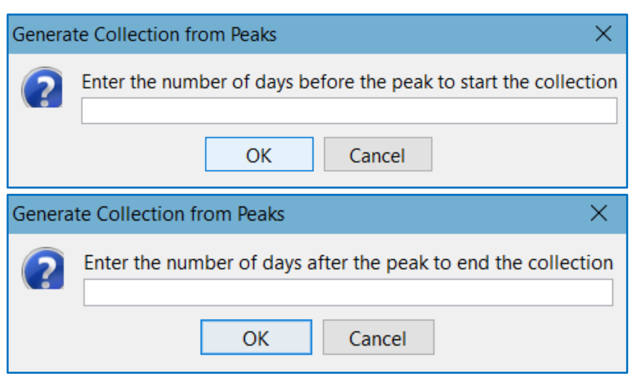


Figure G.6 Prompts for Days Before and After the Peak and a **date** for the **first data point** of each member of the collection (Figure G.4 and Figure G.5 above).

- *Click* **OK** to create the collection.

The plug-in will work its way through the time window of the selected POR dataset, find the peak in each year, and copy (from each year) the data within the time window you specified around the peak into a new dataset. Each new dataset will be assigned a collection member ID number starting with the sequence number you provided or with the year the new dataset was copied from. And, the time of the first value of each new dataset will be the collection start date and time you entered.

G.2.2 Create a New Collection Using Numeric F Part

If a number in the F-part uniquely identifies each dataset in a set of selected datasets, that number can be used as the collection member ID when assembling the selected dataset into a collection. This option will find the numeric value in the F-part of each dataset and use that value as the collection member ID (sequence number).

To create a collection from a set of selected records using a numeric value in the F-part of the pathnames:

- Select the datasets that you want to group into a collection.
- From the **Collections** menu, select **New Collection Using Numeric F Part**.

This utility function will rename the selected records to form a collection. The F part of each resulting collection member will only contain the collection ID number that was found in the original F part of its source dataset, no other information from the original F part will be retained. Figure G.7 and Figure G.8 show the before and after pathnames for a set of selected datasets that were used to form a collection using the **New Collection Using Numeric F Part** function.

	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	RUN:RUSSELL TOTAL - 10 YEAR
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	RUN:RUSSELL TOTAL - 100 YEAR
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	RUN:RUSSELL TOTAL - 2 YEAR
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	RUN:RUSSELL TOTAL - 200 YEAR
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	RUN:RUSSELL TOTAL - 25 YEAR
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	RUN:RUSSELL TOTAL - 5 YEAR
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	RUN:RUSSELL TOTAL - 50 YEAR
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	RUN:RUSSELL TOTAL - 500 YEAR

Figure G.7 Original Pathnames

	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	C:000002
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	C:000005
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	C:000010
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	C:000025
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	C:000050
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	C:000100
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	C:000200
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	C:000500

Figure G.8 Collection Pathnames

G.2.3 Create a New Collection Using F Part Mask

Data may be gathered from a variety of sources and numbers are often used to identify the uniqueness of the data. For example, if you use a rainfall-runoff model to compute flows for a set of frequency events with a variety of storm centerings, you are likely to use a string identifying both the storm centering and the recurrence interval as a way to identify the output. Figure G.9 shows what these pathnames might look like.

With a bit of adjustment, the datasets shown in Figure G.9 can be grouped into collections labeled with the storm centering and using the recurrence interval as the member ID. The reason for the adjustment is because the utility that will create the collection uses a string *mask* to locate the member ID number and the remaining collection F-part. This mask relies on the numeric portion of the F-part to either always have the same number of digits or for the digits to be at the end of the F part.

Part A	Part B	Part C	Part D / range	Part E	Part F
	BRIER CR KEYSVL-LOC	FLOW	01Feb2099 - 09Feb2099	1HOUR	RUN:THURMOND TOTAL - 5YR
	BRIER CR KEYSVL-LOC	FLOW	01Feb2099 - 09Feb2099	1HOUR	RUN:THURMOND TOTAL - 50YR
	BRIER CR KEYSVL-LOC	FLOW	01Feb2099 - 09Feb2099	1HOUR	RUN:THURMOND TOTAL - 500YR
	BRIER CR KEYSVL-LOC	FLOW	01Feb2099 - 09Feb2099	1HOUR	RUN:THURMOND TOTAL - 2YR
	BRIER CR KEYSVL-LOC	FLOW	01Feb2099 - 09Feb2099	1HOUR	RUN:THURMOND TOTAL - 25YR
	BRIER CR KEYSVL-LOC	FLOW	01Feb2099 - 09Feb2099	1HOUR	RUN:THURMOND TOTAL - 200YR
	BRIER CR KEYSVL-LOC	FLOW	01Feb2099 - 09Feb2099	1HOUR	RUN:THURMOND TOTAL - 10YR
	BRIER CR KEYSVL-LOC	FLOW	01Feb2099 - 09Feb2099	1HOUR	RUN:THURMOND TOTAL - 100YR
	BRIER CR KEYSVL-LOC	FLOW	01Feb2099 - 09Feb2099	1HOUR	RUN:THURMOND BLW RUSSELL - 5YR
	BRIER CR KEYSVL-LOC	FLOW	01Feb2099 - 09Feb2099	1HOUR	RUN:THURMOND BLW RUSSELL - 50YR
	BRIER CR KEYSVL-LOC	FLOW	01Feb2099 - 09Feb2099	1HOUR	RUN:THURMOND BLW RUSSELL - 500YR
	BRIER CR KEYSVL-LOC	FLOW	01Feb2099 - 09Feb2099	1HOUR	RUN:THURMOND BLW RUSSELL - 2YR
	BRIER CR KEYSVL-LOC	FLOW	01Feb2099 - 09Feb2099	1HOUR	RUN:THURMOND BLW RUSSELL - 25YR
	BRIER CR KEYSVL-LOC	FLOW	01Feb2099 - 09Feb2099	1HOUR	RUN:THURMOND BLW RUSSELL - 200YR
	BRIER CR KEYSVL-LOC	FLOW	01Feb2099 - 09Feb2099	1HOUR	RUN:THURMOND BLW RUSSELL - 10YR
	BRIER CR KEYSVL-LOC	FLOW	01Feb2099 - 09Feb2099	1HOUR	RUN:THURMOND BLW RUSSELL - 100YR

Figure G.9 Example Data with Numeric F-parts

Figure G.9 shows some datasets whose pathnames reflect two storm centerings and eight recurrence intervals. To use these pathnames with an F-part mask, either the recurrence interval strings must all use 3 digits or the "YR" characters at the end must be removed. If you plan ahead when creating the data, you can use the three digits. If not, as in this example, the easiest way to make the adjustment was to rename the records and remove the *YR*s. The revised pathnames are shown in Figure G.10.

Part A	Part B	Part C	Part D / range	Part E	Part F
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	RUN:THURMOND TOTAL - 500
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	RUN:THURMOND TOTAL - 50
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	RUN:THURMOND TOTAL - 5
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	RUN:THURMOND TOTAL - 25
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	RUN:THURMOND TOTAL - 200
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	RUN:THURMOND TOTAL - 2
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	RUN:THURMOND TOTAL - 100
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	RUN:THURMOND TOTAL - 10

Figure G.10 Revised Pathnames Ending in Numeric F-parts

With the pathnames ready, here are the steps to create a collection using an F-part mask:

- Select the datasets you want to put into a collection
- Select **New Collection Using F Part Mask** from the **Collections** menu
- You will be prompted to "Enter Mask over existing F Part". This means you should type over the original first F-part with a set of characters to build the mask. Figure G.11 shows the prompt dialog with the original F-part and below it the mask we used for he selected datasets shown in Figure G.10.
- After entering the F-part mask, click **OK**.

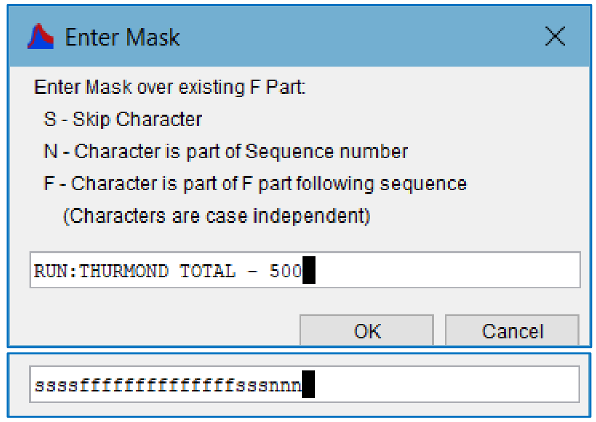


Figure G.11 Prompt for F-part Mask—Before and After

The **New Collection Using F Part Mask** function will parse the F-part of the first selected dataset to determine the F part for all members of the new collection. Then, as it copies each selected data set into the collection, this function will parse the F Part of each dataset to identify the collection member's ID number. Figure G.12 shows the new collection pathnames created for the example datasets shown in Figure G.10 using the F-part mask shown in Figure G.11.

	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	C:000500|THURMOND TOTAL
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	C:000200|THURMOND TOTAL
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	C:000100|THURMOND TOTAL
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	C:000050|THURMOND TOTAL
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	C:000025|THURMOND TOTAL
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	C:000010|THURMOND TOTAL
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	C:000005|THURMOND TOTAL
	BRIER CR KEYSVL-LOC	FLOW	01FEB2099	1HOUR	C:000002|THURMOND TOTAL

Figure G.12 New Collection Pathnames Using F-Part Mask

G.2.4 Assemble a Period of Record from a Collection

You can create a period of record dataset from a collection by appending each member to one another, in sequence. This function assumes that the members of your collection represents a single and complete year. This is useful for doing a statistical analysis, such as the Cyclic Analysis Function. This function also assumes you want the resulting dataset to have the same pathname parts as the collection; so, if you already have a dataset with that pathname, rename it before running this command or it will be overwritten.

- From the DSSVue **View** menu, select **Condensed-Group Collections**.
- Select the collection from which you want to create a period of record data
- From the **Collections** menu, select **Period of Record from Collection**.
- If the collection IDs do not represent years, specify a starting year for the new dataset.

Another assumption used by the **Period of Record from Collection** function is that each member of the collection is placed in the new POR dataset at the same time of year as the start of the collection member itself. So, if all the members of the collection only span the 6 months from 01Apr to 30Sep, then they will be placed at 01April in their designated year in the POR dataset and the other 6 months of will be "missing" in each year of the POR. But, if all the members of the collection span 15 months, when the data of each member is placed at the same start date of each year, 3 months of each collection member will be overwritten by the next member when it is written to the POR dataset.

G.2.5 Duplicating Collections

You can duplicate a collection with new pathname F-parts or sequence numbers. For changing pathname parts A, B, and C, use the standard HEC-DSSVue **Duplicate** function under the **Edit** menu.

To change the F-part, with either a new sequence number or F-part past the sequence:

- Select **Condensed-Group Collections** from the **View** menu
- Select your collection,
- Select **Duplicate** from the **Collections** menu, then select how you want to identify the copy:

  - **Using New Sequence Number**—*Don't use this option, it does not currently work as intended.*
  - **Adding to Sequence Number**—enter a value to add to each collection ID number. Each member of the collection will be duplicated and the copy will get an ID number equal to the original ID value plus the value you entered. Note: this does not actually create a new collection; instead, it doubles the size of the selected collection.
  - **New F-part** (same Sequence)—enter a new F part string. Each member of the selected collection will be copied to a new collection with the same ID number but with a new F-part string following the collection ID. Note: If you want to change both the sequence (ID) number and the F-part, duplicate with a new F-part first, then re- sequence the collection.

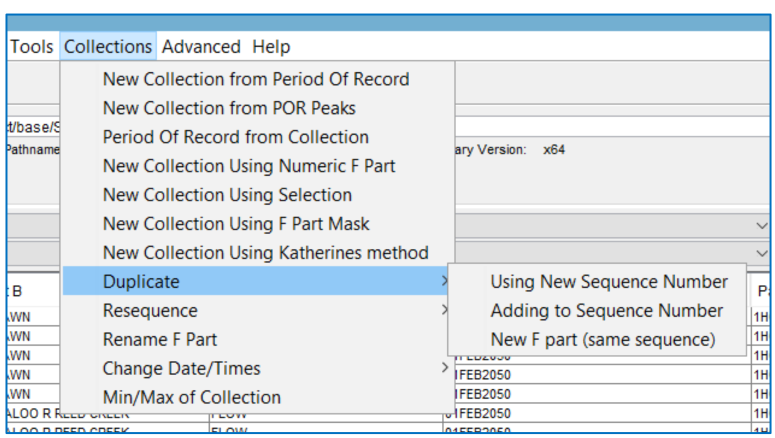


Figure G.13 Collection Utilities—Duplicate Collections

G.2.6 Re-sequencing a Collection

You can re-sequence (re-number) a collection by providing a new starting sequence number or adding to the existing sequence number. Re-sequencing does not duplicate the collection, it renames each member in the collection. To re-sequence:

- Select **Condensed-Group Collections** from the **View** menu
- Select your collection
- Select **Re-sequence** from the **Collections** menu and then select how you want to define the new sequence:
  - **Using New Sequence Number**—enter a new starting number for the collection. Each member will be assigned a new ID number starting at the new sequence number and counting up by 1.
  - **Adding to Sequence Number—**enter a value to add to each collection ID number. Each member of the collection will get a new ID number equal to the original ID value plus the value you entered.

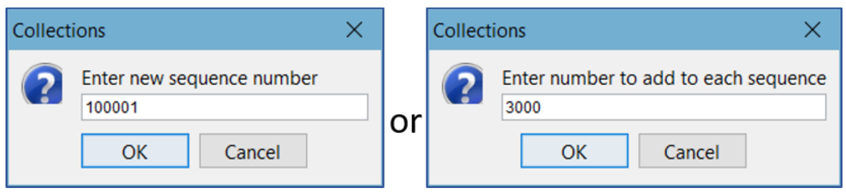


Figure G.14 Resequence Options

G.2.7 Renaming the F-part of a Collection

You can rename the F-part of a collection (the portion of the F part that follows the Collection ID). To do so:

- Select **Condensed-Group Collections** from the **View** menu
- Select your collection
- Select **Rename F-part** from the **Collections** menu. (To rename other parts, use the standard **Rename** function from the DSSVue **Edit** menu).
- Enter the new F-part string (Figure G.15). Each member of the selected collection will be renamed with its original Collection ID number but with the new F-part string following the Collection ID.

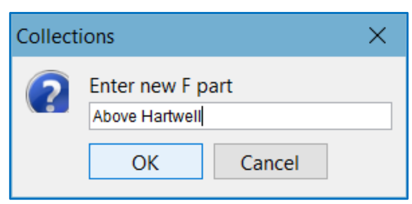


Figure G.15 Collections Rename F-part

G.2.8 Changing the Date and Time of a Collection

You can change the dates and times of the data in a collection. For example, if you had an ensemble that started Jan 1, 3000, you could copy it in time to the current date and use it in your forecast or other real-time analysis. The Change Date/Times function gives you the option to either copy your collection to a new date/time or shift (move) it to a new date/time. When you *copy* a collection in time, a new collection is not created; instead, your current collection is simply enlarged (in time) so that you have the original data at the original dates and times *and* that same data copied to the new time period. When you *shift* a collection, the data will be moved to the new dates and times and will *no longer exist* at the previous dates and times.

To copy or shift a collection in time:

- Select **Condensed-Group Collections** from the **View** menu.
- Select your collection.
- Select **Change Date/Times** from the **Collections** menu, then select either **Copy** or **Shift**.
- Enter the complete date and time to copy or shift the collection to. Use the same format as shown in the example (Figure G.16).

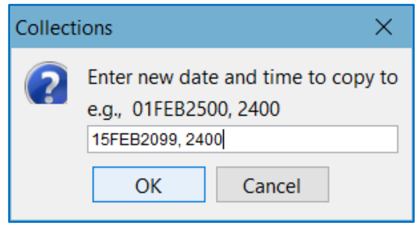


Figure G.16 Change Date/Time Input

G.2.9 Pathname List—Condensed Collection View

With the Collections plugin, if your current DSS file has one or more collections in it, a new option will appear in the DSSVue **View** menu for displaying the Pathname List. The new **View** menu option is **Condensed** - **Group Collections**. If you select this option, the many pathnames representing the members of your collection(s) will condense down to a single pathname. Like the standard condensed catalog which condenses the dates in the D-part into a range, the condensed collection pathname shows a range of the Collection IDs in the F-part as highlighted in Figure G.17.

Number	Part A	Part B	Part C	Part D / range	Part E	Part F
4	ALABAMA	R.F.HENRY	INFLOW	01JAN1938 - 01JAN2012	1DAY	COE_CORR
5	ALABAMA	R.F.HENRY	INFLOW	01JAN1938 - 01JAN2012	1DAY	OBS_ADJ
6	ALABAMA	R.F.HENRY	INFLOW	01Jan3000 - 01Jan3001	1DAY	C:001938 - 002011|OBS_ADJ
7	COOSA	H.N. HENRY	INFLOW	01JAN1938 - 01JAN2012	1DAY	APC_CORR

Figure G.17 Pathname List—Condensed - Group Collections View

If you select a collection dataset (condensed collection pathname) and *click* the plot button, DSSVue will produce a plot window that appears to be all legend (Figure G.18). This is because a collection is usually a large group of individual datasets, each of which must be included in the legend. See the next section for more information.

G.2.10 Plotting a Collection

Plotting a collection is much the same as plotting any other DSS data. However, because a collection represents several datasets, you may have some problems getting a useful plot. Here's a suggestion—to plot a collection:

- Select **Condensed-Group Collection**s from the **View** menu.
- Select your collection.
- Click the **Plot** button on the Toolbar. If your collection has 20 or more members, you plot window may open looking something like Figure G.18.

- To see the curves of your plot, you might try resizing the window, but for large collections, that won't be good enough because the legend is still going to take up the whole plot window. Instead, try removing the legend from the plot window:
    - Select **Hide Legend** from the **View** menu of the plot window, or
    - Select **Legend Placement -> Separate Window** from the **View** menu. The resulting plot from either of these options should look something like the colorful "spaghetti" plot shown in Figure G.19.

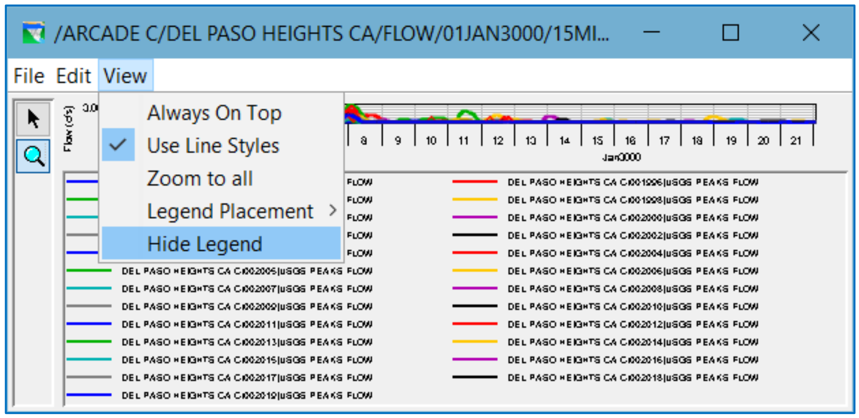


Figure G.18 Collection Plot—All Legend

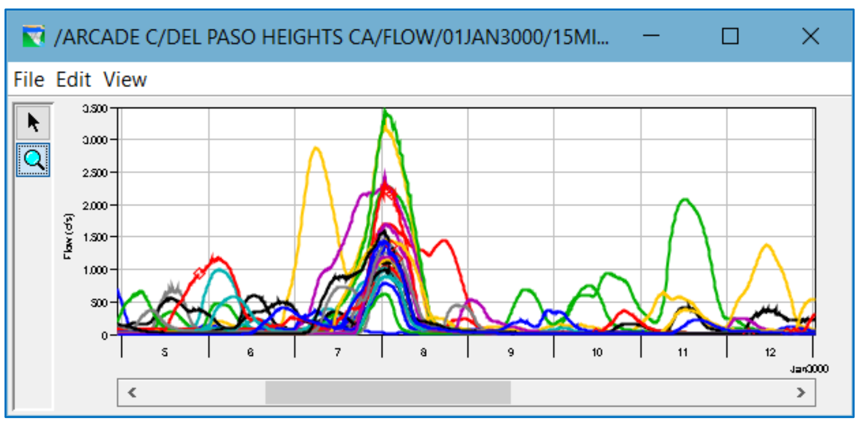


Figure G.19 Collection from POR Peaks—"Spaghetti" Plot, No Legend

G.3 ResSim Ensemble Alternatives

An **Ensemble Alternative** is a new type of alternative available in ResSim. As described in the introduction of this Appendix, it was created to facilitate reservoir operations modeling using ensemble forecasted inflows. So, while a standard ResSim alternative represents only one correlated set of inflow hydrographs for the basin and produces one set of outflow hydrographs, an ensemble alternative represents multiple correlated sets of inflow hydrographs and produces the associated multiple correlated outflow hydrographs.

Ensemble inflows and the associated ensemble results utilize a new HEC-DSS time-series data group called a "collection". Collections are described in Section G.1 of this Appendix. Collections can contain time series dataset that have varying time windows. However, collections that represent ensemble forecasts must contain only time series datasets with the same time window (or that span a common time window).

An ensemble alternative is essentially the same as a standard alternative except that instead of representing a single *run* and computing through a single set of inflow time-series as a standard alternative does, an ensemble alternative represents several *runs*, one for each *selected* member of the ensemble inflow datasets. ensemble set of inflow time-series. To do that, an ensemble alternative must use at least one collection dataset in place of a single time-series for any time-series boundary condition (e.g., inflow) expected by the alternative.

G.3.1 Defining an Ensemble Alternative

There are three basic steps in creating an ensemble alternative in ResSim:

- Set the **Alternative Type** to **Ensemble**
- Map one or more required inflows to a collection time.
- Select the members of the collection to use/compute in the alternative

These steps are described in more detail in the following sections.

G.3.1.1 Set the Alternative Type to Ensemble

Although you *can* create a new alternative and make it an ensemble alternative, the easiest way to create an ensemble alternative is to start with a standard alternative that is already working as desired. That's where these instructions are going to start.

To create an Ensemble Alternative in ResSim,

- In the **Reservoir Network** module of ResSim, open the **Alternative Editor**.
- Select an *existing* alternative that you have tested and validated in the **Simulation** module.
- Select SaveAs… from the Alternative menu of the Alternative Editor.
- Give your new alternative a unique name and click **Ok**.
- Your new alternative is now the active alternative in the Alternative Editor.

- Select the **Run Control** tab (Figure G.20) and change the **Alternative Type** radio button to **Ensemble**.
- Select **Save** from the Alternative menu to save your change.

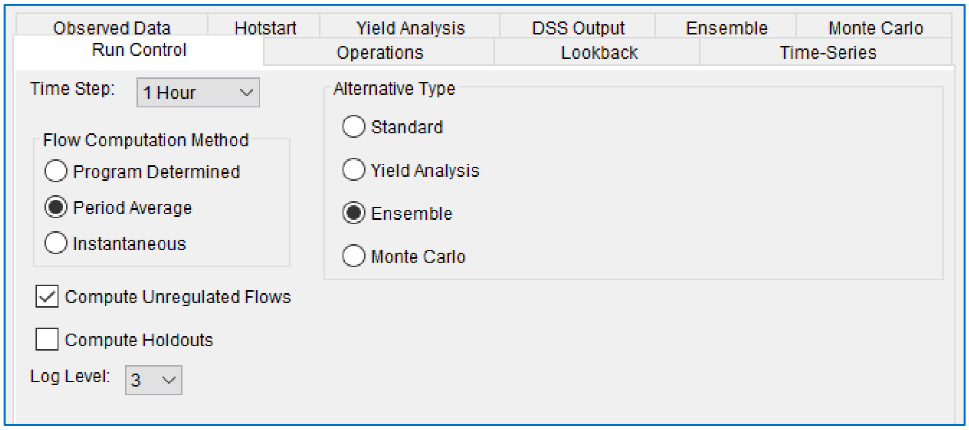


Figure G.20 Alternative Editor—Run Control Tab, Alternative Type

G.3.1.2 Map Required Inflows to Collections

Typically, the boundary conditions that are associated with ensembles are the inflows to your watershed. However, you should remember that your model may have other boundary conditions that could or should be considered for associating with ensembles. These include diversions, zone definitions, and any rule or other element that is defined as a function of an external time-series.

To associate (map) a boundary condition to a collection:

- Be sure that the current/active alternative in the Alternative Editor is an **Ensemble** alternative (check the **Alternative Type** on the **Run Control** tab).
- Select the **Time-Series** tab in the Alternative Editor. When the **Alternative Editor's** active alternative is an Ensemble alternative, the **Time-Series** tab will allow selection of collections for any input **Time-Series** (Figure G.21).

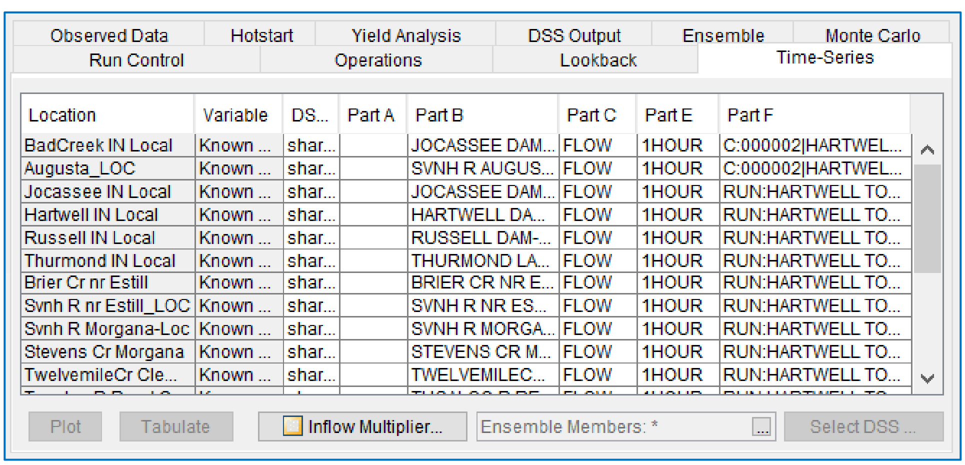


Figure G.21 Alternative Editor—Time-Series Tab, Replacing Single Time-Series with Collections

- Click the **Select DSS Path** button. A DSS Pathname Selector window will open. This window looks and acts a lot like DSSVue.

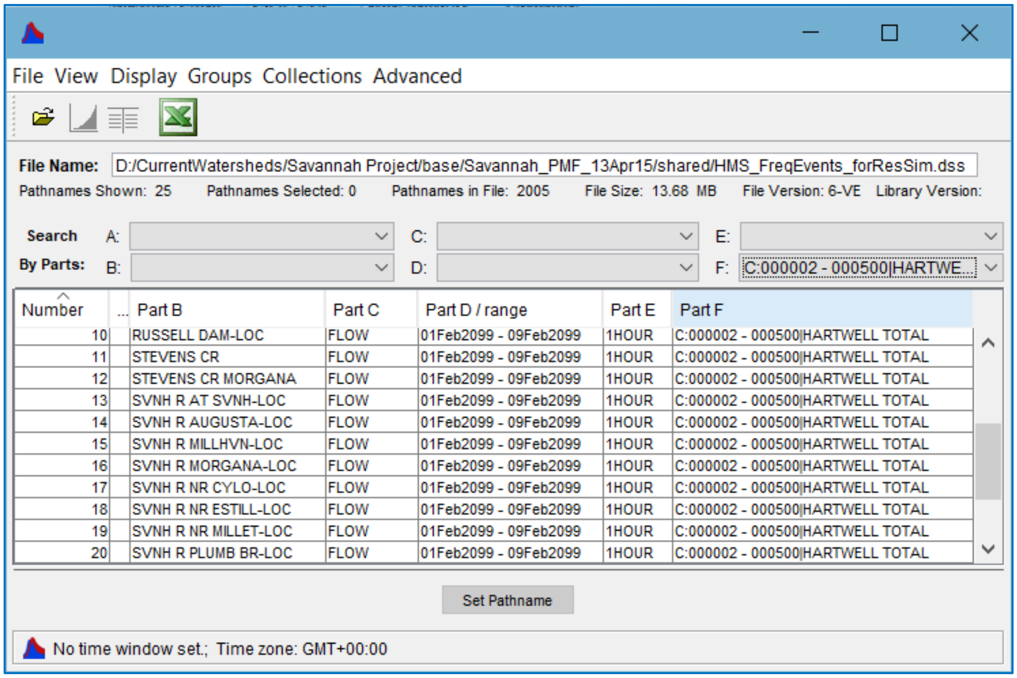


Figure G.22 DSS Pathname Selector Window

- Select **Condensed—Group Collections** from the **View** menu. The collection member pathnames will condense as illustrated in Figure G.22. Filter the pathnames to the specific collection F part you need.
- Move the Pathname Selector window so that you can see both it *and* the **Time-Series** tab of the **Alternative Editor** at the same time.
  - o On the Time-Series tab, select the first boundary condition row that you want to associate with a collection.
  - o In the pathname selector window, find and highlight the collection you want to associate with the selected boundary condition row on the Time-Series tab. Click the **Set Pathname** button or double-click the highlighted pathname. The pathname of *the first member of the selected collection* will appear in the selected row on the Time Series tab (Figure G.21).
  - o Repeat these steps for each boundary condition that should be mapped to a collection.
- When finished, close the pathname selector window and **Save** the alternative.

G.3.1.3 Select the Collection Members to Use in the Ensemble Alternative

Since collections can contain a large number of members, it may be desirable to use only a subset of the available members in your ensemble alternative. To specify which members of the collection to use:

- Select the **Ensemble** tab of the Alternative Editor
- In the **Use Ensemble Members*:** text box, enter the ID numbers of the collection members use/compute as part of the ensemble alternative (Figure G.23). If you want to use all the member of the collection(s), use the asterisk ∗ character. If you want to compute only a subset of the members of your collection(s), enter a comma-separated list of member IDs with each entry consisting of one member ID or a range of member IDs. For example:

  ∗ = all ensemble members

  "3, 7, 10" = ensemble members with IDs 3, 7, and 10

  "1, 3-12, 15" = ensemble members with IDs 1, 3 through 12, and 15

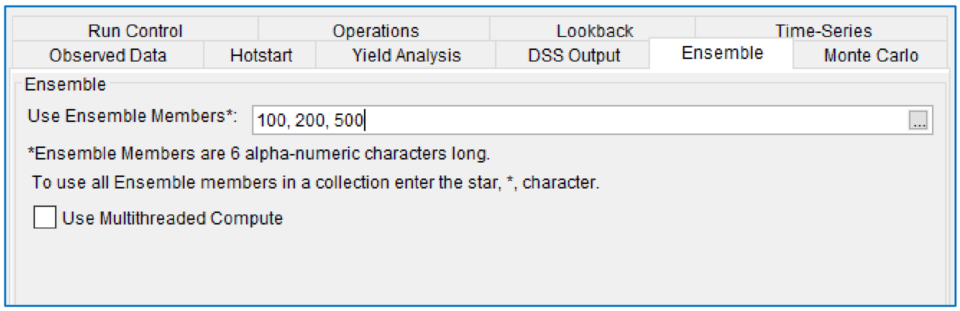


Figure G.23 Alternative Editor—Ensemble Tab

NOTE: An additional option is available on the **Ensemble** tab of the **Alternative Editor**. This option is labeled **Use Multithreaded Compute**. If you turn on (check) this option and your computer and operating system is capable of multithreading, then ResSim will determine the number of threads that can be run on your computer and will divvy-up the ensemble alternative's runs across the available threads. This can substantially reduce compute time for the ensemble alternative. For example, HEC's test computer had a dual-core CPU capable of "hyper threading" making it appear that the computer had a quad-core CPU. The multithreading logic in HEC-ResSim believed the computer was capable of 5 threads. It took an ensemble alternative of 30 members and assigned 6 members to each of the 5 threads. The resulting compute time was approximately 1/4th of the compute time that the same 30-member ensemble alternative took without multithreading.

- Look back at Figure G.21. If you look at the row of buttons at the bottom, you will notice a new box that shows the set of ensemble members identified on the **Ensemble** tab.

G.3.2 Viewing Ensemble Reports

ResSim's standard reports have been updated to recognize when they are provided an ensemble alternative, but they will only show results for a single ensemble member at a time. By default, the report will open showing the results for the first computed member of the ensemble. You can change which member is displayed by selecting **Set Collection Run** from the **Options** menu of the report, then choosing the desired member's ID number from the **Collection Run Options dialog** (Figure G.24).

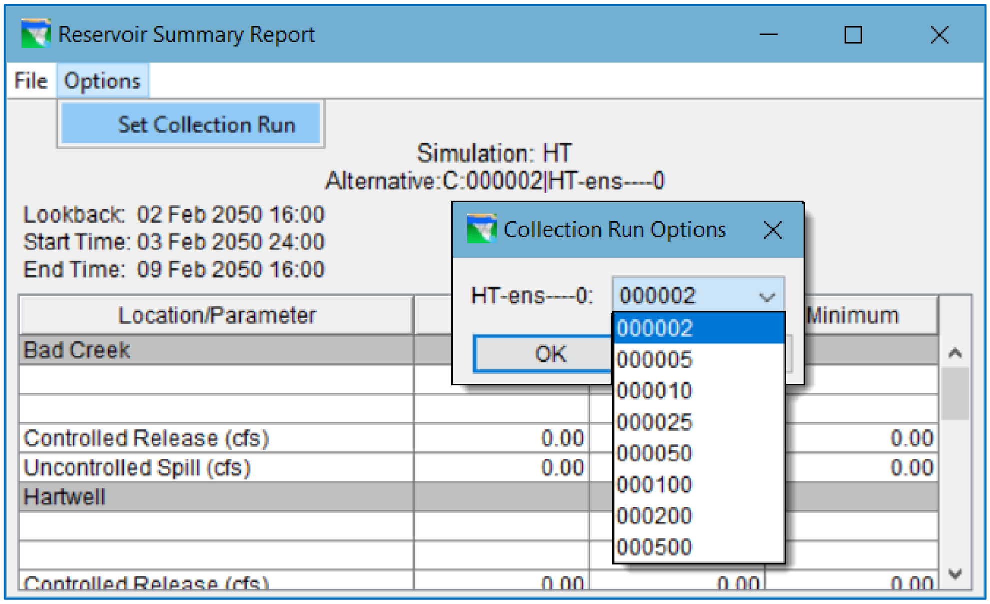


Figure G.24 Reservoir Summary Report—Set Collection Run

HEC-ResSim will remember the selection when you view other reports.

G.3.3 Plotting Ensemble Results

The standard ResSim (and user) plots that you can produce from the context menu of an element in the active schematic have been enhanced to recognize when they are sent an ensemble alternative's results. Unlike the standard reports, ResSim plots will show all the computed ensemble members' results at the same time.

Since there can be many computed members in an ensemble alternative, viewing a rainbow of curves is usually not conducive to understanding the overall results. So, in an effort facilitate results analysis, a new plotting paradigm was created: for each plotted parameter, all members of the ensemble (traces) are drawn in the same color with a slightly thinner line and with a moderately-high level of transparency. When several traces all plot on top of each other, the density of color increases and you can see where the results overlap. To also aid in analysis, five additional curves are added for each plotted parameter to reflect the minimum, maximum, median, and 25% and 75% probabilities computed across the computed members of the ensemble for each timestep. These additional curves are drawn with a thicker line weight and zero transparency.

To plot ensemble results:

- Select an ensemble alternative by placing a checkmark in the box beside it in the **Simulation Control** panel of an active simulation. Although ResSim will plot multiple alternatives at the same time, select only one alternative at a time until you get used to viewing the ensemble results.

- Right-click on an element in the schematic that you want to view results for then select the predefined (or user) **Plot** you wish to view.

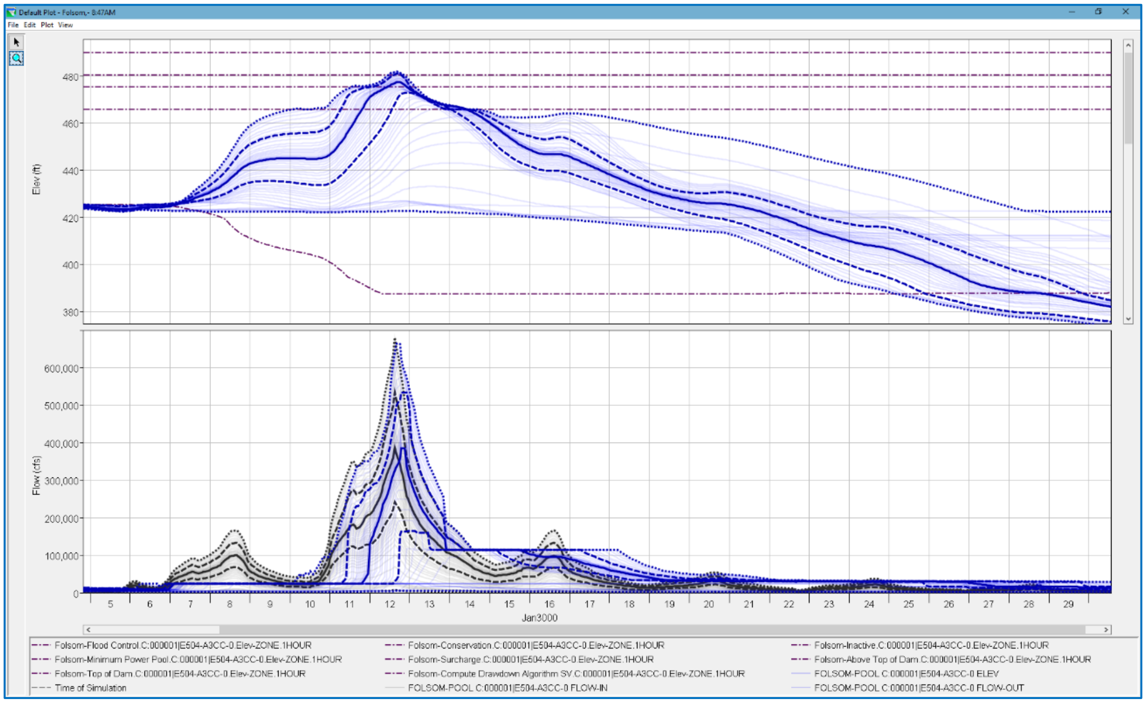


Figure G.25 Ensemble Plot for a Reservoir

An example of a standard reservoir plot is shown in Figure G.25. Although the text in legend in the illustration is quite tiny, it can be noted that there are far fewer entries in the legend than there are curves drawn on in the plot. This is because the legend identifies the results by parameter and collection, not by individual member. In addition, the curves for the computed statistics are not listed since they are considered members of each collection, not separate datasets.

The same plot editing features are available for an ensemble alternative plot that are available for a standard plot. *However, the implementation of the new plotting paradigm used for ensembles is not as complete as you might expect, so some of the plot editing features may not function correctly on an ensemble plot. This will be improved in a future release of ResSim.*

Appendix H
Monte Carlo Analysis

Monte Carlo analysis is a feature in ResSim that enables you to evaluate the impacts on simulation results due to the uncertainty associated with certain input information in your reservoir model, as well as the subsequent uncertainty in those results. This feature uses random sampling of user-selected input variables within specified probability distributions.

Monte Carlo analysis is performed using a ResSim Alternative type called Monte Carlo. When you compute a Monte Carlo alternative, ResSim orchestrates iterative simulations of the alternative based on random sampling of one or more user-selected input variables. The following six types of variables can be selected for random sampling: Time Series Multipliers, Input Time Series, Reservoir Rule Parameters, Lookback Values, Rating Curves, and Scripts. For each input variable, you can select from seven probability distribution types for defining the statistical distribution of each random input variable including: Normal, Log-Normal (natural logarithm and base 10), Gamma, Empirical, Triangular, Uniform, and Discrete distributions. Based on your input parameters for a selected distribution type, HEC-ResSim's Monte Carlo analysis computes the Probability Density Function (PDF) and Cumulative Distribution Function (CDF) for the random variables, and then randomly samples values from the CDF. Output variables and convergence criteria are specified for the Monte Carlo analysis and may use time-series summary values (Maximum, Minimum, Mean, or Volume). You can also specify options for defining the minimum and maximum number of simulation iterations, for continuing the analysis when additional iterations are desired to improve convergence, for restarting the analysis when previous simulation results are desired to be cleared, and for which iterations should save their full standard output.

H.1 ResSim Monte Carlo Alternatives

To perform a Monte Carlo Analysis, you must use a Monte Carlo alternative type. To do so, select the Monte Carlo radio button on the Run Control tab of the Alternative Editor (Figure H.1).

Tip: When creating a Monte Carlo alternative, setup a *Standard* alternative first and get it working. Then, on the Run Control tab of the Alternative Editor, change the Alternative Type from Standard to Monte Carlo.

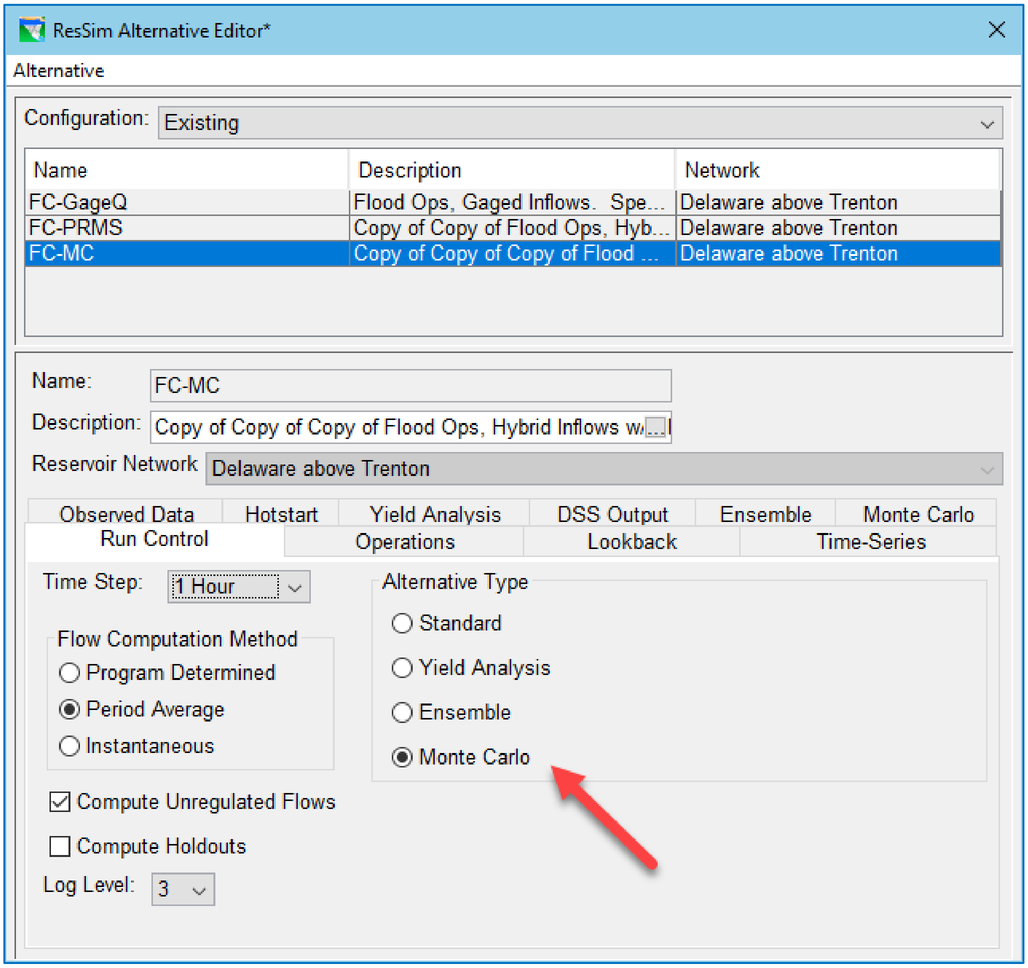


Figure H.1 ResSim Alternative Editor—Run Control Tab—Monte Carlo Alternative Type

H.2 Setting Up a Monte Carlo Alternative

In addition to the **Operations**, **Lookback**, and **Time Series** data needed for any alternative, the setup for a Monte Carlo alternative has three other data requirements. The first is the set of *input variables* that will be randomly sampled with each Monte Carlo iteration. The second is the set of *output variables* that are of interest, and that you expect to be impacted by the varied input. The third is the specification of the Monte Carlo iteration scheme and convergence criteria for the analysis. This information can be defined on the three sub-tabs of the **Monte Carlo** tab in the ResSim **Alternative Editor** (Figure H.2).

When a Monte Carlo alternative is run in a ResSim Simulation, it will create and compute new iterations until (a) the convergence criteria is met, or (b) the maximum number of iterations is reached.

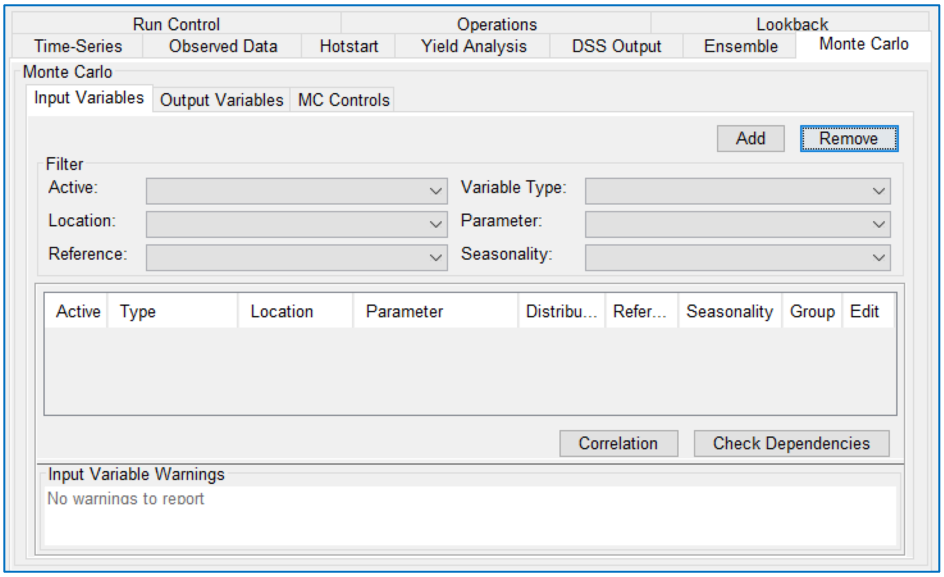


Figure H.2 ResSim Alternative Editor—Monte Carlo Tab—Input Variable Sub-Tab

H.2.1 Input Variables

To identify a Monte Carlo **Input Variable** that you want to have randomly sampled:

- Open the **Alternative Editor**, select your Monte Carlo Alternative, and select the **Monte Carlo** tab.
- Select the **Input Variables** sub-tab (Figure H.2).
- Click the **Add** button at the top right corner of the tab. The **Random Variable Wizard** will open (Figure H.3) to step you through the process of defining an input variable.

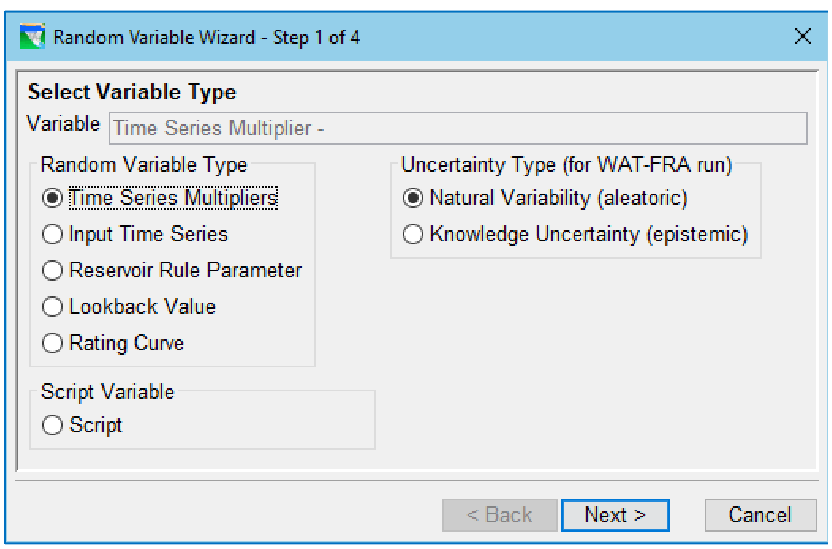


Figure H.3 Random Variable Wizard—Step 1—Select Variable Type

Step 1 of 4: Select the Random Variable Type

The randomly sampled input variable can be a standard ResSim input variable needed by your model or a *script variable* computed by a Jython script you write yourself.

- Click the radio button next to the **Random Variable Type** of your choice.

  The ResSim input variable types that are available to be randomly sampled during a Monte Carlo iteration include:

  - **Time Series Multiplier**—by varying the inflow multiplier (a scalar value) on one or more inflows to your model, you can investigate the impact of the uncertainty of the inflow magnitude on your system. For example, you can determine whether a 10% variance in reservoir inflow have any impact on downstream flooding.
  - **Input Time Series**—by providing a variety of inflow shapes for a given probability event, you can investigate the impact of the uncertainty on the inflow duration. For example, if you provide a collection of inflow hydrographs that represent the same event volume but with different durations and peaks, you can determine if the hydrograph shape has an impact on downstream flooding.
  - **Reservoir Rule Parameter**—by varying the magnitude of a rule limit, you can determine if the uncertainty in that operating constraint has a significant impact on the system. For example, if you vary the maximum channel capacity, you can determine if the maximum pool elevation is significantly affected.
  - **Lookback Value**—you can vary the starting condition(s) of your reservoir, diversion, or state variable to determine if the uncertainty in starting condition has a significant impact on the operation of your system. For example, you can determine if there is a significant relationship between starting pool elevation and downstream flooding or peak pool elevation
  - **Rating Curve**—since stream morphology is ever changing, you can investigate the impact that the uncertainty of the relationship between flow and stage has on your system by varying the stage value returned by the rating curve lookup.

  Or, for additional flexibility beyond the currently available options, you can define your own **Script Variable** by selecting:

  - **Script**—you can develop your own sampling method if standard options are not sufficient. By utilizing script variables and dependencies, you can also incorporate two-stage sampling. Although defined as an input variable, these sampling scripts are really only intermediate variables that another random input variable *depends* on; dependency is defined by clicking on the ellipsis (...) button next to the distribution parameter field—see Step 4 of 4 below.

If you are setting up your Monte Carlo alternative for use in an **HEC-WAT Flood Risk Analysis (FRA)** run, variables can be sampled at different levels of a nested compute loop, so you must select the **Uncertainty Type** for your input variable.
Click the radio button next to the **Uncertainty Type** of your choice. Your options include:

- **Natural Variability**—Natural Variability variables are sampled in the inner WAT loop with each event.
- **Knowledge Uncertainty**—Knowledge Uncertainty variables are sampled in the outer WAT loop with each realization

See the HEC-WAT User's Manual for more information.

Once you have chosen your input variable's type,

- Click **Next.** This will take you to Step 2 of 4, the **Select Location and Parameter** step. The list of locations and variables will vary depending on the variable type you selected.

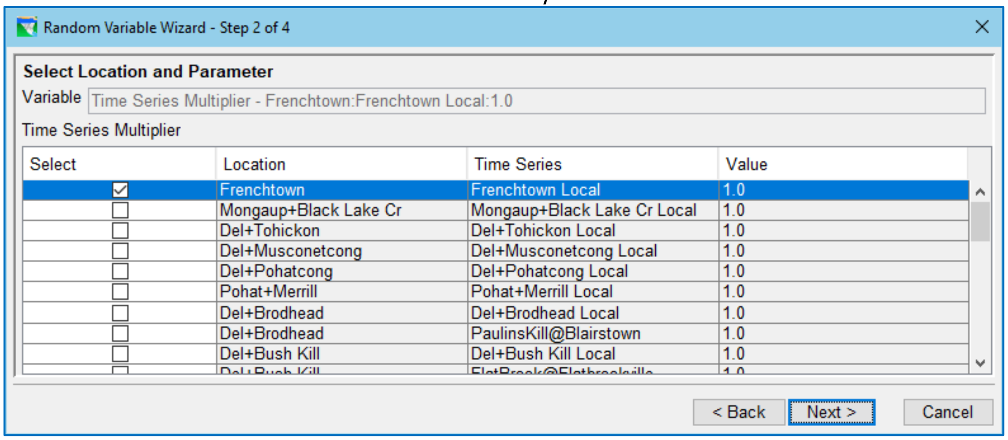
Step 2 of 4: Select *a specific* Location and Parameter

Based on the **Input Variable** ***Type*** selected in the previous step, the **Random Variable Wizard** will provide you with a list of locations and parameters of that type to choose from. Examples of these lists are shown in Figure H.4. The initial value of each scalar variable is shown in the applicable list.

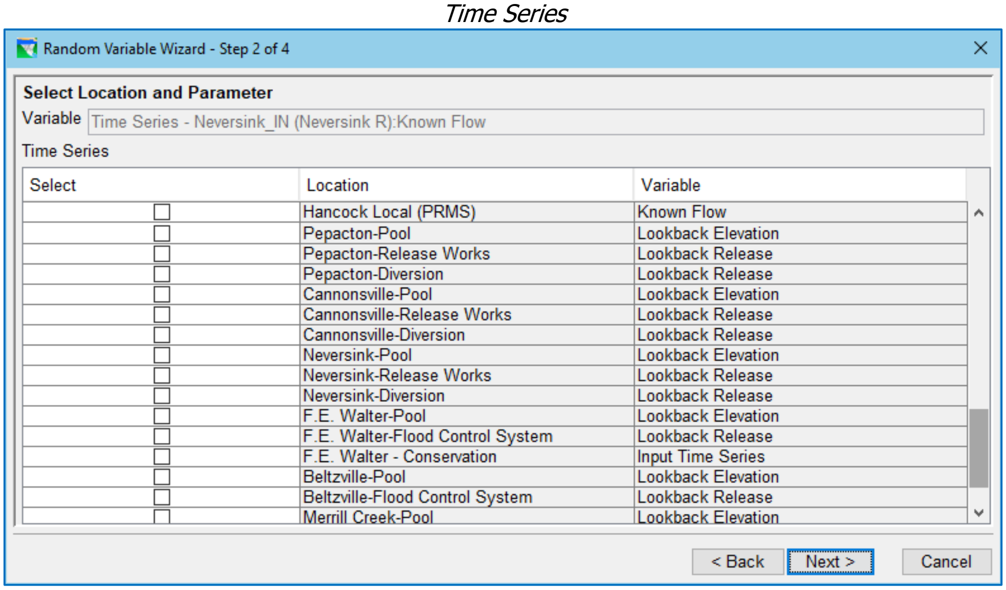
Except for **Input Time Series** and **Script**, each input variable type will let you select more than one variable of that type. For example, the **Reservoir Rule Parameter** variable type lets you select more than one rule, from one or more reservoirs. If you select multiple variables in this step, they will form a single **input variable *group***, and the same random number and relative probability distribution will be applied to each variable in the group.

- Select the input variable from the list of variables of the type you selected in the previous step.
- Click **Next**. This will take you to Step 3 of 4 to select the **Reference-** and **Time- Dependency** types for your input variable's distribution parameters.

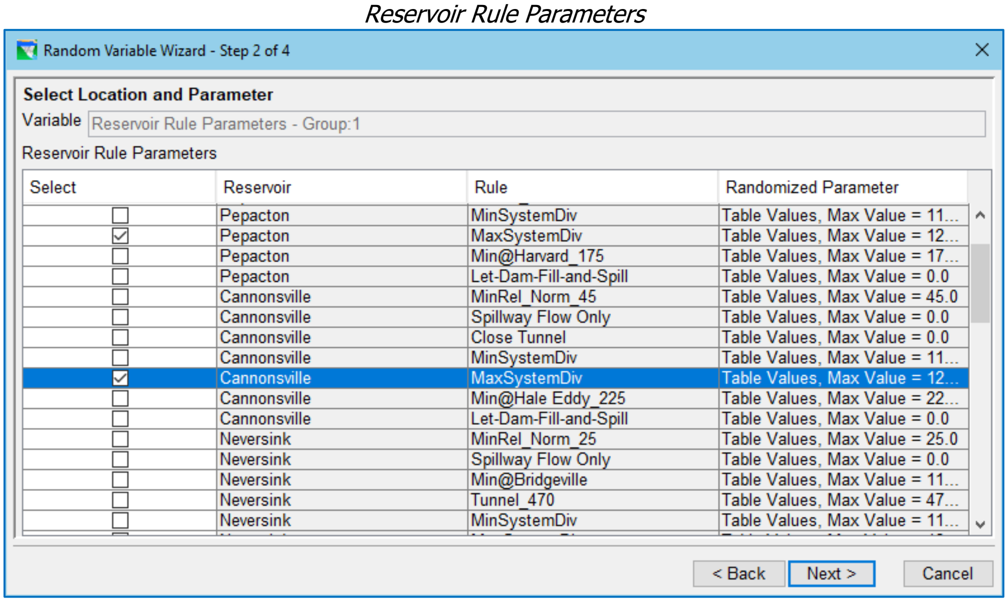
Inflow Multiplier



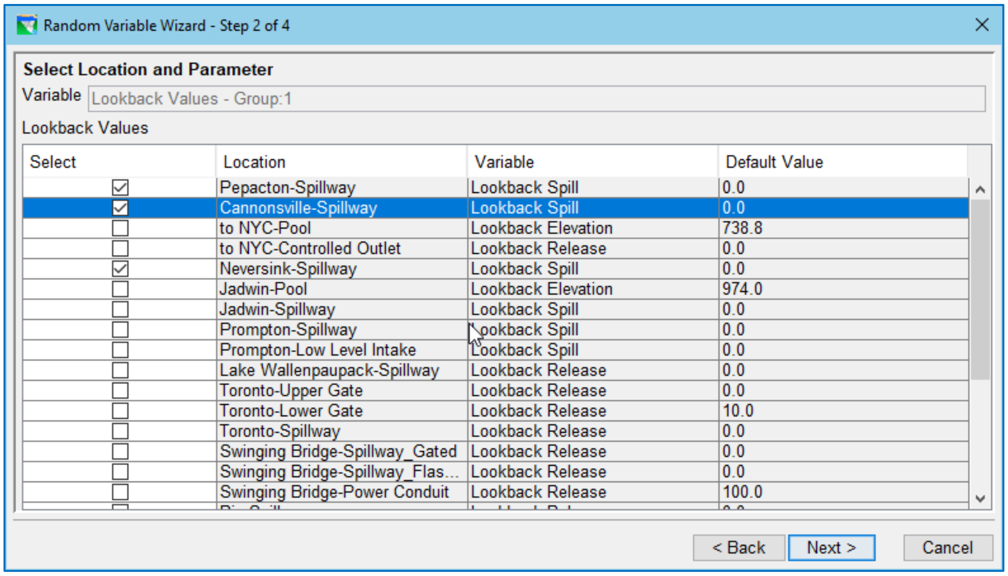
Time Series



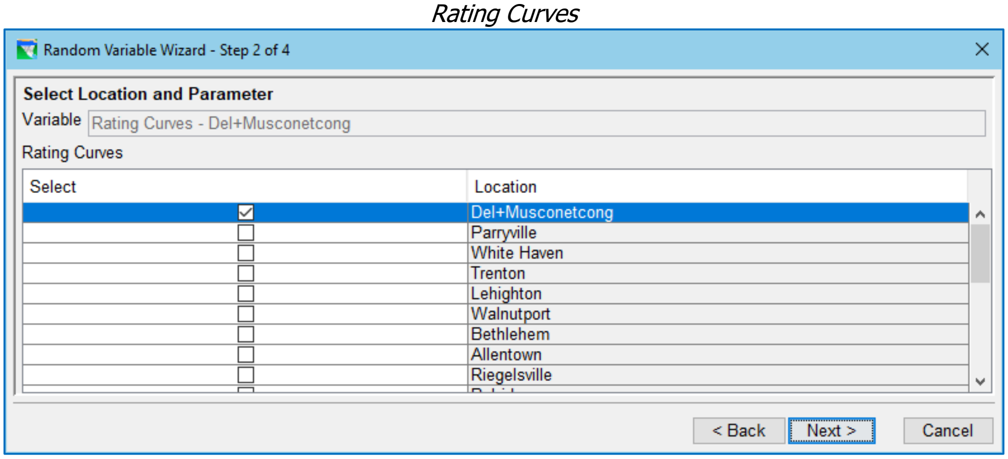
Reservoir Rule Parameters



Lookback Values



Rating Curves



Script

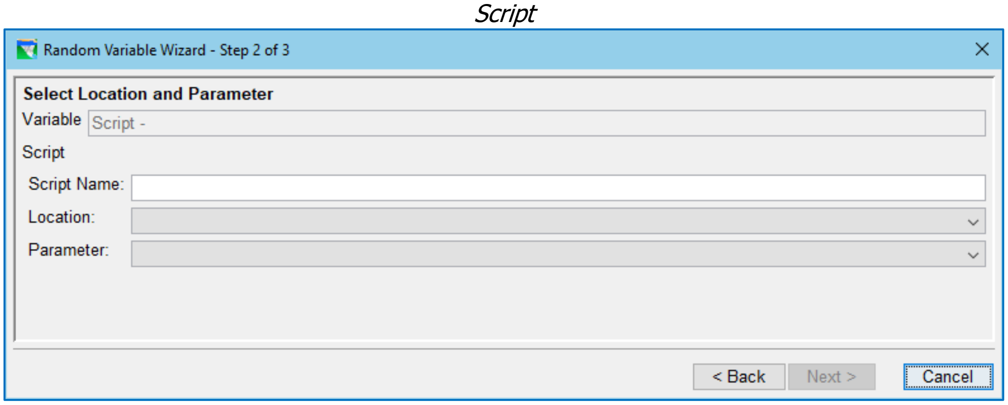


Figure H.4 Random Variable Wizard—Step 2—Select Location and Parameter for each Variable Type

Step 3 of 4. Select Variable Parameters Modality

In step 3, you will select how the probability distribution parameters (defined in step 4) will be specified for your input variable or variable group (Figure H.5).

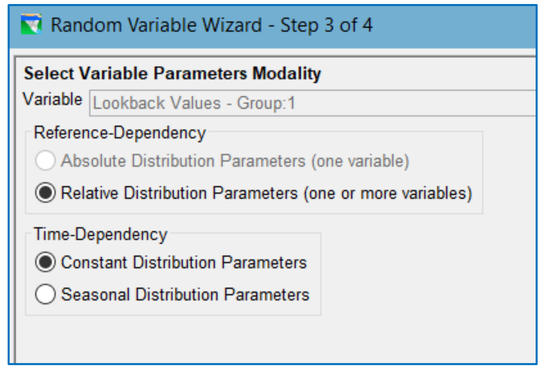


Figure H.5 Random Variable Wizard—Step 3—Reference- and Time-Dependency

- **Reference-Dependency**—If you selected only one variable (rather than a *group* of variables) to be associated with your input variable, then you will have the option to specify whether that input variable will use an absolute probability distribution or a relative probability distribution. Otherwise the **Relative Distribution** option will be selected for you.
  - **Absolute Distribution Parameters**—the parameters of the input variable's probability distribution are explicitly defined and have no relationship to the original value of the variable. You will specify the distribution using the units of the input variable.
  - **Relative Distribution Parameters**—the parameters of the input variable probability distribution are *relative* to the original value of the variable. If different variables of a given type have been defined as an input variable *group,* it is unlikely that they would each share an identical absolute distribution, so the shared probability distribution must be defined with relative parameters that are based on the initial values of each. You will specify the parameters of your probability distribution as percentages of the initial value of the variable.
- **Time-Dependency**—While only one probability distribution type can be selected for each variable or group, you can specify whether the parameters of that distribution will be held constant throughout year or vary seasonally. As such, you must choose between:
  - **Constant**—this option will allow you to define only one set of distribution parameters in the next step of the wizard.
  - **Seasonal**—this option will allow you to vary your distribution parameters seasonally. With this option, in the next step of the wizard, you will need to define the seasons and then the distribution parameters that will apply for each season
- Once you have selected the modality for your input variable, click **Next**.

Step 3 of 3: Script Variable

For a **Script** input variable, **Step 2 of 3** required you to specify a unique name for your script variable and to associate the script with a model location (element) and parameter. **Step 3 of 3** of the **Random Variable Wizard** is the script editor, much like ResSim's state variable editor, which will allow you enter the script that defines your sampling scheme (Figure H.6). This script will receive the random value generated for it in the current Monte Carlo iteration and can use that value any way you wish to compute the script variable's value. A stochastic script may use this number to generate randomness, or it may be ignored in the case of a deterministic script. Similarly, the Monte Carlo alternative data is passed as a parameter (*mcData*). Using the Monte Carlo Variables section of the tree on the left side on the Script Editor panel allows the user to quickly add code to retrieve information about other Monte Carlo input variables. If you retrieve data from another Monte Carlo variable, ResSim assumes that the script is *dependent* on that variable. A return value must be specified in each sampling script; this value may be passed to other variables or sampling scripts dependencies.

Note: When using a script for the random variable, Step 3 is the last step since no distribution is needed. The other types of random variables require a fourth step to set up the distribution.

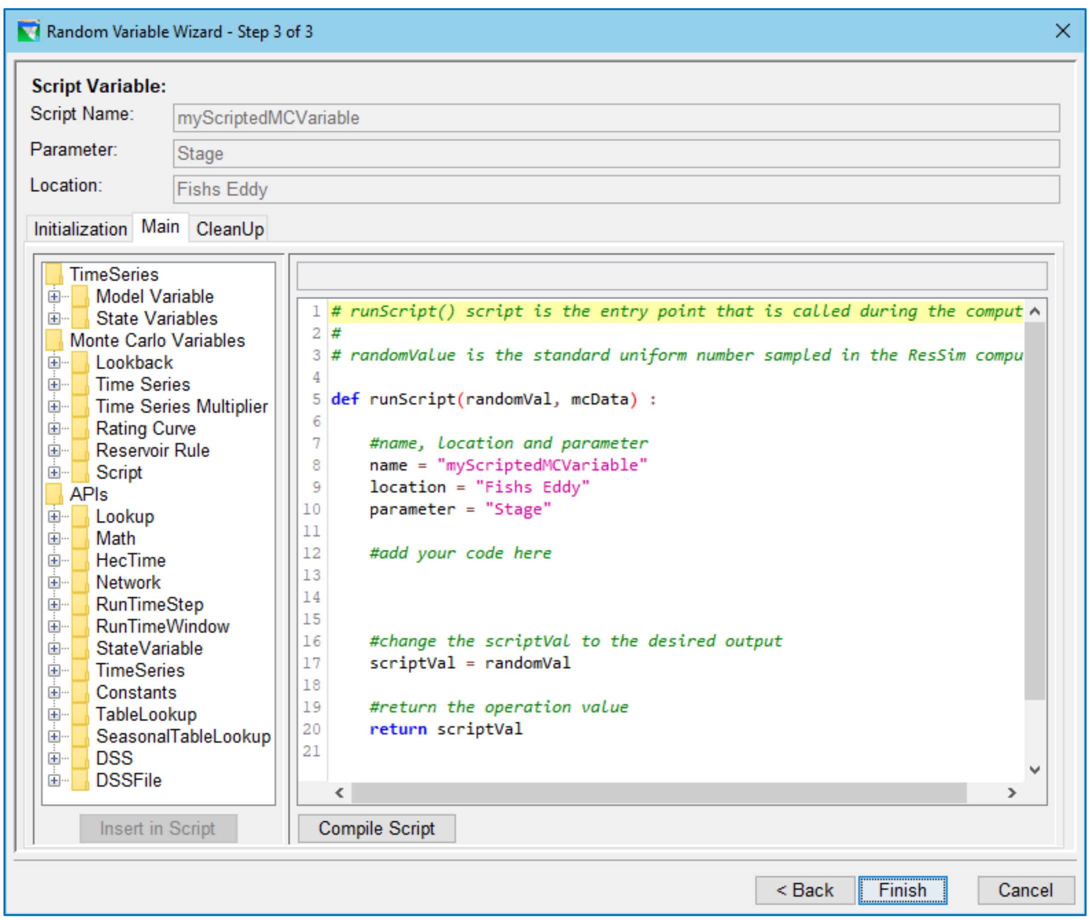


Figure H.6 Random Variable Wizard—Step 3—Script Variable Editor with Script Variable Template

Step 4 of 4. Select and Configure a Distribution Type

For each input variable, the value randomly sampled will be from the probability distribution type and parameters you specify.

- Select a **Distribution Type**. Your distribution type options (Figure H.7) are:
  - Uniform
  - Triangular
  - Normal
  - LogNormal
  - Log10Normal
  - Gamma
  - Empirical
  - Discrete

  These distribution types and their required parameters are described in the next section, Section H.2.2 of this appendix.

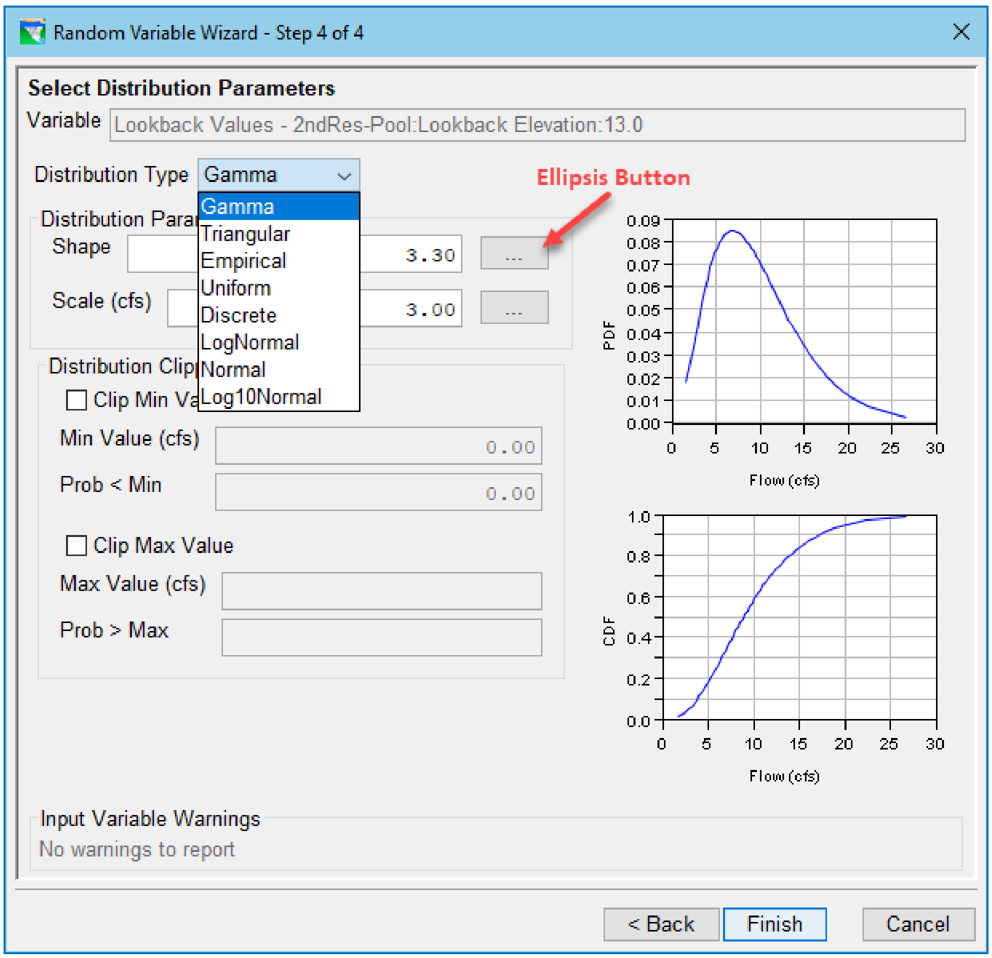


Figure H.7 Random Variable Wizard—Step 4—Select Distribution Type

- Next, enter values for the distribution parameter values needed for the **Distribution Type** you selected. How you specify these parameters will depend on what *modality* selections you made in **Step 3 of 4**:
  - If you selected **Absolute** for the **Reference-Dependency**, then you will define the parameters in the units of the input variable (except for dimensionless parameters).
  - If you selected **Relative** for the **Reference-Dependency**, then you will instead define the parameters as a percentage of the initial values of the variable.
  - If you selected **Constant** for the **Time-Dependency**, then a single field for entering each distribution parameter value will be displayed.

    If you wish to make the distribution parameter *dependent* on another variable, click the ellipsis button that follows the parameter field (as annotated in Figure H.7). The **Variable Dependency** dialog (Figure H.8) will open showing a list of the input variables you have already defined. Select a variable whose value will be used as the parameter value then click **OK** to accept the selected variable and close the dialog; or click **Cancel** to return without defining a dependency.

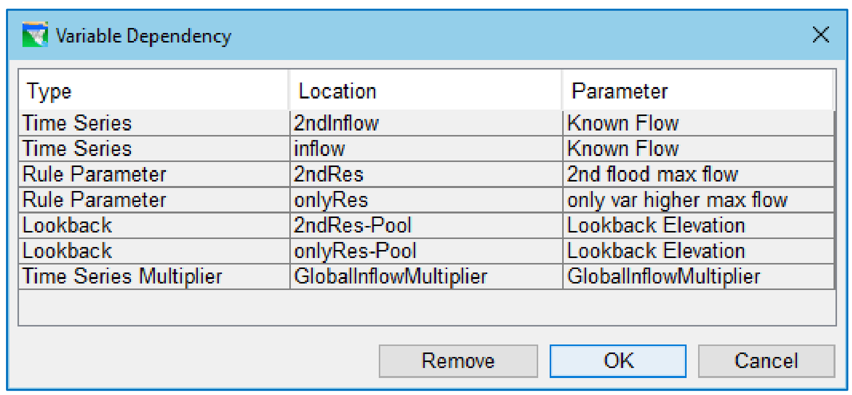


Figure H.8 Variable Dependency Dialog

If you created a variable dependency, the parameter field and ellipsis button in the **Select Distribution Parameters** panel of the Random Variable Wizard will be replaced with a button identifying the selected dependent variable. If you wish to change or delete the dependency, click the dependency button to reopen the **Variable Dependency** dialog. Use the **Remove** button to delete the dependency or select a different variable to change the dependency.

- If you selected **Seasonal** for the **Time-Dependency**, then an **Edit Seasons** button will appear above a table for entry of the seasonally varying distribution parameters. Click the **Edit Seasons** button to open the **Seasonal Variation** dialog (Figure H.9), then enter the start date of each season in the table. When done, click **OK**. A column for each season will appear in the **Distribution Parameters** table (Figure H.10). Enter values for each parameter for each season in the table.

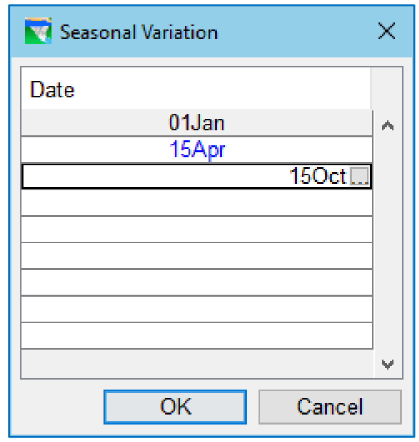


Figure H.9 Seasonal Variation Dialog

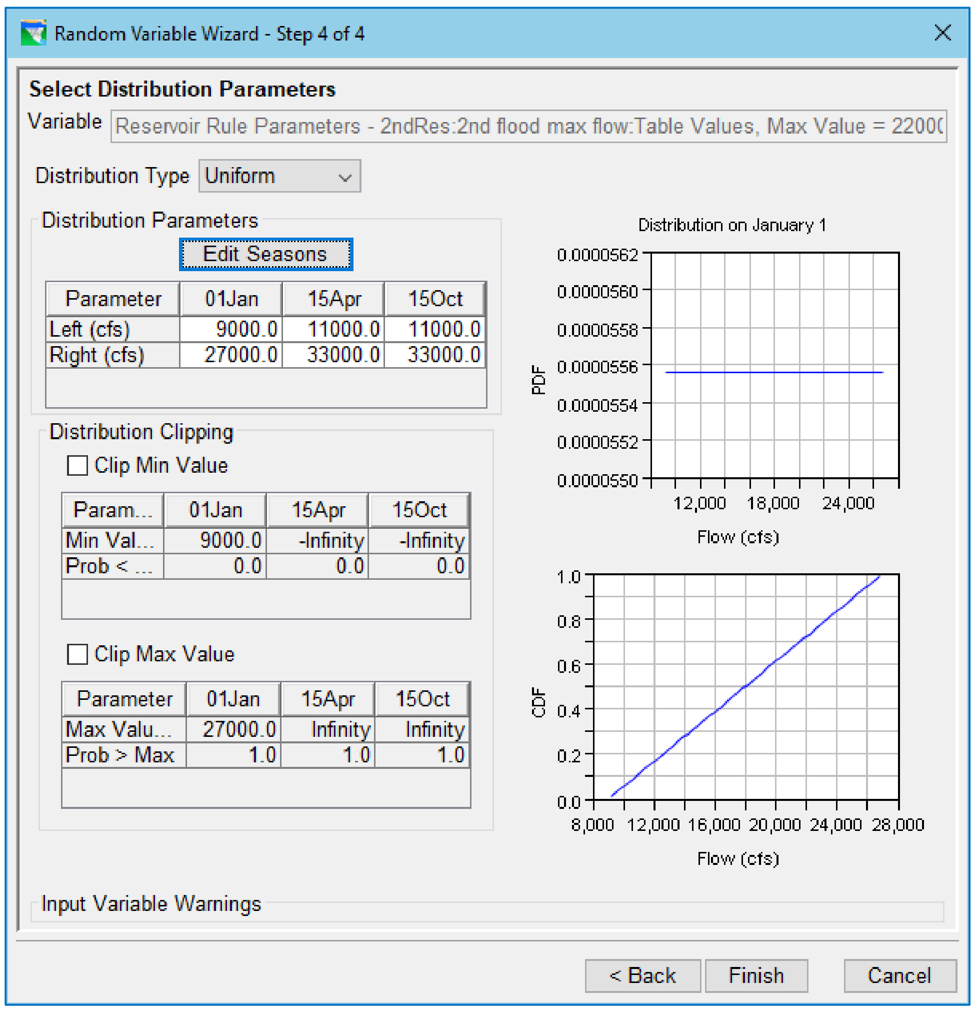


Figure H.10 Random Variable Wizard—Step 4—Select Distribution Parameters—Seasonal Variation

- Next, select the **Distribution Clipping** options you want to apply to your probability distribution. Like the distribution parameters, the clipping options are a function of the modality options you selected in Step 3:
  - If you selected **Absolute** for the **Reference-Dependency**, then the **Distribution Clipping** section will allow you to limit the range of your distribution by setting a minimum and/or a maximum limit (Figure H.10).
  - If you selected **Relative** for the **Reference-Dependency**, (or if your input variable represents a *group* of variables), then the **Distribution Clipping** section will only provide a **Clip at Zero** option allowing you to limit the distribution to positive values (see Figure H.11).

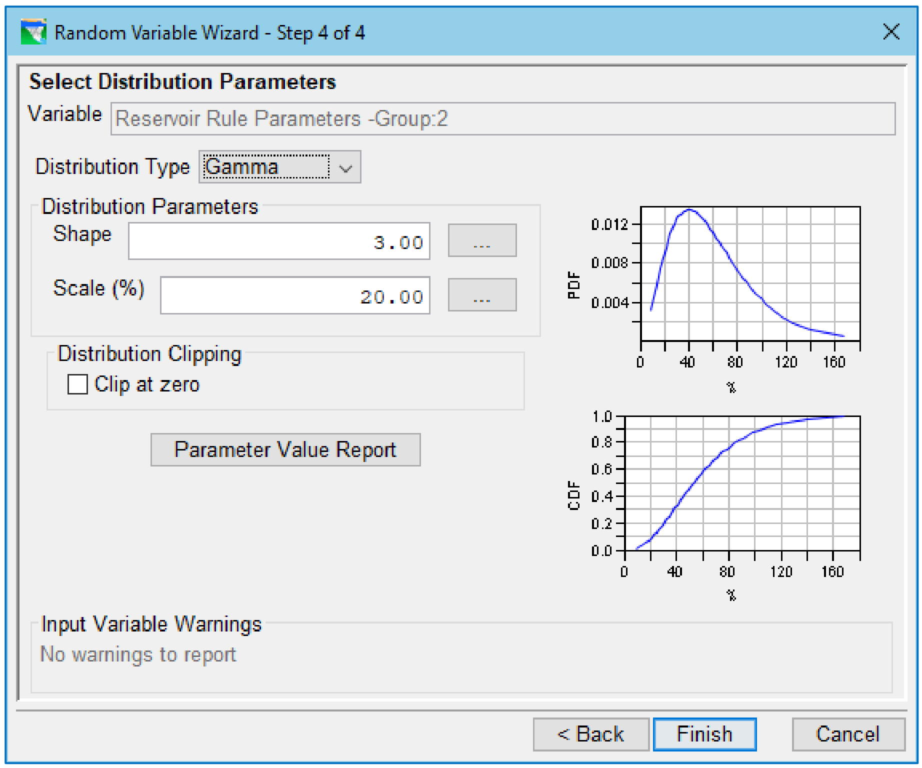


Figure H.11 Random Variable Wizard—Step 4—Select Distribution Parameters for Variable Group

When the **Reference Dependency** is **Relative**, a **Parameter Value Report** button will appear below the Distribution Clipping section. Click the **Parameter Value Report** button to see how the selected parameters will translate from relative to absolute for each variable in the group. An example of this report is shown in Figure H.12.

Note that dimensionless parameters (like shape in the image) are not scaled, and so, show as *N/A* in the relative column of the Parameter Value Report.

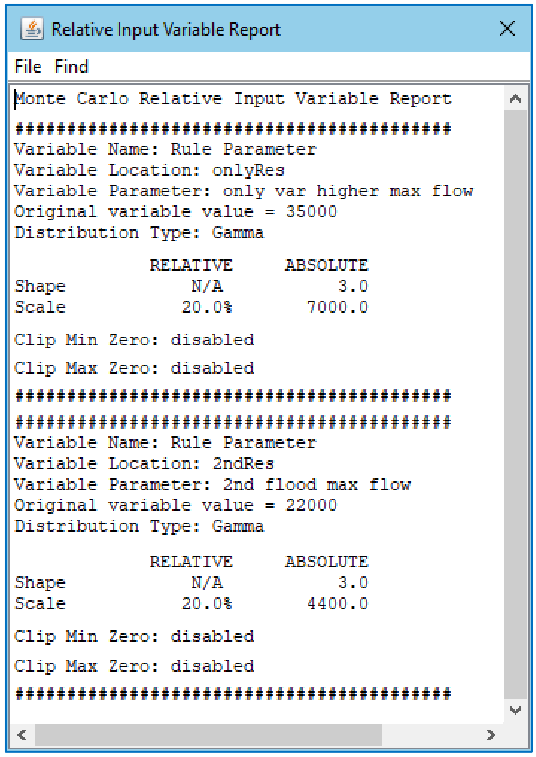


Figure H.12 Relative Parameter Distribution—Paramter Value Report

- Once you have completed the setup of your input variable, click **Finish** to close the **Input Variable Wizard**. A row will appear in the table on the **Input Variables** sub-tab that represents the input variable you just defined (Figure H.13)
- Since a large model could require a significant number of input variables, the table of input variables is preceded by a set of **Filters** that you can use to thin out the list so that you can find a specific variable of interest.
- If you have specified several input variables you may choose to deactivate one or more of them by unchecking the **Active** checkbox at the start of their row in the table. Only "Active" input variables will be randomized during the run of your Monte Carlo alternative (Figure H.13).
- You may also choose to specify a linear *correlation* between two or more of your active input variables by selecting the **Correlation** button located below the list. This will open the **Correlation Matrix** editor (Figure H.14). Use this editor to create a **Correlation Set** (a selection of input variables) and the correlation matrix that relates the probabilities of the variables in the set to one another. The diagonal cells of the correlation table will always have a value of 1, as any variable's correlation with itself is perfect. All other cells will default to zero, implying independence. A value between -1 and 1 entered into an off-diagonal cell indicates correlation between the row and column variables. Only the entries below the diagonal are editable. The entries above the diagonal are updated automatically to show that the matrix is symmetric.

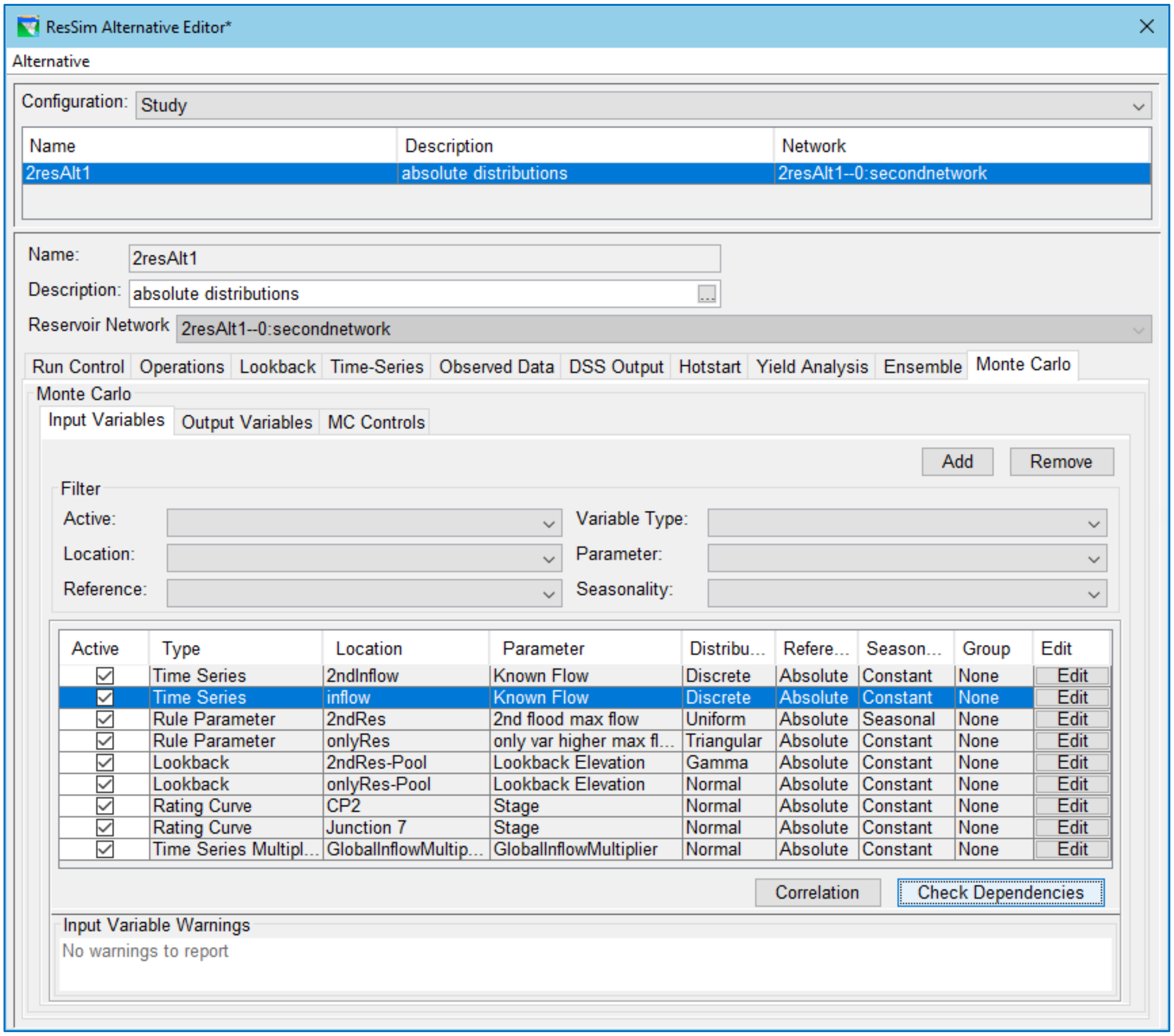


Figure H.13 Monte Carlo Tab—Input Variables Sub-tab with several Variables Defined

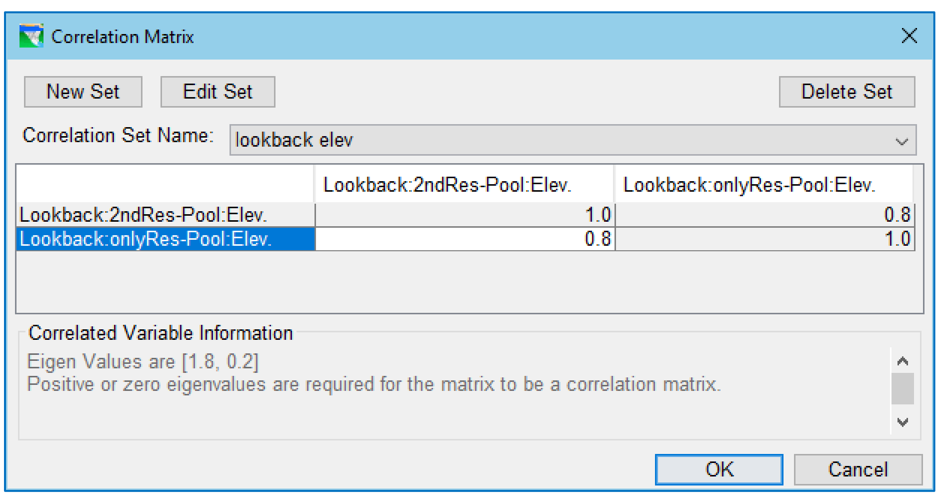


Figure H.14 Input Variable Correlation Matrix Editor

- The **Check Dependencies** button will display a graphical representation of the dependencies you have defined between your input variables. Creating variable dependencies is an advanced feature, so don't be surprised if you get a message saying that "No dependent variables have been defined…".

H.2.2 Random Variable Distribution Types

Gamma Distribution

The gamma distribution (Figure H.15) is a two parameter, flexible, continuous probability distribution. The two parameters are **Shape** and **Scale**. This distribution can be either symmetrical or positively skewed.

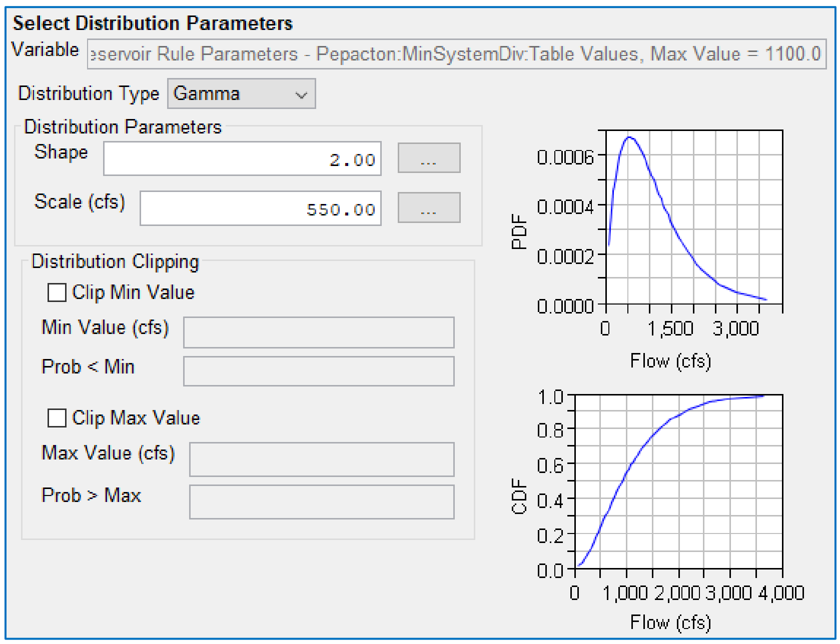


Figure H.15 Input Variable Wizard—Gamma Distribution

Triangular Distribution

The triangular distribution (Figure H.16) is a three-parameter continuous probability distribution. The three parameters are **Left** (minimum value), **Right** (maximum value) and **Peak** (the mode, or most-likely value, where the peak of the triangle is located). This distribution is useful in that it can be symmetrical, or it can be asymmetrical with either a positive or negative skew, i.e., a longer upper or longer lower tail. For example, a negative skew allows the value to be farther below the most-likely value than it can be above.

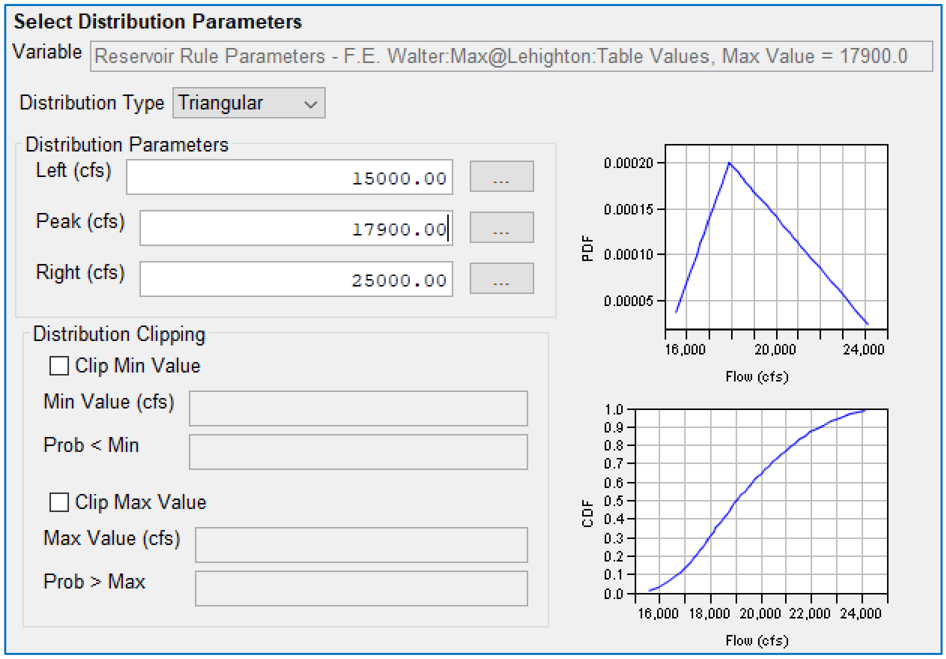


Figure H.16 Input Variable Wizard—Triangular Distribution

Empirical Distribution

An empirical (graphical) distribution (Figure H.17) is continuous but without a function, for which the likelihood of possible values is derived from observed data. The empirical distribution is represented by a lookup table that describes points on the cumulative distribution function (CDF).

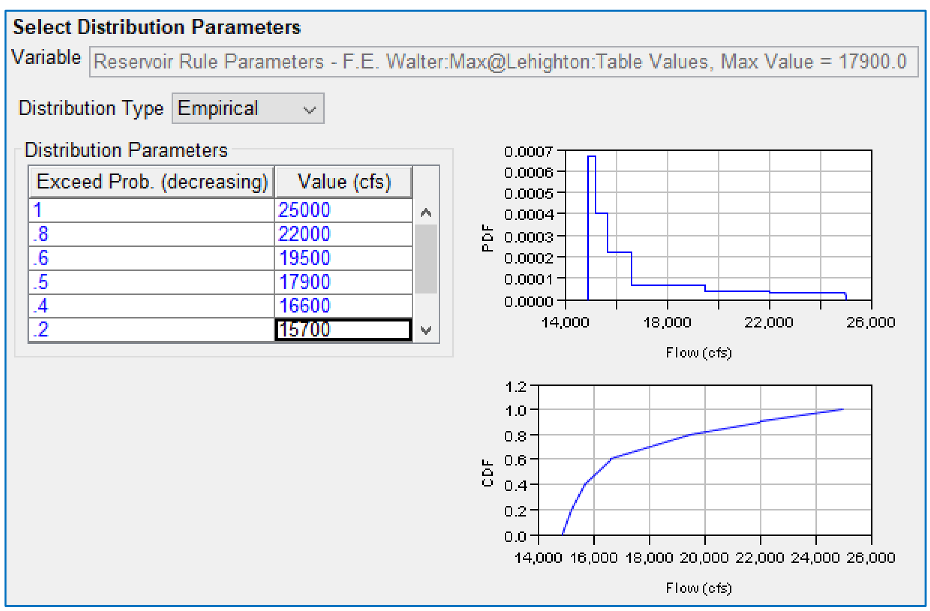


Figure H.17 Input Variable Wizard—Empirical Distribution

Uniform Distribution

A uniform distribution (Figure H.18) is a distribution that has a constant probability across the possible range, with any value having an equal likelihood or probability. The parameters for this distribution are **Left** (minimum value) and **Right** (maximum value), defining the possible range.

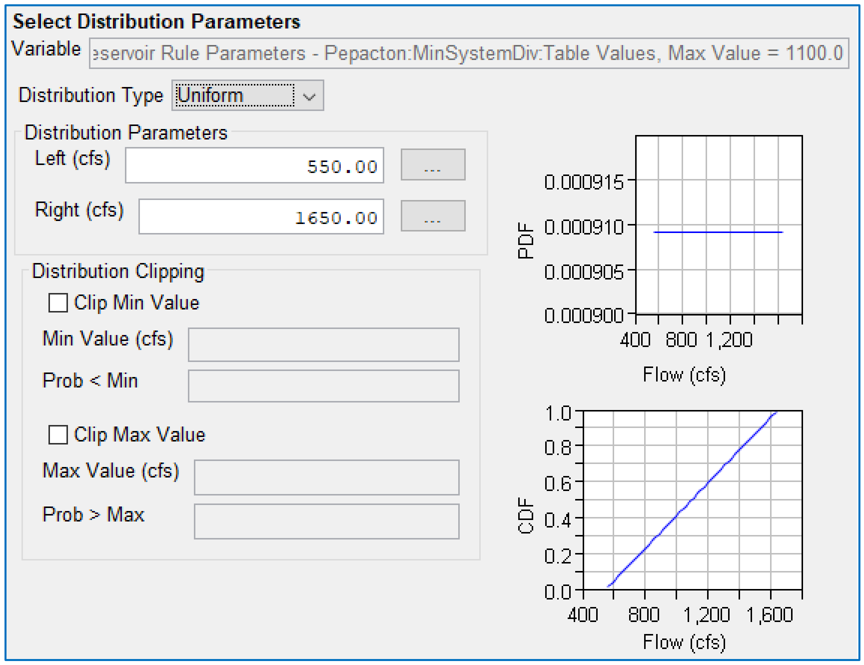


Figure H.18 Input Variable Wizard—Uniform Distribution

Discrete Distribution

A discrete distribution (Figure H.19) describes the probability of occurrence of each possible value of a discrete random variable. A discrete random variable is a random variable that has a limited set of possible values, such as the values on the faces of a six-sided die. A discrete distribution is defined by a table of individual values and their probabilities.

If your input variable is a time series, such as reservoir inflow, the only distribution type available is **Discrete**, since sampling for an input time series is done only from a set of time series records that you identify. Figure H.20 shows how the parameter table for a discrete distribution changes for an input time series variable, as opposed to other variable types. Weight is a relative value, and probability is then computed as weight of each divided by the sum of weights.

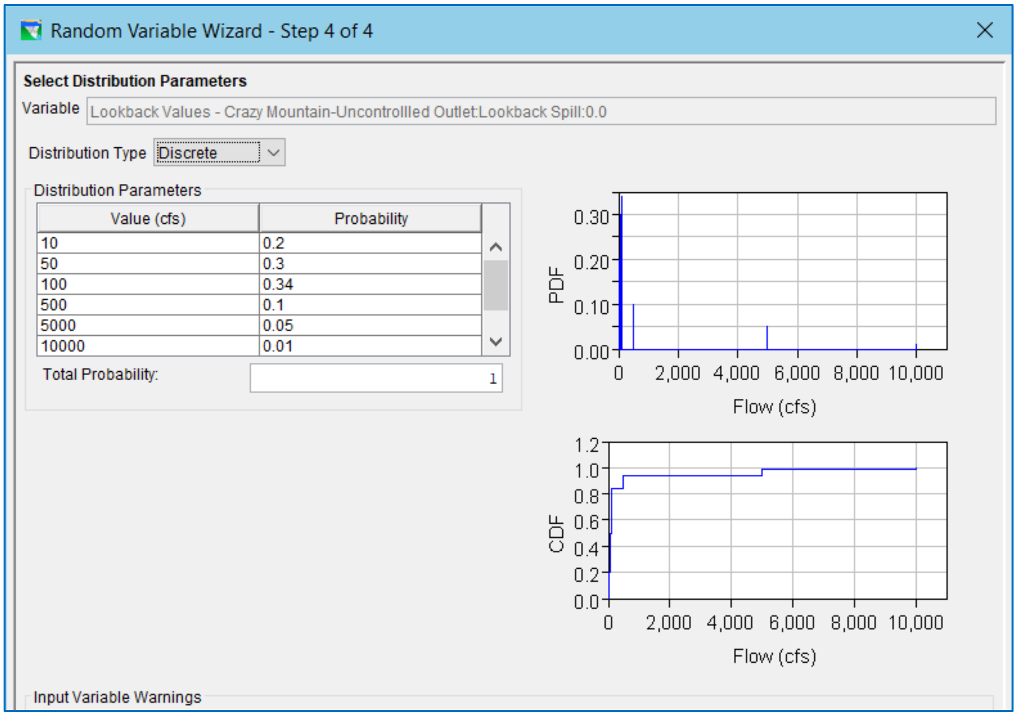
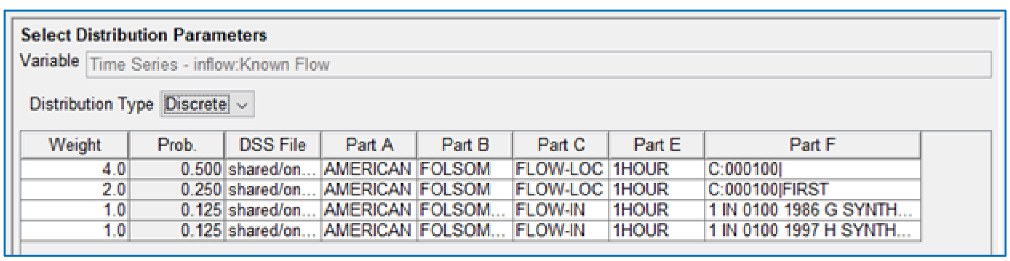


Figure H.19 Input Variable Wizard—Discrete Distribution



Weight	Prob.	DSS File	Part A	Part B	Part C	Part E	Part F
4.0	0.500	shared/on...	AMERICAN	FOLSOM	FLOW-LOC	1HOUR	C:000100|
2.0	0.250	shared/on...	AMERICAN	FOLSOM	FLOW-LOC	1HOUR	C:000100|FIRST
1.0	0.125	shared/on...	AMERICAN	FOLSOM...	FLOW-IN	1HOUR	1 IN 0100 1986 G SYNTH...
1.0	0.125	shared/on...	AMERICAN	FOLSOM...	FLOW-IN	1HOUR	1 IN 0100 1997 H SYNTH...

Figure H.20 Discrete Distribution for Input Time Series

Normal Distribution

A normal distribution (Figure H.21) is a continuous, two parameter, symmetrical probability distribution. The two parameters are **mean** and **standard deviation**.

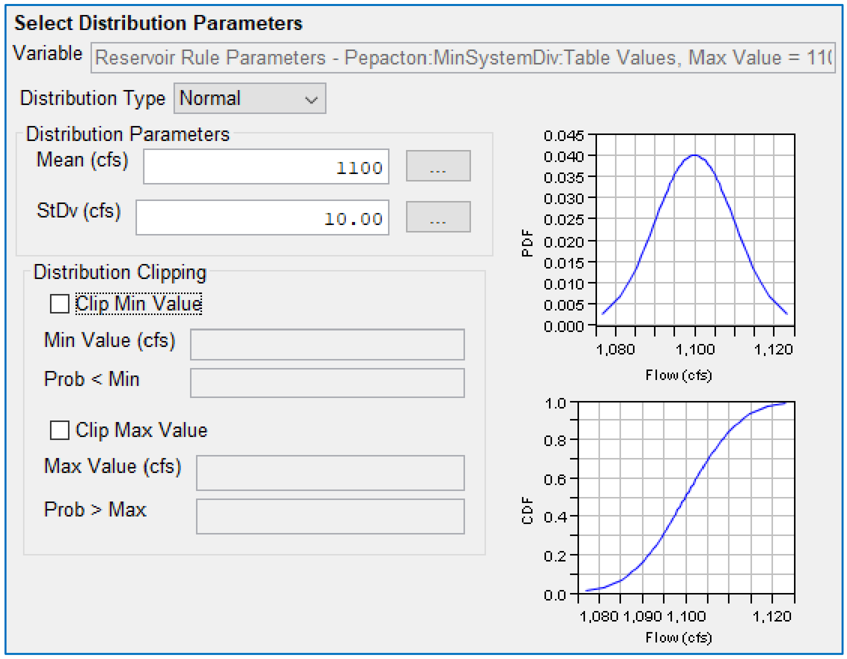


Figure H.21 Input Variable Wizard—Normal Distribution

LogNormal Distribution

A log-normal (or lognormal) distribution (Figure H.22) is a continuous probability distribution of a random variable whose (natural) logarithm is normally distributed. The parameters are the **mean of the logarithm** of the variable, and the **standard deviation of the logarithm** of the variable. It is always positively skewed, by a degree dependent on the ratio of mean to standard deviation.

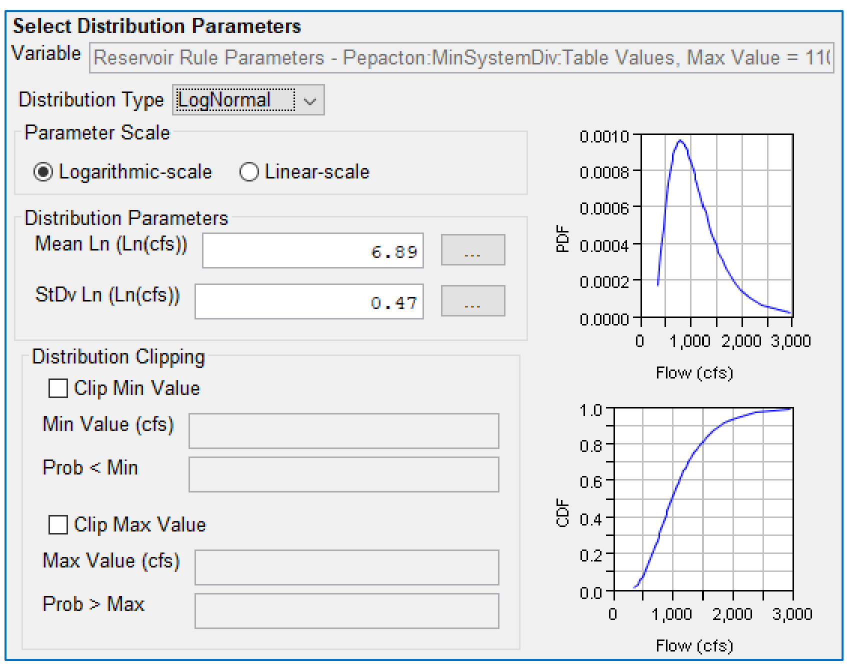


Figure H.22 Input Variable Wizard—LogNormal Distribution

Log10Normal Distribution

A log10-normal (or log10normal) distribution (Figure H.23) is a continuous probability distribution of a random variable whose base10 logarithm is normally distributed. The parameters are the **mean of the logarithm** of the variable, and the **standard deviation of the logarithm** of the variable. It is always positively skewed, by a degree dependent on the ratio of mean to standard deviation.

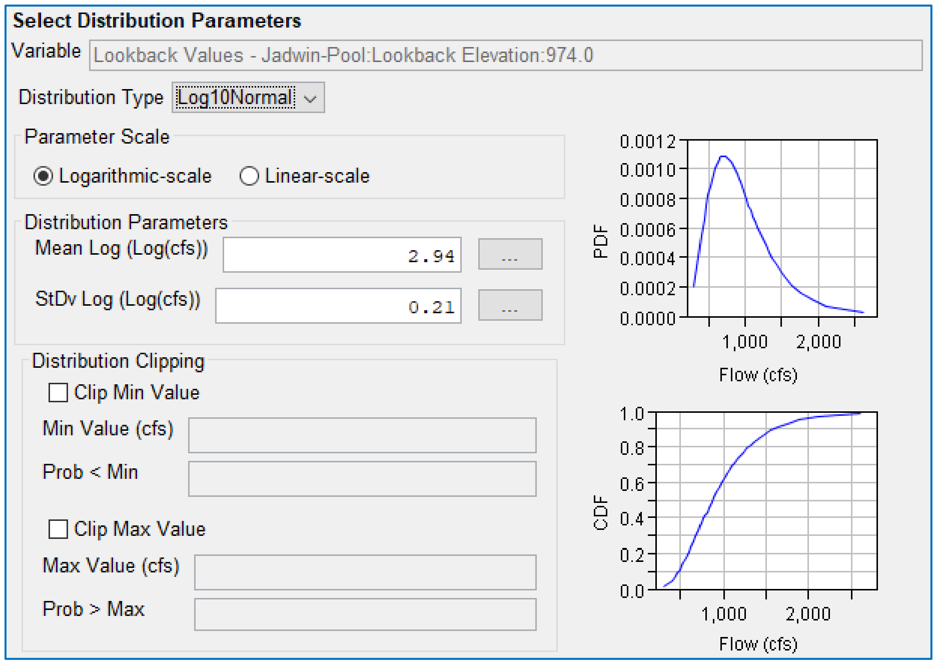


Figure H.23 Input Variable Wizard—Log10Normal Distribution

H.2.3 Output Variables

Output variables are those variables whose response to changes in the input variable(s), and resulting probability distributions, are of interest in the Monte Carlo analysis. For each output variable you select, you must choose what statistic (min, max, mean, or volume) of that variable you want to track. In addition to selecting the statistic, you can also mark the variable as an indicator of an "interesting" iteration using the **Flag** checkbox, or you can mark it as a determinant of Convergence using the **Convergence** checkbox.

Iterative HEC-ResSim Monte Carlo simulations can produce very large amounts of data, because every iteration of the run is capable of producing as much data as a standard HEC-ResSim simulation, and typically a Monte Carlo simulation will consist of hundreds to thousands of iterations. In order to minimize data storage, the Monte Carlo Plug alternative will not write detailed output for all iterations. The user can override this for certain iterations by using the *interesting* Flags—when the Flag Condition of the output variable is triggered, the iteration is deemed interesting enough to cause the full output of the iteration to be written.

Setting *convergence* criteria for an output variable is a way to determine the stopping point of a Monte Carlo Alternative. Similar to Interesting Flags, Convergence criteria are conditional expressions that are evaluated at the end of every iteration. Convergence is reached when the average value of the output variable (over all iterations) is known *accurately* and *confidently* enough to stop iterating.

At least one of the output variables you define should be setup for and chosen to define convergence for your Monte Carlo alternative.

To specify an Output Variable:

- Select the **Output Variables** sub-tab on the **Monte Carlo** tab of the **Alternative Editor** (Figure H.24).
- Click the **Add** button. This will open the **Output Variable Wizard** that will step you through the process of selecting a ResSim Output Variable and defining the variable's statistic and flag conditions (Figure H.25).

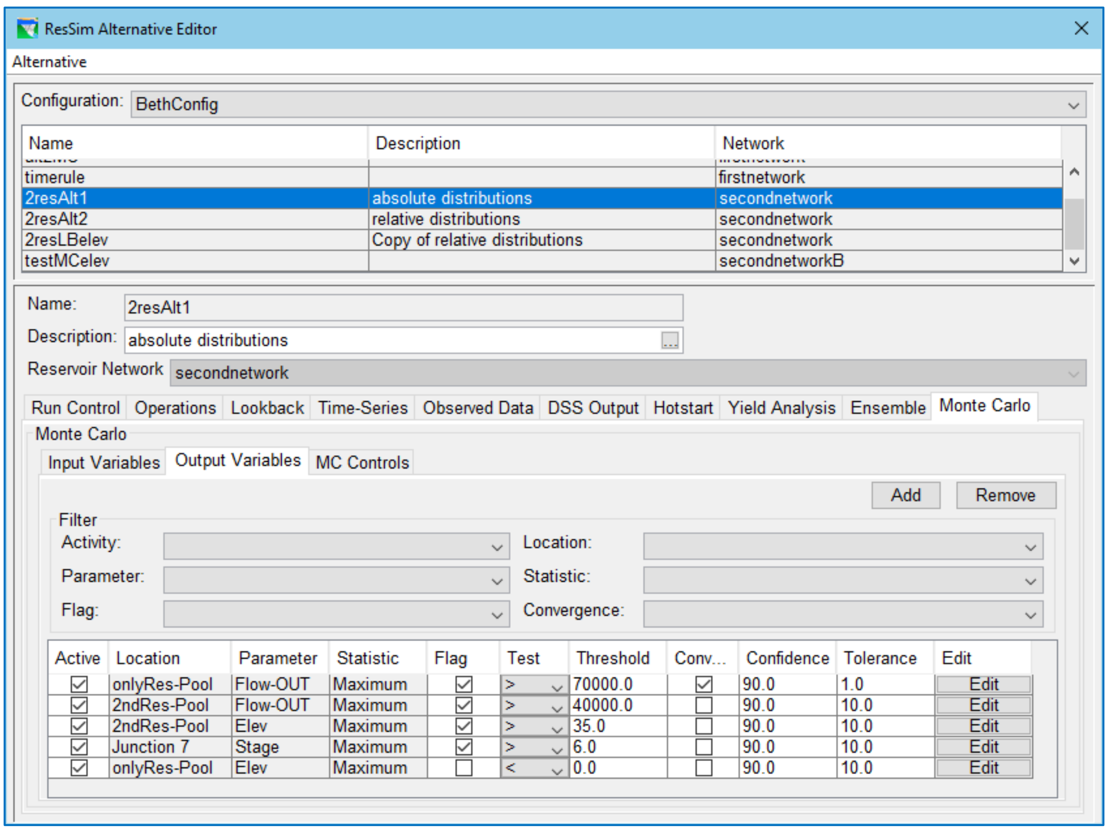


Figure H.24 Alternative Editor—Monte Carlo Tab—Output Variable Sub-Tab

Step 1 of 2—Select the Output Variable.

This first step of the Output Variable Wizard is a variable chooser (Figure H.25) much like the one found in ResSim's **Independent Variable Definition** editor (Appendix C, Section 2). Use one or two of the three filters available above the list of ResSim model variables to help you find the variable you need. The first filter lets you thin out the list by element name (location), the second filter lets you thin out the list by element type, and the third lets you thin out the list by parameter (or variable type).

Tip Since ResSim produces a large quantity of output, it is often easiest to filter the list first by element type or parameter. Either or both of these filters will usually shorten the list substantially, making it much easier to find the location and parameter you need.

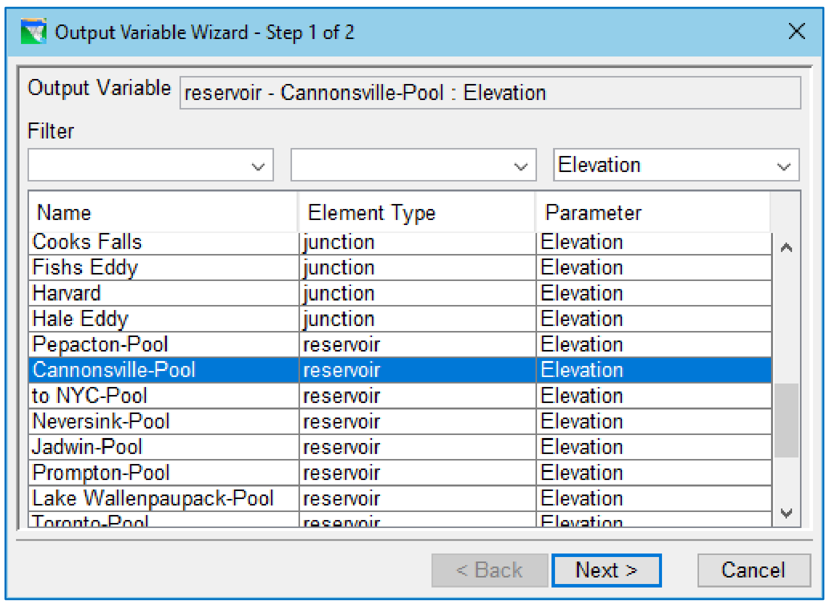


Figure H.25 Monte Carlo Output Variable Wizard—Step 1—Select Variable

Step 2 of 2—Specify the analysis criteria.

The Monte Carlo Output Variable analysis criteria has three parts, which are selected in the **Output Variable Wizard** step 2 (Figure H.26):

- **Statistic**—this criterion is a required selection for the convergence calculations. Your options are **Minimum**, **Maximum**, **Mean**, and, for some variables, **Volume**. ResSim will analyze the computed time series for the selected model variable to compute the statistic and store the value for the current iteration of the Monte Carlo compute.
- **Flag Condition**—this criterion is optional and off by default. It is intended to mark extreme iterations for closer review. To use, place a check in the **Flag** checkbox then select the **Test** (comparison operator) and **Threshold** value.

- **Convergence Condition**—this criterion is also optional and off by default but must be on for at least one output variable. To use the convergence of an output variable (i.e., lack of change in the random sample between iterations) as a means of stopping the Monte Carlo iterations, place a check in the **Convergence** checkbox and define the parameters for convergence: **Confidence** and **Tolerance**. Confidence is the percentile interval (e.g. a 90% confidence interval) upon which convergence is based, and Tolerance is the maximum percent change (accuracy) in the average of the output statistic necessary to consider the variable "converged."
- **Values of Interest** can be specified, and the output will include the percentages of the iterations that exceed those values.

Once you have completed entry of the analysis criteria for your output variable, click **Finish** to close the **Output Variable Wizard**. A new row will be added to the table of variables on the **Output Variables** sub-tab, representing your new output variable.

Since a large model could require a significant number of output variables, the table of input variables is preceded by a set of **Filters** that you can use to thin out the list so that you can find a specific variable of interest.

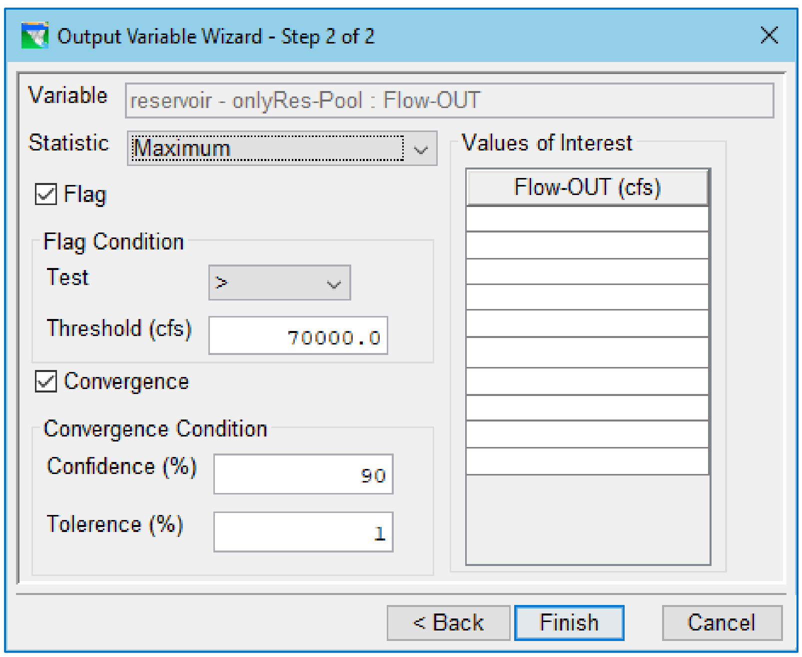


Figure H.26 Monte Carlo Output Variable Wizard—Step 2—Define Criteria

H.2.4 Monte Carlo Controls

A number of options are available to give you control over the simulation of your Monte Carlo Alternative. These options are located on the **MC Controls** sub-tab of the **Monte Carlo** tab in the ResSim Alternative Editor (Figure H.27).

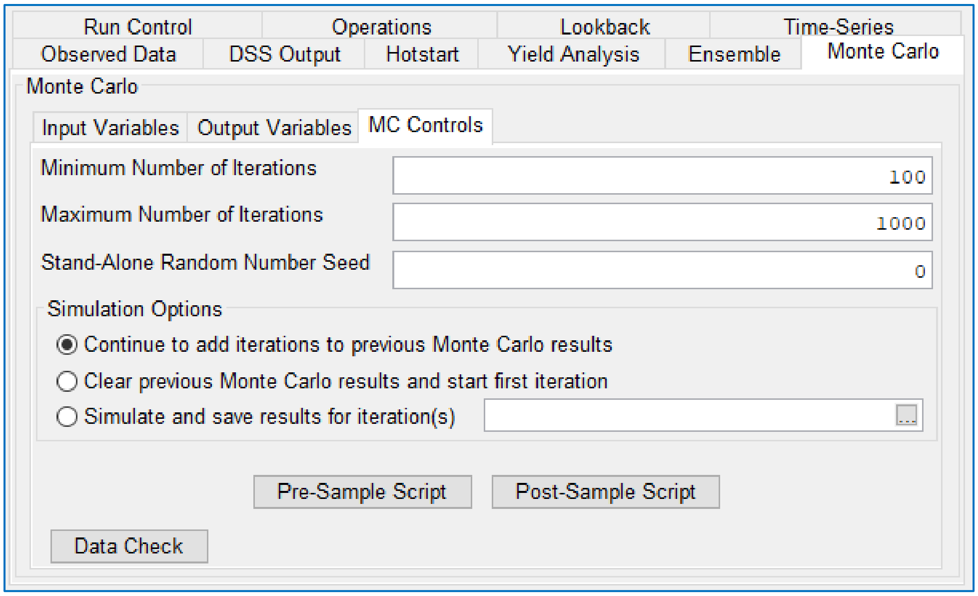


Figure H.27 Alternative Editor—Monte Carlo Tab—MC Controls Sub-Tab

The first two options control the number of iterations the Monte Carlo alternative may perform

- Minimum Number of Iterations—the Monte Carlo alternative will perform at least the minimum number of iterations entered in this field, even if convergence criteria are met before the minimum iterations have been performed.
- Maximum Number of Iterations—If convergence criteria have been met before the maximum number of iterations have been performed, the run will stop; however, if convergence criteria have not been met, the run will stop when the maximum number of iterations have been performed. The value entered in this field must be greater than or equal to the minimum number of iterations.
- Stand-Alone Random Seed—when the Monte Carlo alternative is run within a ResSim simulation (in standalone ResSim, as opposed to ResSim within HEC-WAT), the random number seed is used by ResSim's random number generator to initialize the random numbers used for each input variable in each iteration. The same seed will always produce the same random values. This seed can be changed if you want to produce different random values. This field is not used when the Monte Carlo alternative is used in an HEC-WAT stochastic Flood Risk Analysis run.
- Simulation Options—three simulation options control how you start and stop Monte Carlo simulations:
  - Continue to add iterations to previous Monte Carlo results. This option may be used when you want to run the Monte Carlo alternative in batches and check the results between each batch.
  - Clear previous Monte Carlo results and start first iteration. This option should be used if you make changes to the Monte Carlo alternative in any way and want to restart the run.

  - **Simulate and save results for iteration(s)**... This option allows you to identify specific iterations that you want to rerun and store the complete set of output that a standard ResSim alternative would save. This option is used after computing a Monte Carlo alternative and reviewing the limited output stored by the run. This option should be used if something in the output calls for a closer look at the specific iteration(s), such as triggering a flag.
- **Pre-Sample Script**—this button opens a standard script editor with an initialization, main, and cleanup tab. Code in the initialization tab will only be executed at the very start of a Monte Carlo compute right before the first iteration, while code in the cleanup tab will only be executed at the very end of a Monte Carlo compute after the vary last iteration. For the Pre-Sample script, code in the main tab will be executed every Monte Carlo iteration right before the random variable sampling.
- **Post-Sample Script**—this button opens a standard script editor with an initialization, main, and cleanup tab. Code in the initialization tab will only be executed at the very start of a Monte Carlo compute right before the first iteration, while code in the cleanup tab will only be executed at the very end of a Monte Carlo compute after the vary last iteration. For the Post-Sample script, code in the main tab will be executed every Monte Carlo iteration right after the random variable sampling.
- **Data Check—**this button reviews the definition of the Monte Carlo alternative to be sure the necessary data is adequate for a successful execution of the run.

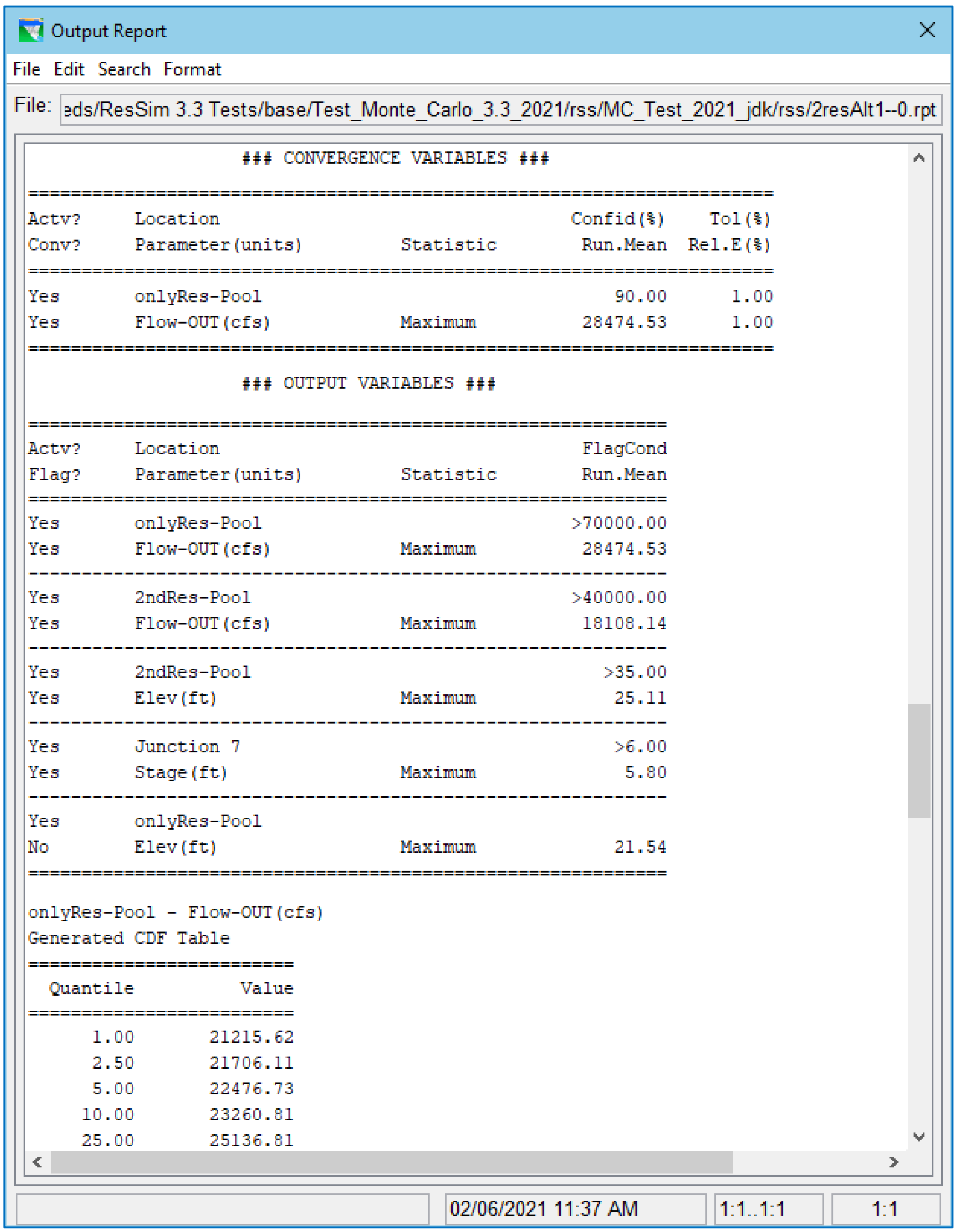
H.3 Analysis Tools

A number of features have been included in ResSim to assist in analyzing the results from a Monte Carlo alternative. These features can be found in the **Reports** menu under **Monte Carlo**. They include:

- **Output Report**—a text report generated by the Monte Carlo run that summarizes key aspects of the run including the number of iterations, the status of convergence, statistics on the input and output variables and a table showing the non-exceedance probability of the "values of interest" for each output variable (Figure H.28).
- **Histogram Display**—this "Report" includes a graphic showing a histogram of the selected input or output variable, sampling statistics, and parameter values (Figure H.29). The displayed information may include the following (depending on which variable is selected):
  - **Number of Realizations** (iterations),
  - **Running Average**,
  - Distribution Parameters (e.g., mean, standard deviation) specified for the variable's distribution,
  - **Clipping** limits.
  - Convergence Parameters (confidence, relative tolerance, relative error),
  - Flag Test Parameters

- **Convergence Plot**—this "Report" is a plot of the Output Variable's statistic, the running average and error bounds on the statistic, and the convergence criteria on the variable (Figure H.30).

In addition to the predefined Monte Carlo "reports", all the output generated by the run is stored in the **simulation.dss** file, so you can look for and find anything else you might need to analyze the results.


Output Report
File Edit Search Format
File: eds/ResSim 3.3 Tests/base/Test_Monte_Carlo_3.3_2021/rss/MC_Test_2021_jdk/rss/2resAlt1--0.rpt

```
                    ### CONVERGENCE VARIABLES ###

======================================================================
Actv?     Location                                  Confid(%)   Tol(%)
Conv?     Parameter(units)         Statistic         Run.Mean  Rel.E(%)
======================================================================
Yes       onlyRes-Pool                                  90.00     1.00
Yes       Flow-OUT(cfs)            Maximum           28474.53     1.00
======================================================================

                    ### OUTPUT VARIABLES ###

============================================================
Actv?     Location                                  FlagCond
Flag?     Parameter(units)         Statistic        Run.Mean
============================================================
Yes       onlyRes-Pool                             >70000.00
Yes       Flow-OUT(cfs)            Maximum          28474.53
------------------------------------------------------------
Yes       2ndRes-Pool                              >40000.00
Yes       Flow-OUT(cfs)            Maximum          18108.14
------------------------------------------------------------
Yes       2ndRes-Pool                                 >35.00
Yes       Elev(ft)                 Maximum             25.11
------------------------------------------------------------
Yes       Junction 7                                   >6.00
Yes       Stage(ft)                Maximum              5.80
------------------------------------------------------------
Yes       onlyRes-Pool
No        Elev(ft)                 Maximum             21.54
============================================================

onlyRes-Pool - Flow-OUT(cfs)
Generated CDF Table
=========================
  Quantile         Value
=========================
      1.00      21215.62
      2.50      21706.11
      5.00      22476.73
     10.00      23260.81
     25.00      25136.81
```

02/06/2021 11:37 AM 1:1..1:1 1:1

Figure H.28 Monte Carlo—Sample Output Report

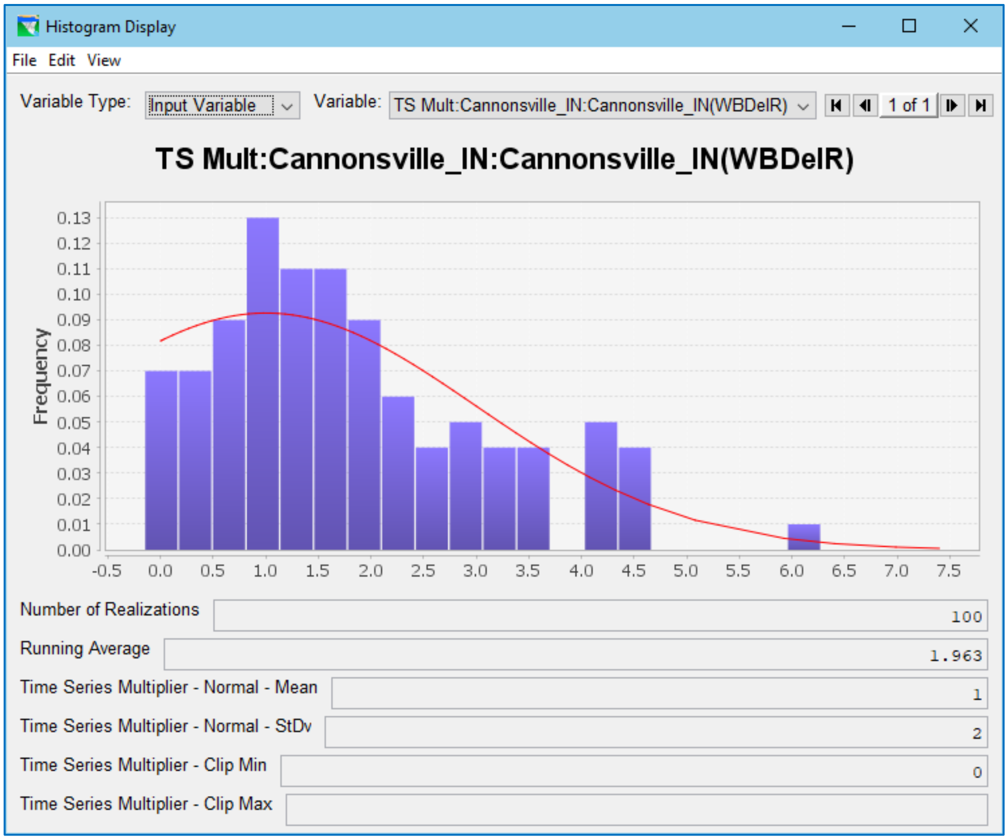


Figure H.29 Monte Carlo—Sample Histogram Display

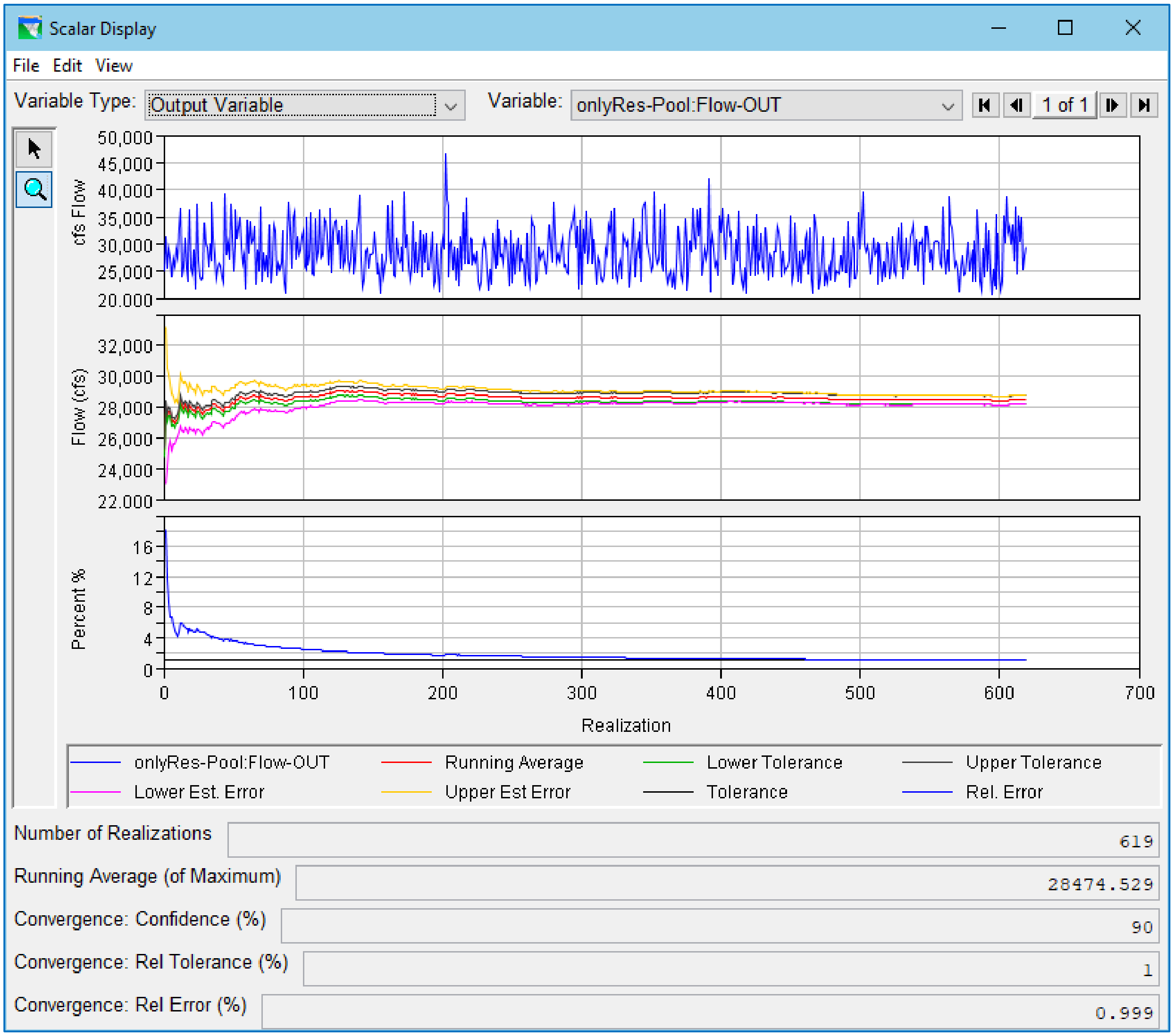


Figure H.30 Monte Carlo—Scalar Display showing Convergence

Note that the reports shown in this appendix do not contain meaningful results. They are merely screenshots to help familiarize the reader with the appearance of the features in the interface. Likewise, many of the figures depicting editors do not contain real data.

Appendix I
Operation Support Interface

The **Operation Support Interface (OSI)** is a user-configurable data visualization and editing tool designed to enable you to create your own custom interface to perform one or more of the following activities:

1) Review results
2) Develop an operation schedule using release (or elevation) overrides
3) Compute local inflows between gaged flow locations and estimate local inflows where gaged data is unavailable.

These activities revolve around the type of variables you include on your OSI tabs. Since these activities are quite different, they are usually setup on different tabs (or sets of tabs) of the OSI.

To understand those two statements, you'll need to learn some OSI basics. In the first three sections of this Appendix, we'll provide an explanation of OSI variables, the organization of the OSI user interface, and the basics of OSI setup. In the subsequent sections, we'll describe how to setup and use the OSI to perform each of the activities listed above. Examples will be used to illustrate the process and to help explain why you'd want to use the OSI in the first place.

I.1 OSI Variables

(Model) **Variables** are the fundamental data objects of the OSI; they are the time series datasets input to or computed by your model that the OSI makes available to you for viewing and editing. The variables which the OSI allows you to view and edit are grouped into four categories:

Local Inflows—These variables correspond to the "Known Flow" time series entries you see in the **Time-Series** tab of the ResSim **Alternative Editor** (Chapter 14). You created one of these variables each time you created an entry in the table on the **Local Flow** tab of the **Junction Editor** for the various junctions in your reservoir network. **Local Inflows** are editable (from the OSI) over the entire simulation time window, from the start of the Lookback Period to the end of the Simulation (or Forecast) Period.

Reservoir Releases—These variables correspond to the **Release Override** time series for each *controllable* release element of the reservoir outlet hierarchy of each reservoir; they can be viewed/edited from the **Release** tab of the ResSim **Overrides** editor (Chapter 15). **Reservoir Releases** are only editable in the Simulation (or Forecast) Period (i.e., the *lookback* releases are considered "observed" and are not editable).

Elevation Targets—These variables correspond to the **Elevation Target Override** time series for each reservoir; they can be viewed/edited from the **Elevation Target** tab of the ResSim **Overrides** Editor (Chapter 15). Like **Reservoir**

Releases, **Elevation Targets** are only editable in the Simulation (or Forecast) Period.

Computed Parameters—These variables correspond to the set of computed (output) time series produced by all the elements in your reservoir network. **Computed Parameters** are not editable from the OSI.

I.2 Exploring the OSI User Interface

The OSI can only be opened and configured from the ResSim **Simulation** module. Each OSI configuration is associated with an *active* alternative, so you must open a simulation and identify an active alternative in the ResSim **Simulation** module.

When you first open the OSI, you will be presented with a fairly large *blank* window. Don't despair, it's not actually blank; it has a **Title Bar** and a **Menu Bar** on it above a large white panel (Figure I.1). The Menu Bar contains everything you will need to configure the OSI.

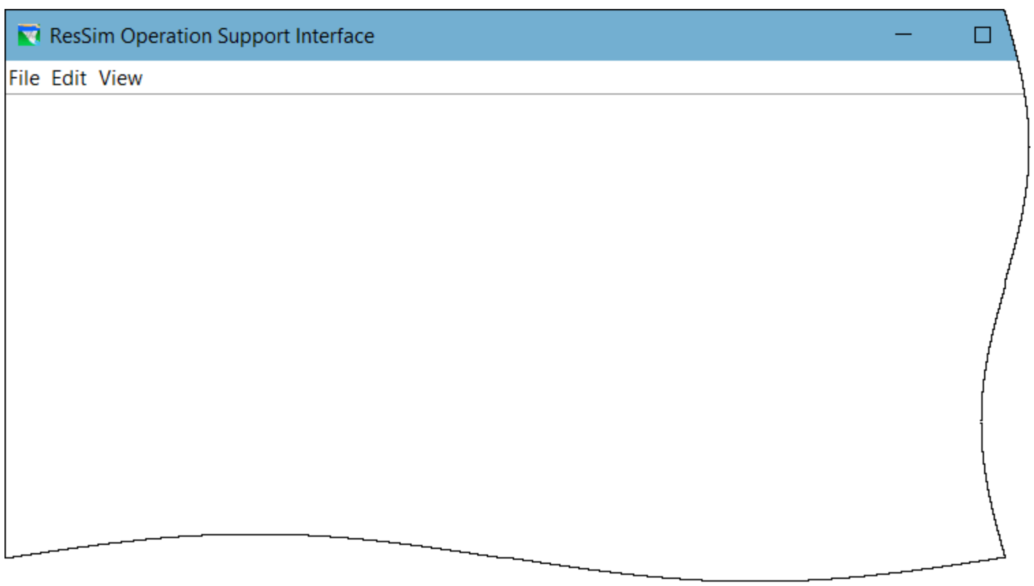


Figure I.1 Operation Support Interface—Unconfigured (Blank)

Configuring the OSI is fairly easy. Instructions for getting started are in Section I.3 below, but they begin with adding a tab.

I.2.1 Tabs

The OSI user interface was designed to allow you to group *Variables* for viewing and editing in *Tabs*. Once you add your first tab to the OSI, you will see what the OSI's user interface is going to look like (Figure I.2).

The top portion of each tab is a **Plot Panel** (and/or **Graphical Editor**). A **Table Panel** takes up most of the bottom portion of the tab with a stack of **Action Buttons** taking up the rest. As you add **Variables** to the tab, they will appear as *columns* in the **Table**

Panel. A barely-noticeable **Resize** bar separates the top and bottom portions of the tab; you can "grab" this bar with your mouse and drag it up or down to resize the two panels.

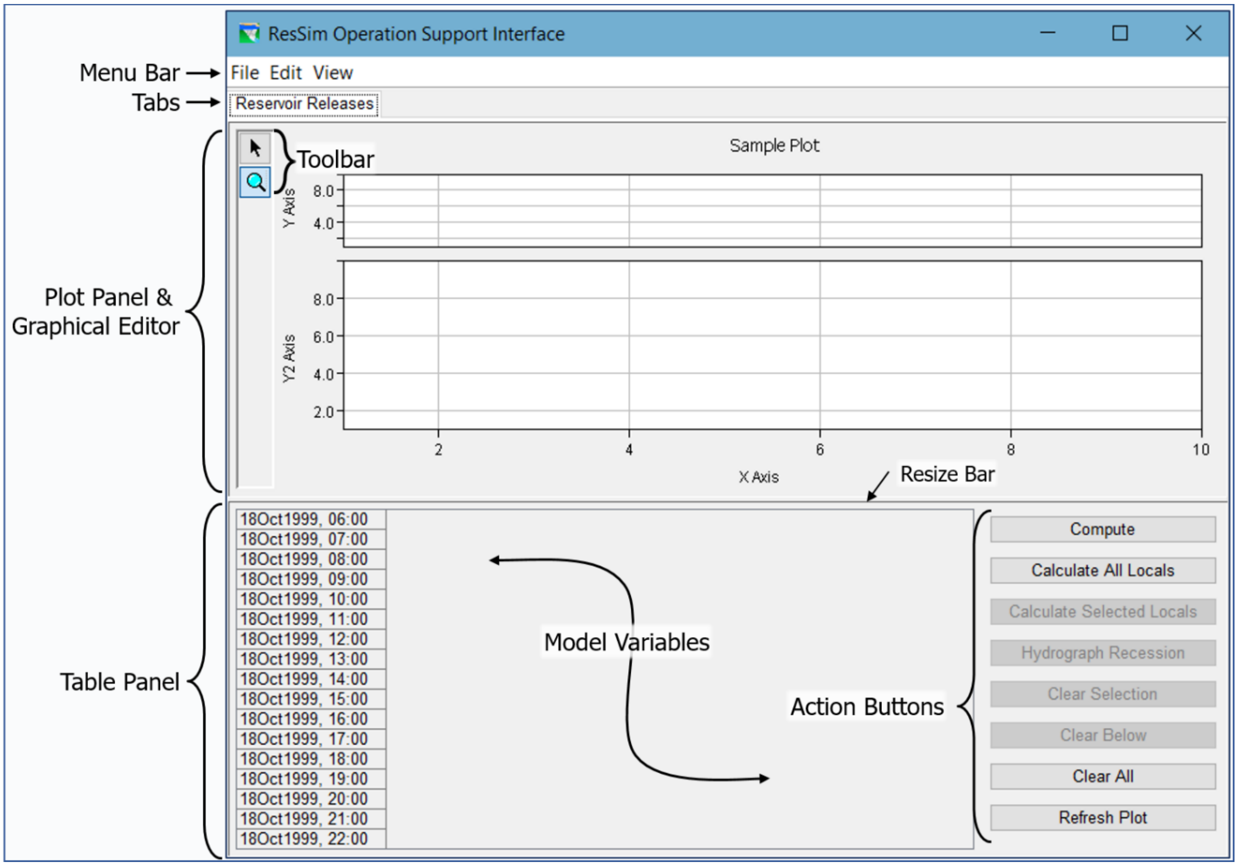


Figure I.2 Operation Support Interface with a New Tab Added—Annotated

The **Plot Panel** and the **Action Buttons** are context-sensitive. That means that how they appear and the functionality they provide are related to the *active* variable's type or *category*. To make a variable *active*, you only need to click on the data in that variable's column in the **Table Panel** (*clicking* in the header won't do it).

For example, Figure I.3 shows the **Plot Panel** displaying time-series data for the selected (active) variable as well as another time-series dataset that was selected to be plotted with it. The plot also shows a green cross-hatched vertical bar indicating the range of the data currently displayed in the table and a blue vertical line indicating the point in time of the selected cell in the table.

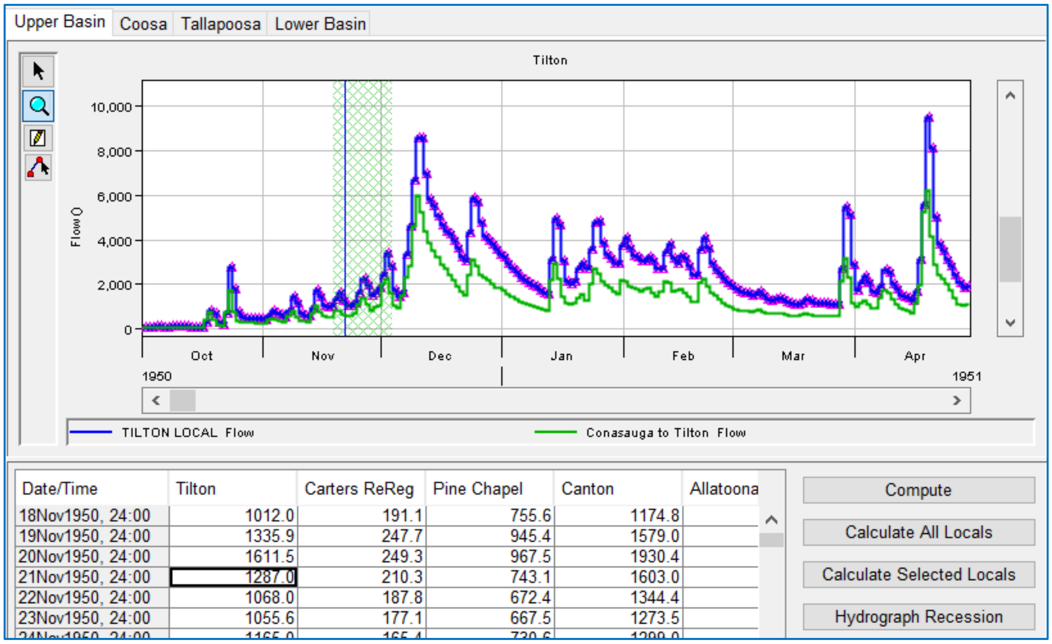


Figure I.3 OSI Tab—Plot Panel Showing Active Dataset

I.2.2 Menu Bar

The Menu Bar in the OSI is important to the setup and configuration of the OSI. After getting the OSI configured the way you like it, you'll use the menus less frequently.

I.2.2.1 File Menu

The **File** menu (Figure I.4) is straight-forward; you only have **Save** and **Close** to choose from.

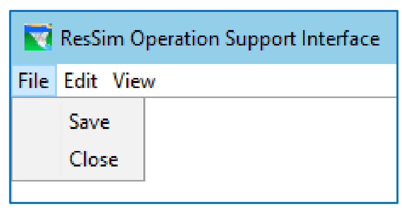


Figure I.4 OSI—File Menu

Save has two different things to save:

- The OSI configuration—As you make changes to the configuration of the OSI, you should be sure to save it frequently.
- The Variable data. If you make changes to the time-series data of one or more of the variables, it will be saved each time you click **Compute**. If you wish to save your changes without computing, select **Save** from the **File** menu before closing the OSI.

Close—closes the OSI. Closing can also be accomplished by clicking the **X** in the upper right corner of the window.

You might be wondering why you are NOT given the opportunity to name your OSI configuration nor are a Rename or SaveAs options included in the File menu...
It's because the OSI configuration is stored as part of the alternative definition. That fact results in a few hidden truths:

- You can only create one OSI configuration per alternative.
- You must configure the OSI for each alternative independently.
- You can configure the OSI differently for different alternatives.
- And, unfortunately, you cannot easily copy an OSI configuration from one alternative to another. Look for that option in a later version of ResSim. However, if you configure the OSI in an existing Alternative, then use SaveAs in the Alternative Editor to create a new alternative as a copy of the existing one, the OSI configuration *will* get copied (along with everything else that defines the alternative) to the new alternative.

I.2.2.2 Edit Menu

The **Edit** menu (Figure I.5) contains most of the features you will need to configure the OSI. Since the OSI is organized around Tabs and Variables, the Edit menu gives you the options you need to configure them.

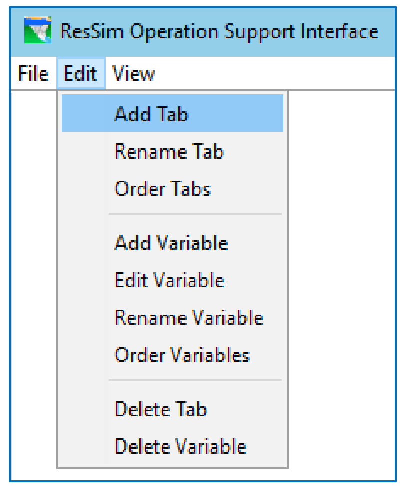


Figure I.5 OSI—Edit Menu

Add Tab—adds a tab to the OSI (after prompting you to enter a name and description). As illustrated in Figure I.2, each tab has a **Plot Panel** and a **Table Panel** for viewing and editing OSI Variables. Tabs are always appended to the end (right) of the "list" of tabs.

Rename Tab—allows you to rename the current (active) tab.

Order Tabs—opens the **List Order Editor** with a list of the existing OSI tabs to enable you to reorder the list. The first entry in the list is the left-most tab in the OSI. Select a tab name in the list then click the Up (^) or Down (v) buttons to move the selected tab left or right with respect to its neighbors.

Add Variable—adds a variable (column) to the Table Panel of the current tab. Like tabs, variables are always appended to the end of the "list" of variables. When you create a variable in the OSI, you will be asked to give it a name. The name you give the variable will appear in the header of this variable's column in the Table Panel and in the title of the Plot Panel when the variable is the current (or active) variable.

Edit Variable—opens the **Operations Support Model Variable Editor** for configuring the variable and its plot(s).

Rename Variable—allows you to rename the current (active) model variable.

Order Variables—opens the **List Order Editor** with a list of the existing OSI Model Variables on the current tab to enable you to reorder the list. The first entry in the list is the left-most variable in the Table Panel of the current tab. Select a variable name in the list then click the Up (^) or Down (v) buttons to move the selected variable left or right with respect to its neighbors.

Delete Tab—deletes the current tab from the OSI, variables and all. This does not delete the time series associated with the OSI variables from the model, it just deletes the tab configuration from the OSI configuration.

Delete Variable—deletes the current variable from the current tab. This does not delete the time series associated with the OSI variable from the model, it just deletes the OSI variable from the OSI configuration.

I.2.2.3 View Menu

The View menu (Figure I.6) provides options for how the Variable columns in the Table Panel will display. These options include:

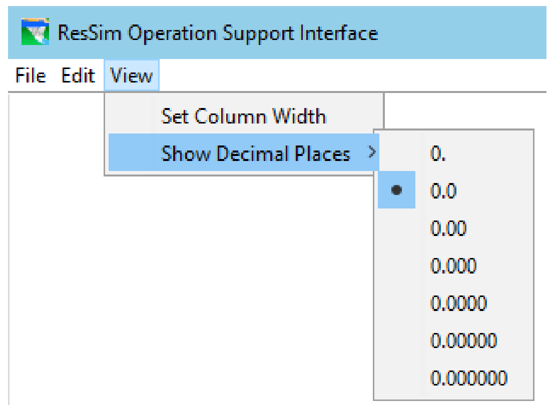


Figure I.6 OSI—View Menu

Set Column Width—Use this option to specify the width of every column in the Table Panel on every tab. The units are pixels (screen units).

Show Decimal Places—Use this option to specify how many digits after the decimal to display. Your options range from 0 to 6 digits after the decimal. Like **Set Column Width**, **Show Decimal Places** is also a global setting for this OSI configuration; it applies to all columns on all tabs of this OSI.

I.2.3 OSI Plot Panel and Graphical Editor

The Plot Panel displays a time series plot of the currently selected Variable. If that Variable is editable, the Plot Panel's Toolbar will expand to include the Graphical Editor tools for editing the time series from the plot. These tools are:

 **Single Point Edit—**to use, select this tool, then move the cursor into the plot to a time and value that you want to change the variable to. Double-click your mouse button at that position. A new point will be drawn at that position. You can also use this tool to drag an existing point up or down in the plot to a new position and therefore a new value.

 **Multi-Point Edit—**to use, select this tool, move the cursor into the plot, *click* the cursor at each desired location of a series of new data points. A line will appear connecting the data points you drew. *Right-click* after the final data point has been placed to stop drawing data points.

Changes made through with the **Graphical Editor** tools are immediately shown in the Table Panel and vice versa.

Additional time series may be displayed in the **Plot Panel** to provide supplemental context. These time series are added through the OSI Variable Editor.

Since the plot can show a much longer portion of the data than a column in the **Table Panel** can, the plot uses a cross-hatched background to illustrate the portion of the data is that is currently shown in the table. If an editable cell in the table is selected, the time corresponding to that cell is shown as a vertical line in the Graphical Editor. If the Table Panel can show the entire time series (without a scrollbar), then the green cross-hatched background is not drawn.

I.2.4 OSI Table Panel

The Table Panel shows the data for each OSI Variable as a column in the table. Note - the additional time series that you can add to plot with a given variable are NOT shown in the Table Panel.

A **Date/Time** column is included in the table panel to indicate which timestep of the current simulation time window is associated with each data value of each Variable displayed in the Table Panel.

The columns to the right of the **Date/Time** column show the time series for each Variable you added to the tab. The cells holding data values from the lookback period of the simulation window are shown with a medium-grey background. A grey background in an editable table typically means that the values are not editable; however, even though lookback values for Local Inflows *are* editable in the OSI in the lookback period, the cells still display as grey. Computed Parameter variables are not editable for any part of the simulation window and they display with a light grey background for the whole simulation time window.

To change a value in a cell of an editable variable, just click in the cell and type in the new value. When you move to another cell, the cell you just changed will appear with Green text to indicate that you have entered an override value. You can also use the table **Fill** options from the table's context menu to edit a range of values.

I.2.5 Action Buttons

The Action Buttons (Figure I.7), located to the right of the Table Panel on each tab of the OSI, include the following:

Compute—Computes the current alternative associated with the OSI. Can be used to compute results after changes are made using the Table or Graphical Editors.

Calculate All Locals—Computes the values for all Local Flow Variables across all tabs for the entire simulation period when observed flows are available. See Section I.6 below for a description of how to configure your alternative and the OSI to compute Local Flow Variables.

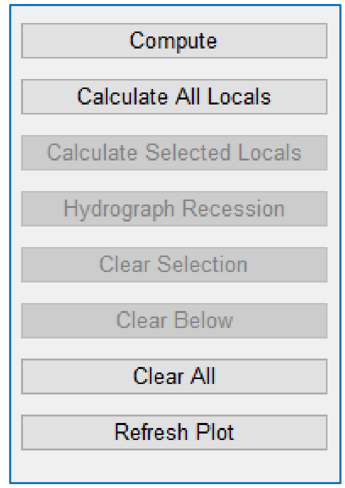


Figure I.7 OSI—Action Buttons

This compute option runs a special ResSim simulation type to override computed Junction outflow values with observed time series data whenever the observed data is available. When the simulation is complete, all local flow time series associated with a Local Flow OSI Variable are adjusted to make up the difference between total inflow to the Junction and the observed value at the junction outflow. Note: for this calculation to work, observed flow data must be provided for the Junction outflows. Also Note: the Known Flow time series in the ResSim simulation.dss file will be overwritten with the calculated local flows. A second, normal ResSim compute is then launched to use the computed local inflow data and thus refresh all output data.

Calculate Selected Locals—Computes the values for the selected Local Flow Model Variables in the current tab and only for the time steps before and including the currently selected row in the table.

Hydrograph Recession—Computes the recession hydrograph for the selected Local Flow Model Variables from the selected time to the end of the simulation period. If the only cell selected is in the Date/Time column, then the recession hydrograph will be calculated for *all* Local Flow Model Variables on the current tab. Note: Known Flow time series in the ResSim simulation.dss file will be overwritten with the calculated results. Recession parameters are defined independently for each Local Flow Model Variable through the OSI Model Variable Editor.

Clear Selection—Sets local flows to zero and removes release and elevation target overrides from the selected model variables for the selected time range.

Clear Below—Sets local flows to zero and removes release and elevation target overrides for the selected model variables *beginning with the first time step of the selected time range and ending at the end of the simulation time window*.

Clear All—Sets local flows to zero and removes release and elevation target overrides for the selected model variables for the whole simulation time window. For local flows, this includes the lookback period.

Refresh Plot—Changes made to the configuration of a variable will not show in the Plot Panel until you take an action that forces the plot to refresh. Clicking in the column of another variable then clicking back in the one you reconfigured is one such action; clicking the **Refresh Plot** button is another.

I.3 Basic OSI Setup

Before you can setup an OSI for an alternative, you must have a simulation in which the alternative has successfully computed. Then, after opening the OSI, the process for configuring it involves two basic steps—adding a tab and adding one or more *variables* to the tab.

> **Tip** CWMS/RTS—You can also configure (and use) the OSI from a CAVI Forecast. Just click on the ResSim alternative of a computed forecast run, then click the Operations Support action button.

Compute the Alternative

- **Open** (or create) a simulation in the **Simulation** module of ResSim.
- Select an alternative for which you want to configure the OSI and make it active. If the alternative has not been computed, compute it.

Open the OSI

- Select **Operations Support Interface** from the bottom of the **Simulation** menu. The OSI will open as illustrated in Figure I.1

Add a Tab

- Select **Add Tab** from the **Edit** menu. A name dialog will open (Figure I.8). Enter a name (and an optional description) and *click* **OK**.
- The newly created tab will appear directly below the menu bar, along with an empty *Sample Plot* and Table Panel as illustrated in Figure I.2

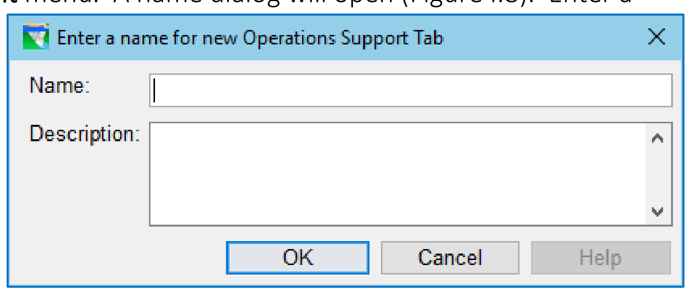


Figure I.8 OSI New Tab Name Dialog

> You can add additional tabs as needed; the only limit on the number of tabs you can add is the number you can keep track of—so don't go overboard. And, remember the other tab management functions available on the Edit menu:
>
> Rename—you can rename a tab by making it active (by clicking on it), then selecting Rename Tab from the Edit menu.
>
> Rearrange—you also rearrange your tabs. Select Order Tabs from the Edit menu, then use the List Order Editor to re-order them.
>
> Remove—you can delete a tab from the OSI by making it active, then selecting Delete Tab from the Edit menu. You do not need to delete the variables from a tab before deleting the tab, but be careful, there's no UNDO.

Add a Variable

- Select **Add Variable** from the **Edit** menu.
- A name dialog will open; enter a **Name** and an optional **Description** and *click* **OK**.
- The OSI **Model Variable Editor** will open (Figure I.9)
- Select the **Element Type**. Your choices include:
  - Junction
  - Reservoir
  - Reach
  - Diversion
- Select the **Element**. These are the elements you created in the reservoir network that the active alternative is based on.
- Select the Variable **Type.** The Variable **Types** or categories were described in *Section I.1, OSI Variables*. The list of available Variable **Types** is dependent on your selection of **Element Type**.
- Table I.1 shows the list of available Variable **Types** per **Element Type** and their **Additional Parameters.**
- After you have configured your OSI Variable (made all your selections in the *upper portion* of the **OSI Model Variable Editor**), click **Apply** to save your variable settings but keep the editor open.

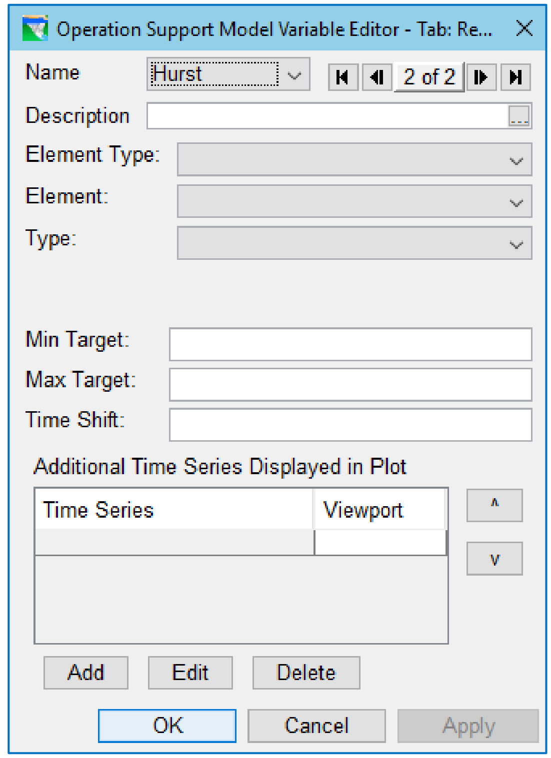


Figure I.9 Variable Editor Dialog Box

*Depending on your **Element** and Variable **Type** selection(s), additional parameters may appear in the editor. These parameters affect the behavior or computation of the variable.*

Table I.1 OSI Variable Types and their Additional Parameters

Element Type	Variable Type	Additional Parameters
Junction	Local Inflow	Local Flow
		Recession Method
		Recession Constant
		Max Recession Period (days)
	Computed Parameter	Computed Time Series
Reservoir	Reservoir Release	Reservoir Release From
	Elevation Target	
	Computed Parameter	Computed Time Series
Reach	Computed Parameter	Computed Time Series
Diversion	Computed Parameter	Computed Time Series

Configure the Plot Panel

The *lower portion* of the **OSI Model Variable Editor** (Figure I.10) provides options for configuring the **Plot Panel** of the OSI when displaying your variable. These options include:

Min Target—if you want a marker line plotted with your Variable to indicate a minimum target, enter that value in this field.

Max Target—similarly, if you want a marker line plotted with your Variable to indicate a maximum target, enter that value in this field.

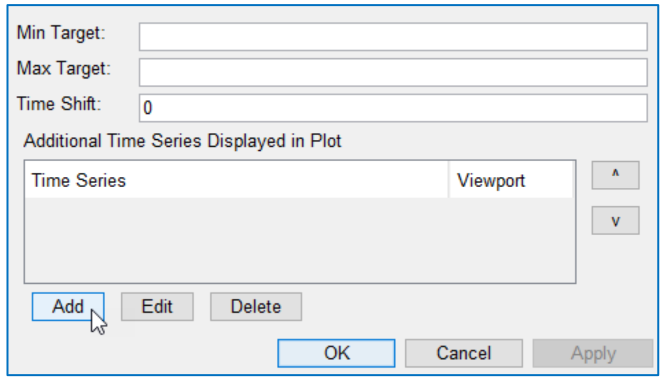


Figure I.10 OSI Variable Editor—Plot Panel Configuration Options

Time Shift—if you want your variables plotted with a time offset (shift), enter the number of hours you want the data shifted by in the plot. This may be useful when you plot additional time series with your variable—such as a downstream flow.

Additional Time Series Displayed in Plot—this table shows the list of time-series datasets that will display with the OSI variable in the **Plot Panel**. Use the **Add**, **Edit**, and **Delete** buttons below the table to manage the entries in the Additional Time Series… list. For example, to **Add** one or more ResSim output time series to be plotted with your variable in the plot panel:

- *Click* the **Add** button. This will open the **Independent Variable Definition** dialog (Figure I.11). (If the **Independent Variable Definition** dialog doesn't appear, click **Apply**, then click **Add** again.)

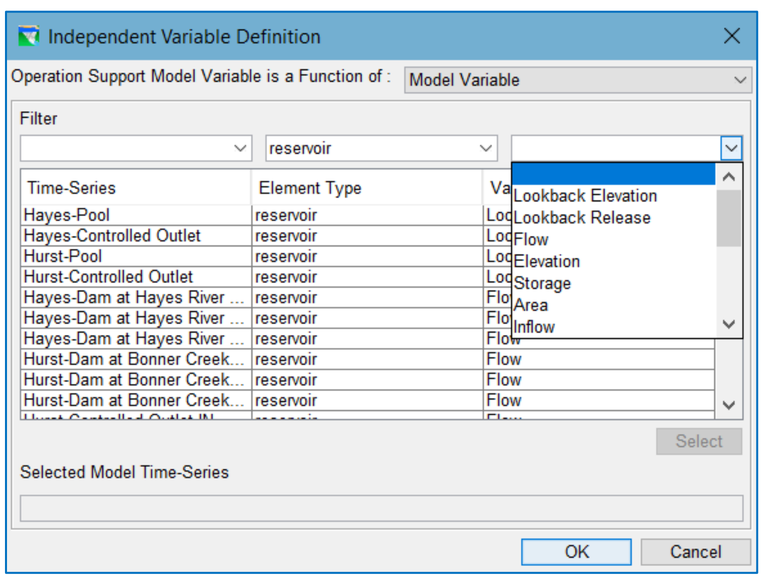


Figure I.11 Independent Variable Definition Dialog

- Use the **Independent Variable Definition** dialog to select a variable to include with the plot of the OSI variable. Detailed instructions for using the **Independent Variable Definition** dialog are available in Appendix C.

If you added more than one time series to **Additional Time Series…** table, you can change their display order by highlighting a time series in the table and using the arrows buttons to move the selected entry higher or lower in the list (Figure I.12).

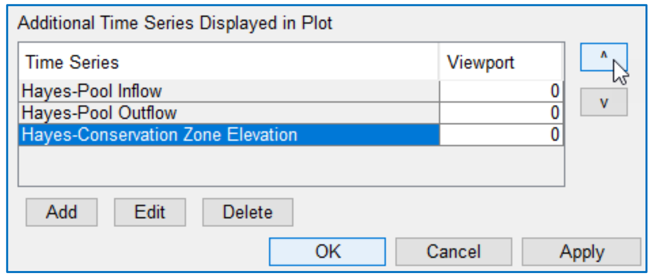


Figure I.12 OSI Variable Editor—Rearranging Display Order of Additional Time Series

When you have finished configuring your OSI variable, *click* **OK** to accept your settings can close the **OSI Variable Editor**.

Figure I.13 shows the plot panel of the OSI tab for the variable configured to use the additional time series shown in Figure I.12. Since the parameter types of one or more of the additional time series had a different parameter than that of the OSI variable, an additional plot viewport was automatically added to display the different parameter type. Note—the vertical axis in each viewport starts at zero by default. This may be inappropriate for some of the data you are plotting so be sure to zoom in on the axis to show a valid plot of the data.

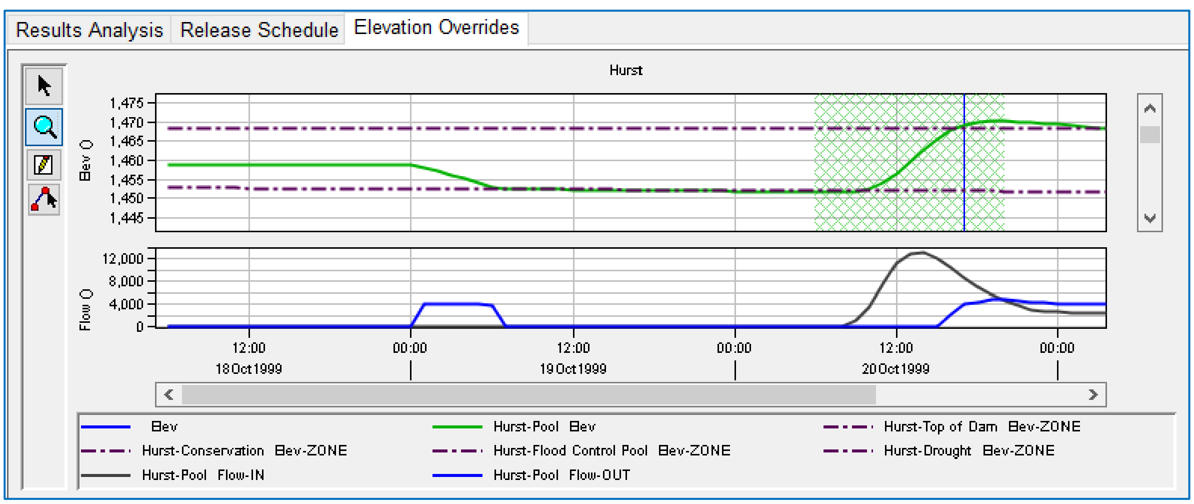


Figure I.13 OSI Plot Panel for a Variable with Additional Time Series Selected

Save the OSI Configuration.

When you have finished defining the tabs and their variables, select **Save** from the OSI's **File** menu to save your OSI configuration. Then you can close the OSI.

To retain your OSI configuration for use in another simulation, save the alternative to *base* by using the **Save to Base Directory…** option in the alternative's context menu in the Simulation Control Panel.

The OSI configuration is saved as part of the data of an Alternative. This means that the OSI can (and must) be configured separately for each new alternative. However, if an alternative is created, using **SaveAs**, as a copy of an existing alternative that already has an OSI configuration, the new alternative will include the OSI configuration from the original alternative.

I.4 Using the OSI for Reviewing Results

Depending on your model, you probably have at least a few locations and parameters whose results should be reviewed to determine how well your model is performing or if your operations are meeting your objectives. With the OSI, you can assemble these variables on one (or more) tabs to make the review of the results quick and easy.

For Example:

The Hayes Basin model (Figure I.14) has two reservoirs, *Hayes* and *Hurst*, and two downstream control points, *Carmichael* and *Smithford*. Further downstream is a water supply withdrawal location at *Maryland*.

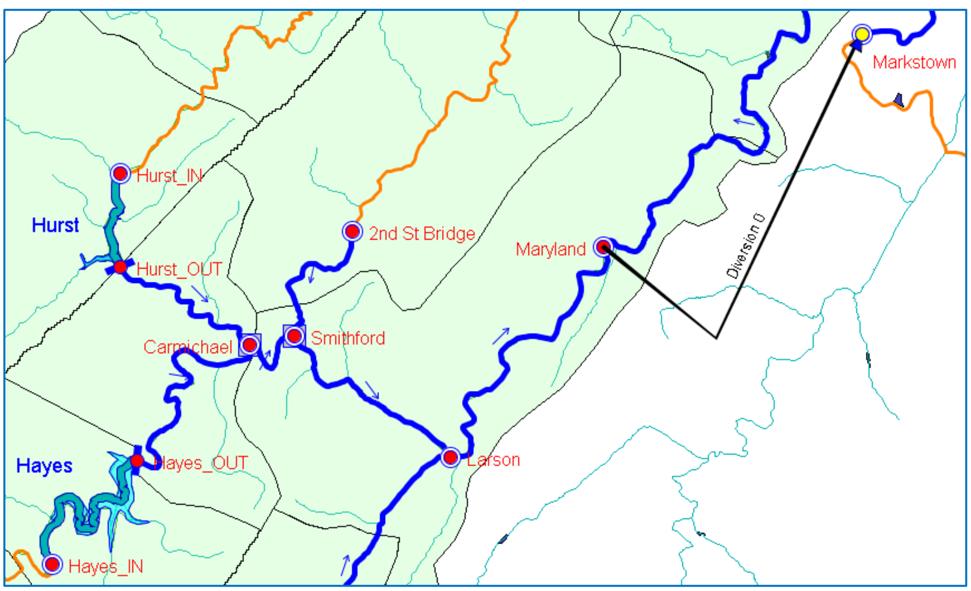


Figure I.14 An Example Watershed—Hayes Basin

We chose to create a tab to monitor results at the control points and the withdrawal location and we named it Results Analysis. On this tab we added three OSI Variables to display the:

- Carmichael Flow
- Smithford Flow
- Maryland Withdrawal Flow.

But the flow results at a location usually cannot stand alone. We needed additional information to analyze the results at the three locations including:

- the flow constraints at the control points
- releases from the reservoirs
- the local inflows
- and, for the diversion,
  - the inflow to the Maryland junction, and
  - the withdrawal demands.

To view the additional information, they can either be represented with additional OSI Variables or they can be plotted as "Additional Time Series…" with one or more of the first three identified Variables. We chose the latter as you will see below.

I.4.1 Adding Computed Parameter Variables

The Variable Type to use for simply reviewing results is the **Computed Parameter** type. Computed Parameter variables are available for all four elements types—Junction, Reservoir, Reach, and Diversion. Computed Parameter variables are not editable.

To add a Computed Parameter Variable to your tab:

- Select **Add Variable** from the OSI's **Edit** menu.
- A **New Operations Support Variable** (name) dialog will open (Figure I.15). Give the new variable a **Name** and an optional **Description** and click **OK**. (**Cancel** will abort the create process.)
- The **Operations Support Model Variable Editor** will open next. Use this editor to specify the variable and identify any additional time-series that you want plotted with it. Start by selecting the **Element Type**.
- Then, select the **Element** from the list of elements in your model.
- Now, select **Computed Parameter** as your variable **Type**.
- A field labeled **Computed Time Series** will appear below the **Type** field. Click the **Select** button to the right of this field.

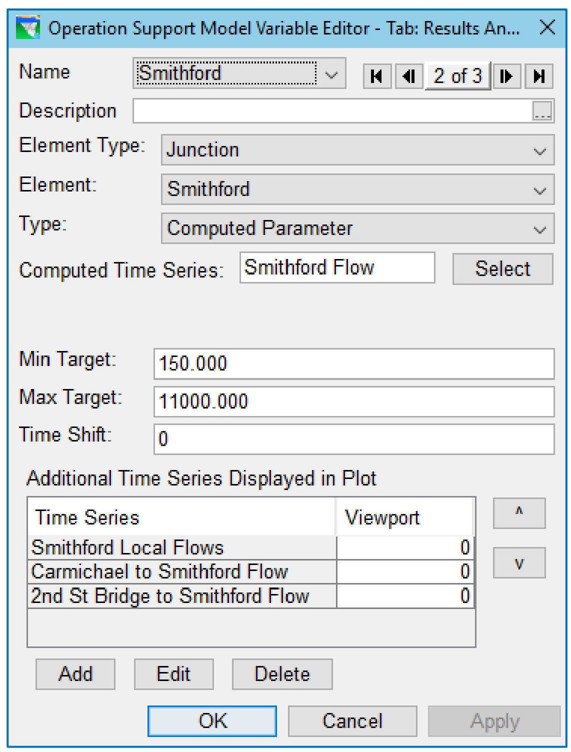


Figure I.15 OSI Variable Editor—Configured for a Computed Parameter with Additional Time Series

- The **Independent Variable Definition** dialog will appear (Figure I.11). Use it to select a variable to include with the plot of the OSI variable. Detailed instructions for using the **Independent Variable Definition** dialog are available in Appendix C

Figure I.15 above shows how we configured the Smithford OSI Variable for our *Results Analysis* tab. Notice that we chose to add the local and routed flows as **Additional Time Series…** to be displayed in the **Plot Panel** with the Smithford flow.

Figure I.16 shows the Results Analysis tab we assembled with the Smithford OSI Variable selected so that you can see how this variable is displayed. The **Plot Panel** shows a single plot window (viewport). In this plot window are curves for the Smithford flow as well as the other time series we chose to be plotted with it. In addition, the Minimum and Maximum limits that were specified for the variable are drawn with marker lines and a hatched fill. Since this variable is a computed parameter, it is not editable, so its data column in the Table Panel of the OSI is grey.

Although you might design a tab whose purpose is to simply display data for several variables (like we did in this example), you may find it useful to include a few Computed Parameter variables on the tabs you design for performing release overrides or computing incremental local flows. Look for Computed Parameter variables in the upcoming examples.

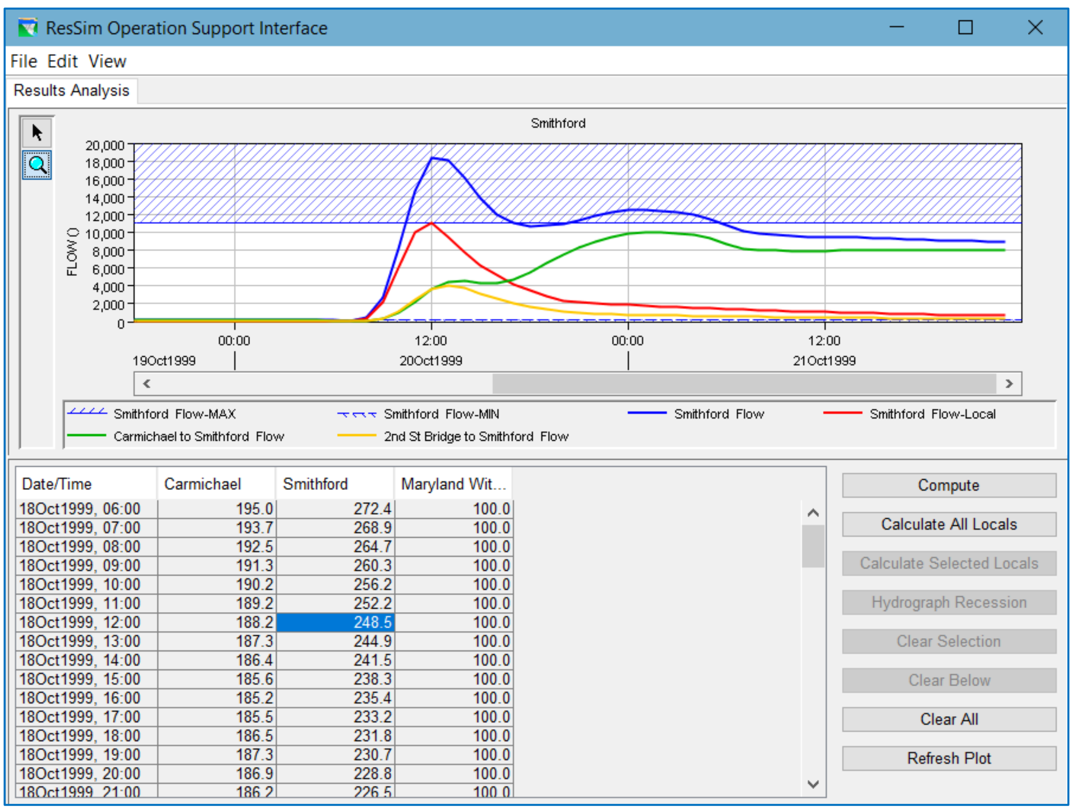


Figure I.16 OSI Example—Results Analysis Tab

I.5 Using the OSI to Perform Release and Elevation Target Overrides

Using a ResSim model to support real-time release decision making has always been assumed to include the use of release overrides to enable the user to develop a release schedule that best meets the overall requirements of the basin that few (if any) models could do without user interaction.

Unfortunately, the existing ResSim Overrides Editor requires too many "clicks" to define the overrides, compute, review results, and repeat… for the process to be considered "quick". The OSI was designed to streamline the process in a user-customizable way.

We will illustrate how to setup the OSI to perform Release and Elevation overrides using our Hayes Basin watershed:

> The Hayes Basin model has two reservoirs, Hayes and Hurst, and two downstream control points, Carmichael and Smithford. The reservoirs are operated as a system to meet minimum and maximum constraints at the control points. Hayes is about 4 times larger than Hurst but its drainage basin is only about twice that of Hurst's drainage basin.
>
> Hayes has substantial controlled release capacity of up to 15000 cfs at the top of conservation with a gated spillway added in near top of flood control. But the channel capacity below the dam is only 5000 cfs. Hurst has a maximum controlled release capacity of only 5000 cfs with a channel capacity of 4000 cfs. In order to pass large events, Hurst has an uncontrolled spillway whose crest is at the top of the flood pool.
>
> An unregulated basin, about half the size of Hurst's, flows into the system between Carmichael and Smithford. Flooding starts at Carmichael at approximately 8000 cfs and at 11000 cfs at Smithford.
>
> Although standard operations rules perform well during low to moderate inflows, they perform poorly during large events over the basin. To give the water management staff more modeling control, we developed an OSI with a tab for developing a release schedule using Release Overrides and a separate tab for using Elevation Target Overrides. How each tab was setup is described in the next two sections.

I.5.1 Configuring a Release Overrides Tab and Variable

To minimize the "*clicks*" needed for specifying a release schedule and reviewing the impacts, you may decide—like we did—to use the OSI as the tool for performing your release overrides (instead of ResSim's Overrides Editor) and to assemble the necessary reservoir release override variables in one tab along with the variables that show the response to the releases in the system.

As discussed in Chapter 15, a release override can be applied to any controllable outlet or outlet group in the reservoir outlet hierarchy, including the total release from the reservoir pool. For many systems, a single OSI variable to override the total release from each reservoir is good enough to meet the release schedule needs. For some systems, however, you may need to setup a release override OSI variable for multiple release elements of one or more of your reservoirs.

> For the Hayes Basin, we created a tab we called *Release Schedule*. On this tab we created a variable for each reservoir's total release override and a variable for each control point.

Figure I.17 shows our *Release Schedule* tab with the *Hurst* variable selected. The Plot Panel shows two viewports; the upper viewport contains two flow time series and the lower viewport contains an elevation time series. The flows in the upper viewport are the pool inflow and outflow; the outflow is the OSI Reservoir Release (override) variable and the inflow is one of the two variables we added to be plotted with the OSI variable. The pool elevation, displayed in the lower viewport, is the other additional time series we added to be plotted with the reservoir release variable.

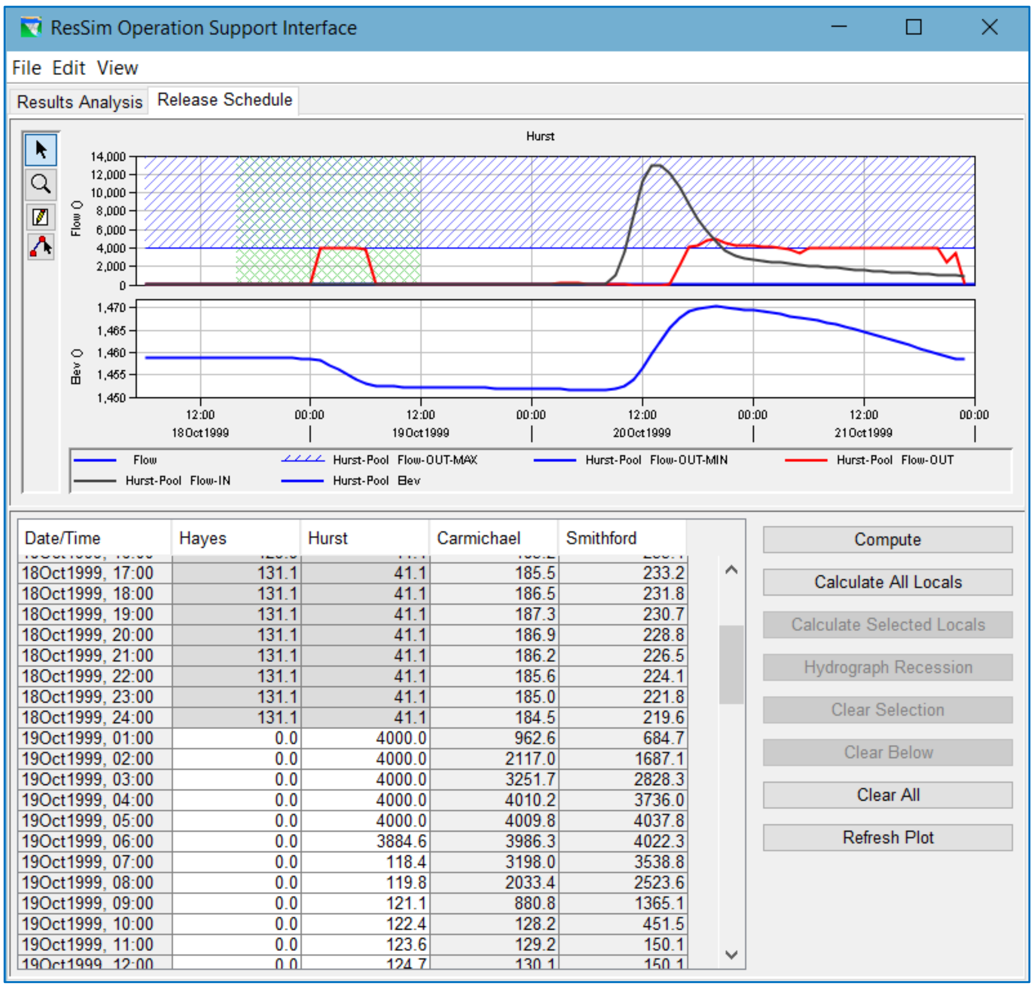


Figure I.17 OSI—Release Schedule Tab

Configuring a Release Override Variable

To create an OSI variable for performing release overrides:

- Create a new OSI variable by selecting **Add Variable** from the **Edit** menu. After you give the variable a name and click **OK** in the **New Variable** dialog, the **OSI Variable Editor** will open (Figure I.18).
- Select **Reservoir** as the **Element Type.**
- Select the reservoir from the **Element** list.
- Select **Reservoir Release** as the variable **Type**.
- Select the release element from the **Reservoir Release from** selector. The list includes all controlled outlets and outlet groups from your selected reservoir's outlet hierarchy. Consider carefully which release element you want to apply to your variable. Depending on your circumstances, you may want to setup a separate OSI variable for each controlled outlet or you may want just one variable to cover the total release. To override the total reservoir release choose the **Pool**; if you have a diverted outlet that you do not want affected by the override, choose the **DamTailwater**.
- If your reservoir has constant, at-site, minimum and/or maximum release limits and you would like to see those limits in the plot, enter those limits in the **Min Target** and **Max Target** fields.
- Consider adding key reservoir output time series to the plot with your reservoir release; we like to add pool inflow and pool elevation. To do so, use the **Add** button to open the Independent Variable Definition dialog.
- When finished, click the **OK** button to close the **OSI Variable Editor**.

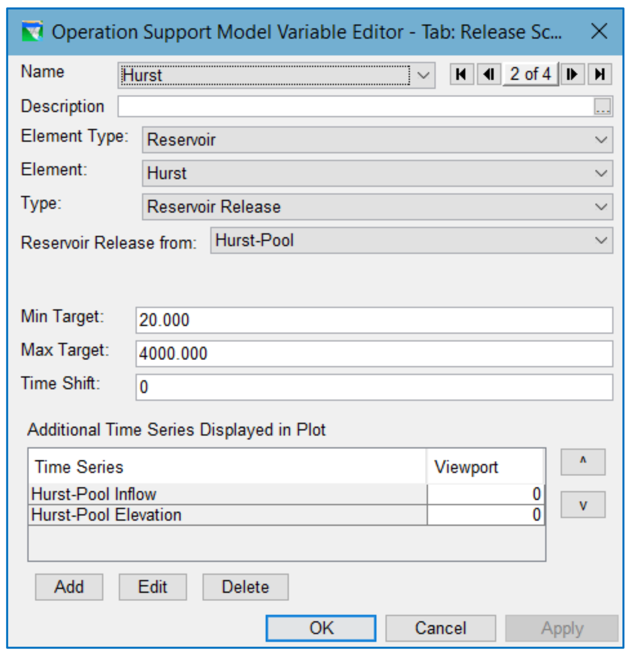


Figure I.18 OSI Variable Editor—Configured for a Release Override

Figure I.18 shows how we configured the *Hurst* variable in our example watershed. As suggested, we included the reservoir pool's inflow and elevation as **Additional Time Series…** and we entered the minimum and maximum at-site release limits in the **Min** and **Max Target** fields.

After setting up the **Reservoir Release** variables for *Hayes* and *Hurst*, we added two **Computed Parameter** variables, *Carmichael* and *Smithford* to show results at the two control points so that we could see—from the OSI—how our overrides were affecting the control points without having to go back to the main ResSim interface. In addition to the control point flow, we added additional time series to our two variables so that we could see how much unregulated (cumulative local) flow was reaching the control points, flow that the reservoirs could do nothing about.

Enter your Overrides and Compute

The OSI interface is designed to facilitate the specification of overrides, computing the alternative using the overrides, and reviewing results.

To enter overrides, you have two options:

- Specify the override values by entering data for the variable into the white cells in its column in the table panel. Fill options are available from the column's context menu to assist in entering data into the column. Each override value will appear in the table in green text and will also be drawn, in blue with magenta symbols, in the same viewport of the Plot Panel as the variable itself.
- Specify the override values by using the **Edit** tools in the **Plot Panel's** toolbar to draw the desired override on the plot viewport where the variable is displayed. Use the **Multi-point Edit** tool to draw a series of line segments by clicking with the tool's cursor in the plot region to draw each point. After placing the last point, *right-click* anywhere in the plot to stop the draw process. The values you specified in the plot will be drawn as a blue line and with magenta symbols; the same information will appear in the variable's column in the **Table Panel** in green text.

To compute the current alternative, you can use the **Compute** button at the top of the stack of **Action Buttons**; no need to return to the main ResSim interface.

I.5.2 Setting Up Elevation Target Overrides

Elevation Target overrides act as an override to the reservoir's guide curve. But, more than that, they short-circuit the reservoir's release decision logic—for each timestep that has an elevation target override, all reservoir operation rules and IF_Blocks are ignored and the reservoir decision logic follows only guide curve operation using the elevation target override as the guide curve objective and constrained only by physical capacity. Elevation Target overrides can be useful for lock & dam operation, hinge pool operation, or to assist in modeling unusual real-time situations where the operation of the pool is more important than the releases.

Configuring an Elevation Target Override Variable

To configure an OSI variable as an **Elevation Target** override:

- Select **Add Variable** from the **Edit** menu to create a new OSI variable.
- Give the variable a name and click **OK** in the **New Variable** dialog. The **OSI Variable Editor** will open (Figure I.19).
- Select **Reservoir** as the **Element Type.**
- Select the reservoir from the **Element** list.
- Select **Elevation Target** as the variable **Type**.

- If your reservoir has constant minimum and/or maximum pool limits and you would like to see those limits in the plot, enter those limits in the **Min Target** and **Max Target** fields.
- Consider adding key reservoir output time series to the plot with your pool elevation; we decided to add pool inflow and outflow as well as one or more key zone elevations. To do so, use the **Add** button to open the **Independent Variable Definition** dialog. Use it to select each time series you want to include in the plot with your release.

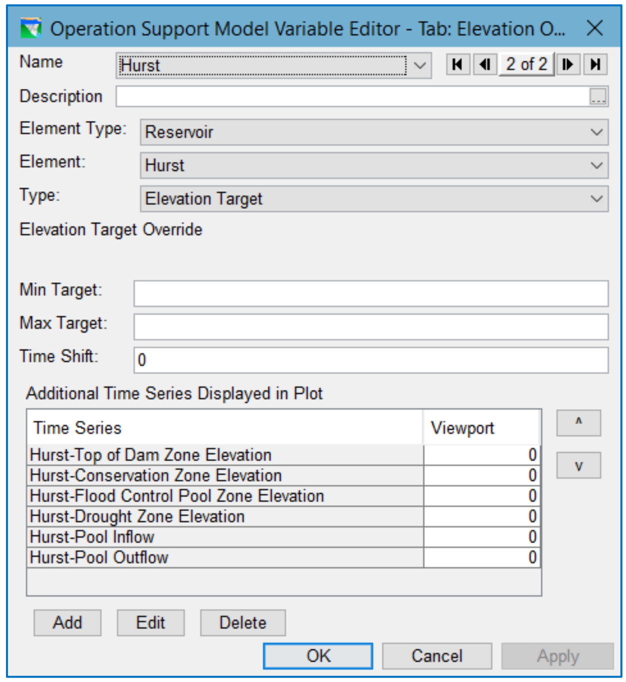


Figure I.19 OSI Variable Editor—Configured for an Elevation Target Override

- When finished, click the **OK** button to close the **OSI Variable Editor**.

I.6 Using the OSI to Compute Local Inflows

In general, there are two situations in which you may want to compute incremental local inflows from total flows at a location:

- In real-time operation where you want to compute the local inflow from observed data, then either use a recession equation to estimate the local inflow into the future or hand enter an estimate for the local inflow.
- To compute incremental local flow data for a study model using observed or computed total flow data at key locations throughout the watershed.

Although each of these situations use the same OSI configuration they each may require a significantly different setup in the network and/or its alternative. To reflect this the next two sections present the generalized set of instructions for configuring a fairly simple model to compute local flow and some addition concepts and instructions for configuring a complex study model for computing local flow.

I.6.1 Configuring the OSI (and Your Alternative) to Compute Local Inflows

To compute incremental local inflows from total flows requires that your network and alternative, as well as the OSI, be configured carefully. The basic steps you need to perform are described below.

NOTE—If your objective is to compute local inflows from observed data for use in a complex study model, see Section I.6.2 for more additional considerations and instructions for configuring your network and alternative.

Prepare the Alternative and Its Network

The equation used by ResSim to compute incremental local flow is:

Local Flow = Total Flow — Routed Flow

Where:

- Local Flow is the incremental local flow at the junction.
- Total Flow is the total flow leaving the junction; more specifically, it is the ***Observed*** flow at the junction.
- Routed Flow is the flow routed to the junction from upstream; more specifically, the upstream ***Observed*** flow routed to the junction.

In order to provide the correct information to the junction to compute the local flow, some changes may be necessary in both the reservoir network and the alternative.

The configuration requirements include:

Local Flow cannot be computed at headwater junctions since ResSim cannot route flow to these junctions. However, if an Observed flow is provided at a headwater junction, it will be used instead of the total inflow for the computation of local flow downstream.

Only one incremental local flow can be computed per junction element. Each junction should have no more than one entry on the Local Flow tab of the Junction Editor.

An Observed flow must be associated with each junction where local flow will be computed.

An Observed flow (representing the reservoir release) must be associated with the outflow junction of each reservoir upstream of a junction where local inflow will be computed.

To accomplish these requirements:

- Review the local inflow tab of each junction in your network. For any with more than one inflow defined, replace the set with a single composite inflow—e.g., use a flow-local output from an HMS junction rather than multiple subbasin flows. If a composite external time series option is not available, consider modifying the network to add additional junctions to receive the various locals—e.g., if one of the local inflows is a reach flow coming from a tributary in HMS that is not currently represented in ResSim, add the tributary and a junction at its mouth (to receive the reach flow) to the ResSim network.

- At each junction where you want ResSim to compute an incremental local and at each outflow junction of the reservoir above the junctions where you want incremental local flow computed, place a checkmark on the **Observed** tab of the junction editor to indicate that you have observed data for the total **Flow** from the junction (Figure I.20).

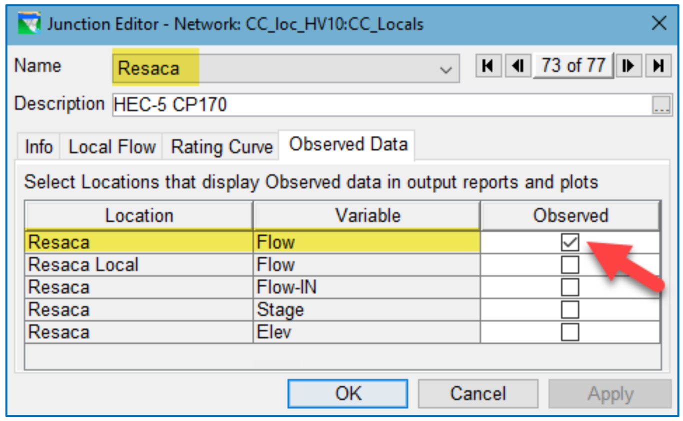


Figure I.20 Junction Editor—Activate Observed Data for Total Flow

- Save the network.
- Open the **Alternative Editor**. *Select the alternative to be used for computing local flows. This should be the same as or a copy of the real-time or study alternative that will be using the local flows.*
- *On* the **Observed Data** tab, identify an observed total flow time series for each junction's total flow variable in the list. Unless needed for other purposes, leave all other listed entries blank.

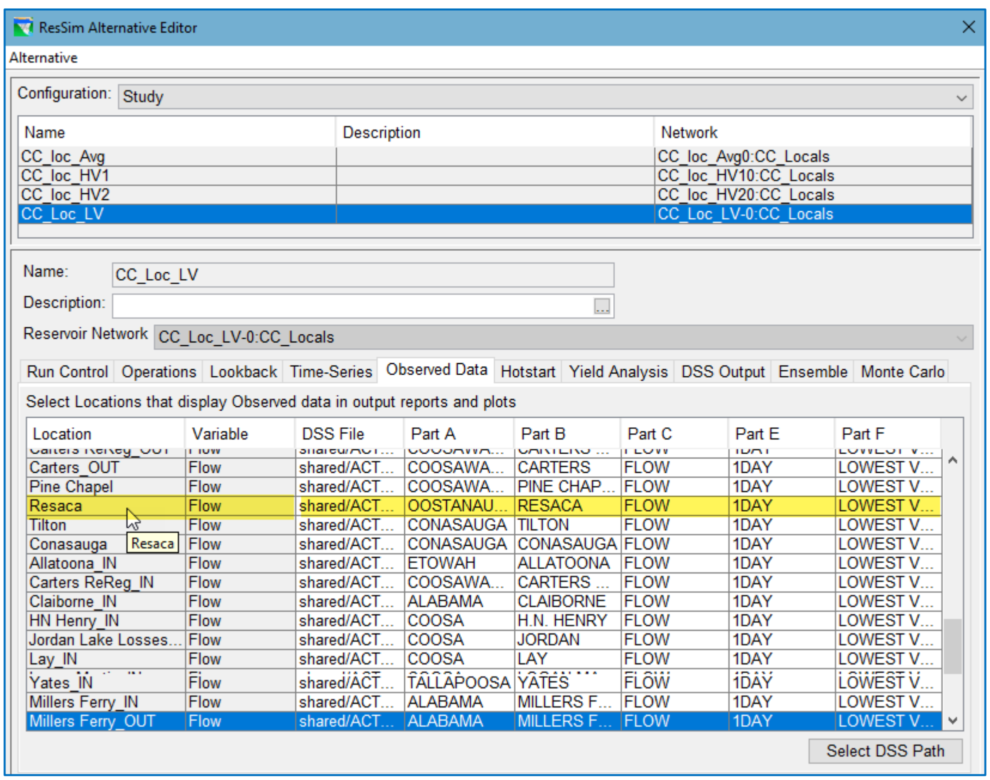


Figure I.21 Alternative Editor—Observed Tab—Identify the Total Flow Time Series for the Junction

- On the Time Series tab, identify an input time series for each inflow in your model. This may mean changing some entries that are already filled-in as well as filling-in the blank entries.

Caution

You may think that because ResSim is going to compute most of the inflows listed on the Time Series tab of the Alternative Editor, you can use a single dummy time-series record for each local inflow that will be computed. Unfortunately, you would be wrong!

Each inflow must have a uniquely named time series record associated with it in the Time Series tab of the Alternative Editor. This is because ResSim is going to write the locals that are computed by the OSI into that record in the simulation. If you do not have a unique time series for each location, create one and fill it with a constant value for the full time window over which you have observed data and for which you will be computing the locals.

OSI Configuration

ResSim and the OSI will only compute incremental local flow for those locations represented by an OSI Local Inflow variable.

The OSI can only be configured for a computed alternative in the Simulation module, so that where you must start:

- Open or create a simulation that spans the time window of the observed data. If the local inflow computation is for a real-time alternative, the time window should include the forecast period.
- Add your alternative to the simulation and compute it.
- Open the OSI and create a Local Flow tab. You may need more than one tab so consider creating a tab for the local flows within each major tributary of your watershed.

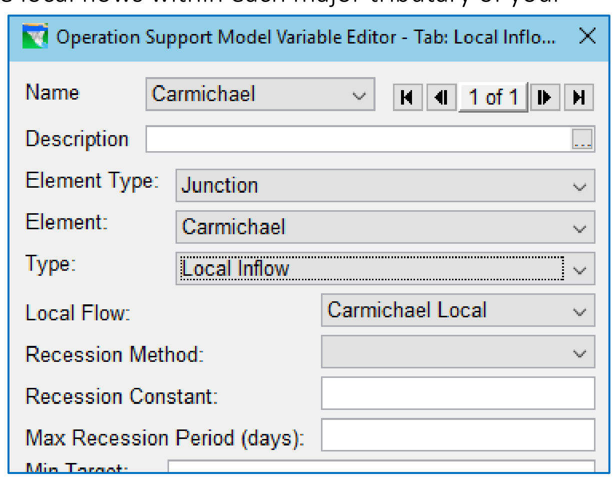


Figure I.22 OSI Variable Editor—Defining a Local Inflow Variable

For each incremental local inflow you want to compute:

- Create an OSI variable.
- In the **OSI Variable Editor**…
  - o Set its **Element Type** to **Junction**.
  - o Select the particular junction **Element**.
  - o Set the variable **Type** to **Local Inflow**.
  - o The **Local Flow** field will fill with the first (and hopefully, only) local flow listed on the **Local Flow** tab for that junction in the **Junction Editor**.

  - If you are computing locals from observed data for use in a study, you can leave the **Recession Method** and its parameters blank. However, if you are computing the locals for use in a real-time alternative, then select the **Recession Method** and specify the Recession Constant and the Max Recession Period in days.
  - Click **Apply** in the **OSI Variable Editor** to save your settings, then set the data in the rest of the editor to configure the plot of the variable. Add one or more variables to be plot with the local flow so that you can see and analyze the validity of the computed local flow. Suggested variables to add to the plot include: total computed junction flow and observed junction flow.
  - Click **Ok** when you are finished with the current local inflow variable and close the **OSI Variable Editor**.
  - Repeat these steps until you have created an OSI Variable for each local inflow to be computed.
- Once you have created an OSI variable for each local flow to be computed, save the OSI configuration. You may also want to copy the alternative back to the network module so that your OSI configuration can be used in other simulations. To do so, close the OSI, then *right-click* on the alternative in the **Control Panel** and select **Save to Base Directory…** from the context menu.
- You should now be ready to compute the local flows. See Section I.6.3 for instructions.

I.6.2 Preparing Your Study Model for Computing Local Inflows

The preparations needed in a study model may be more extensive than a real-time model depending on the complexity of the model. Although the "observed" data being used plays a role, the relevant aspects are the identification of the local inflows and specifying where observed data is required.

Create a Copy of the Network and an Associated Alternative.

Since the changes to your network can be extensive and potentially problematic for your study, we strongly recommend that you create a copy of the network and an associated alternative *before* making any changes for computing local flows. By making a copy of the study network and using the copy exclusively for the computation of the local inflows, any changes you make to the network for the computation of local flows will not affect the study model.

See Appendix F, Section F.3.1.2 for instructions and shortcuts for making the copies of the network and alternative.

Configure the Network for Computing Local Flows

The network changes that should be considered and addressed for a complex study model include:

Remove distributed locals. A local Inflow should be identified only at a gage location, where a total (observed) flow is available. More specifically—the only local flows that should be identified in the network are at the headwater junctions and at the interior junctions that represent gage locations for which (observed) total flow data is available.

Any local flow currently identified at a junction where there is no gage should be deleted. This will include the local flows that represent a fraction of a single local flow hydrograph that has been distributed to two or more junctions; the total local flow should (already) be identified at the appropriate gage location.

Any headwater junctions (on an ungaged tributary) that used a distributed inflow hydrograph or a basin-weighted multiple of a neighboring basin's inflow hydrograph should be carefully labeled so that it can be assigned a zero-inflow hydrograph in the alternative.

Associate an Observed flow with each junction where local flow will be computed and *with the outflow junction of each reservoir upstream of a junction where local flow will be computed.*

To address both these points requires two steps. The first step is performed on the reservoir network and is described here. The second step is performed on the alternative and is described later in this section.

In the network, place a checkmark next to the junction's flow (outflow) variable on the **Observed** tab of the **Junction Editor** for all junctions with an entry on the local flow tab and for all reservoir outflow junctions. Hint: the junction outflow is usually the first *Flow* variable in the list of **Variables** on the **Observed Data** tab of the **Junction Editor**; the associated **Location** name should be the name of the junction itself.

Remove losses from the routing reaches. Seepage losses are already accounted for (hidden) in the observed flow at a gage. If your reaches also remove water due to seepage, it will be "doubly-accounted for" in the computed locals.

Remove diversions…unless they are gaged. Like seepage, ungaged diversions are already accounted for in the observed flow at a gage. If you don't want to delete the diversions, set the Diversion **Method** of each diversion to **Constant** and set the value to zero. If you have a *gaged* diversion, set its Diversion **Method** to **Time-Series**.

If your model has one or more reservoirs with multiple inflow junctions and the inflows are not gaged, reconfigure each reservoir so that:

The reservoir has only one inflow junction. This is because there is no way to compute the individual local inflow from each tributary without observed data at each tributary inflow to the pool.

The flow from all upstream tributaries is appropriately routed to that single inflow junction. This is because all upstream flow must be routed to the inflow junction in order to compute the local inflow to the pool;

the computed inflow to the pool is used as the observed flow at the inflow junction.

Verify the routing. With the local inflows farther apart than they may have been in the study network, you should double-check that each routed observed hydrograph from an upstream gage compares well with the observed hydrograph at the next downstream gage. Also, pay attention to the timestep of the observed data. If it does not match the timestep of the study model alternatives, your routing parameters, or even your routing method, may need to be adjusted to properly route the observed flows.

Be sure to save the network when you are done making the necessary change an before you start configuring the alternative.

Prepare the Input Time Series Data

Before configuring the alternative you will need for computing local flows, you should prepare the data you are going to need for those computations. There are two types of data you'll need—the data that represents the "observed" data at a gage and the dummy data that will be over-written by the computed local flows.

The data that represents the observed data at the gages in your watershed will likely fall into one of the following three categories:

Actual observed data—this data is the historic gage record of observations that has been cleaned up and verified, a process that involves filling-in missing values, correcting invalid values, and verifying or correcting questionable values. This data also includes the computed data that is generated using the observed data such as flows values from observed stage and reservoir inflows from observed reservoir elevations and releases.

Unimpaired or Modified data—this data uses the historic record and adjusts it by adding back in estimates of evaporation losses and diversions. By making these adjustments to the observed data, the modified data can be used to generation the local inflows needed as input for a variety of different operational scenarios. These alternatives often result in the pool elevations that deviate significantly from what was observed which equates to evaporation quantities that did not occur historically. These alternatives may also use diversion demands that do not reflect what occurred historically.

Computed or Synthetic data—this data is generated by an external process that is informed by the historic record in order to simulate or estimate potential future hydrology. An example of this type of data would be the flows computed by a climate change model.

Although each of these potential data sets *may* require that you make specific changes or adjustments to the network and alternative before using them to compute local inflows, but for the most part they are interchangeable if the network is set up as described above. What remains are some data management tasks in preparation for configuring your alternative:

Gather the data. If you haven't done so already, assemble the observed data for all the gage locations and reservoirs in the model. This includes, if necessary, computing the inflows to the reservoirs using the reservoir elevation-storage tables and the observed elevations and releases. Both reservoir inflow and outflow will be used as observed data for the computation of the local flows into and below the reservoir.

Create a zero flow time series that spans the same time frame as the observed data. This time series will be used as the inflow to the headwater junctions where observed inflow is not available.

Create a time series for each local inflow to be computed. For each location where local inflow is to be computed, make a copy of the zero flow time series and name it for its location and parameter. For example, if you will be computing a local flow for the Carmichael junction, give the associated zero flow time series a pathname such as:

//Carmichael/Flow-Local//1HOUR/Computed/

Configure the Alternative for Computing Local Flows

Most of the work in configuring the alternative is in mapping (associating) the right data to the right input. To be sure you don't forget a key requirement, implement the following steps carefully and completely.

- Open the **Alternative Editor** and select the alternative to be used for computing local flows. This should be the alternative you created when you made a copy of the network.
- On the **Run Control** tab, <u>verify</u> that <u>the time step</u> matches the interval of the observed data; it doesn't make much sense to compute hourly inflows from daily data. But be careful here—if the alternative has a different timestep than the observed data, then the network was probably originally configured for that same timestep. Double-check that discrepancy this was addressed during the routing verification step. If not, you may need to go back to the network and adjust the routing parameters to properly route the observed data.
- On the **Operations** tab, <u>assign an operation set to each reservoir</u> in your network. Although this may have already been taken care of when you first created the alternative, if you had to re-build one or more reservoirs due to multiple inflows, you may find there are reservoir that need your attention. Note—when your network and alternative are properly configured, the reservoir operations are irrelevant to the computation of local flows. However, if you have a guide-curve-only or an at-site-rules-only operation set at each reservoir, use it instead of the study model's operations; there's no point wasting compute time on downstream or system operations.
- On the **Lookback** tab…
    - Set the **Lookback Release** for one of the outlets of each reservoir to **Time-Series** and set the rest of the outlets to **Constant** with a value of 0.0.

  - Set the **Lookback Elevation** for each reservoir to **Time-Series**. Since the reservoir operations are irrelevant to the computation of locals, the lookback elevation values should also be irrelevant; however, when you review results, you don't want to confuse yourself with nonsense at the reservoirs so realistic starting elevations should be used.

    If you do not have observed elevation data for a reservoir, set the **Lookback Elevation** to **Constant** and set the value to the guide curve elevation for that reservoir.

  - Set the **Lookback Diversion** of each ungaged diversion to **Constant** with a value of zero. For each *gaged* diversion, set the **Lookback Diversion** to **Time-Series**.

- On the **Time-Series** tab…
  - For each gaged headwater inflow, set the pathname to the observed flow data for that gage.
  - For each ungaged headwater inflow, set the pathname to the generic zero flow time series that you created.
  - For each interior junction where local flow will be computed, set the pathname to the uniquely-named zero flow time series that you created for that location.
  - For each reservoir **Lookback Release**, set the pathname to the observed reservoir release data.
  - For each reservoir **Lookback Elevation**, set the pathname to the observed reservoir pool elevation data.
  - For each diversion **Lookback Diversion** and **Input Time Series**, set the pathname to the observed flow at the diversion gage.
- On the **Observed Data** tab…
  - For each reservoir outflow junction, set its outflow to the pathname of the reservoir's observed release data.
  - For each reservoir inflow junction, set its outflow to the pathname of the reservoir's computed inflow data.
  - For each remaining interior junction where local flow will be computed, set its outflow to the pathname of the associated gage's observed flow data.
  - For each gaged headwater junction, set its outflow to the same pathname you used for the junction's inflow, the observed flow data for that gage.

Configure the OSI

Once your network, alternative, and input data are ready, you can configure the OSI to compute the local flows. Refer to Section I.6.1 for the instructions on configuring the OSI and defining Local Flow OSI Variables.

I.6.3 Computing Local Flows

- Open or create a simulation that spans the time window for which you want to compute local flows and for which you have the observed data needed.
- If necessary, edit the simulation and add the local flow alternative you created and configured for use in computing the local flows. If the alternative is already in the simulation, you may need to update it to include any changes you made in the network module. To do so, right-click on the alternative in the Control Panel and select **Replace From Base Directory…** from the context menu
- Compute the alternative.
- Open the OSI and select (one of) the tab(s) you created for the local flow OSI variables.
- *Click* on the data in one of the variable columns to make that variable appear in the plot. Review the flow for the junction to verify that the standard compute produced valid results for the input provided.
- *Click* the **Calculate All Locals** button in the stack of **Action Buttons**. As described in Section I.2.5, a special compute type will be launched in ResSim to compute the local flow for each Local Flow OSI variable defined in the OSI.
- Review your results carefully to be sure the local inflows were calculated correctly. It may take a few iterations before you have the network, alternative, and OSI configured correctly for your system. Between each attempt, if may be necessary to **Rerun** the **Extract** of the model data so that your computed local flows are replaced with the original input data.
- When you are satisfied with the computed local flows, copy the data to a file in the shared folder of the watershed. Be sure to name both the files and the records carefully to identify the data and its source.

I.7 Workflow

The OSI configuration is saved with the Alternative data. This means that the OSI can (and must) be configured separately for each new alternative; however, if an alternative is created using SaveAs from an existing alternative that already has an OSI configuration, the new alternative will include a copy of the OSI configuration from the original alternative.

To configure the OSI for a given alternative, a simulation must first be created and the alternative computed.

With the desired Alternative active, select **Simulation→Operation Support Interface** from the main menu of the **Simulation** module. The **ResSim Operation Support Interface** will open. The OSI will initially be empty. Add tabs and variables (as described in the following sections). A basic arrangement will be to show headwater local flows first, then other locals, then reservoir releases and supporting time series. From the menu bar select **File→Save** to save the OSI configuration. From the Alternative, the **Save to Base** command allows the OSU to be used with the Alternative in new Simulations, while the **Replace from Base** command allows the configuration to be used in existing Simulations.

Typical workflow once the OSI has been configured for an Alternative is as follows:

- Create a new Simulation using the desired Alternative
- Run the simulation at least once from the Simulation Model
- Open the Operations Support Interface
- Compute All Local Inflows
- Perform Hydrograph Recession on Headwater Locals
- Perform Hydrograph Recession on other Locals
- Edit Reservoir Releases or Elevation Targets

The OSI overwrites time series data for Junction Known Flows in the simulation.dss file and release and elevation targets in the overrides dss file associated with the Alternative.