US Army Corps
of Engineers
Hydrologic Engineering Center

Real-Time Simulation (HEC-RTS) User's Manual

Version 3.1.1

Approved for public release; distribution is unlimited.

REPORT DOCUMENTATION PAGE

Form Approved OMB No. 0704-0188

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

4. TITLE AND SUBTITLE
HEC-RTS, Real-Time Simulation
User's Manual
Version 3.1.1

5a. CONTRACT NUMBER

5b. GRANT NUMBER

5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S)
CEIWR-HEC

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-96

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

14. ABSTRACT
The HEC-RTS (Real Time Simulation) is based on the CWMS (Corps Water Management System) software for use by non-USACE offices. HEC-RTS still provides the same data and information that is part of the CWMS software, HEC-RTS just performs these functions in a different manner than CWMS. This document is the User's Manual for Version 3.1.1 of the HEC-RTS software. This manual describes the capabilities of the software and illustrates its graphical user interface (GUI). The manual details how HEC-RTS is configured to use analysis software that is part of the system. The manual explains the flow of data and information between analysis software and the data components used within HEC-RTS.

15. SUBJECT TERMS
HEC-RTS, CWMS; Corps Water Management System; U.S. Army Corps of Engineers; HEC-HMS; HEC-ResSim; HEC-RAS; HEC-FIA; HEC-DSSVue; DSS; modules; data, reservoirs; lock and dams; projects; visualization; forecast; alternative; model; water resources; water management; mission; Civil Works; river; flow; flood control; navigation; hydroelectric; water quality; authorized purposes; Federal; precipitation; watersheds; consequences; water managers

16. SECURITY CLASSIFICATION OF: a. REPORT	b. ABSTRACT	c. THIS PAGE	17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
U	U	U	UU	485	19b. TELEPHONE NUMBER

Real-Time Sumulation (HEC-RTS) User's Manual

Version 3.1.1
Exported on 05/28/2025

System Developer:
U.S. Army Corps of Engineers
Institute for Water Resources
Hydrologic Engineering Center
609 Second Street
Davis, CA 95616-4687

(530) 756-1104
(530) 756-8250 FAX

Table of Contents

Search

1 Recently Updated Pages

- .HEC-RTS Users Manual v3.1.1
  Dec 16, 2023 • updated by Matt Fleming • view change
- .HEC-MetVue Model Alternative v3.2.3
  Jun 26, 2022 • created by Fauwaz Hanbali
- .What's New About HEC-RTS Version 3.2.3 v3.2.3
  Apr 25, 2022 • updated by Alex Davis • view change
- .Forecast Runs v3.2.3
  Apr 22, 2022 • created by Alex Davis
- .Forecasts v3.2.3
  Apr 22, 2022 • created by Alex Davis

2 Introduction

The U.S. Army Corps of Engineers (USACE) operates about 700 storage reservoir and lock and dam projects constructed under the USACE's Civil Works water resources program. The USACE water management mission is to regulate river flow with these projects to provide national benefits of flood control, navigation, hydroelectric power generation, water supply, irrigation, erosion control, water quality, environmental enhancement, and other authorized purposes. The Corps Water Management System (CWMS) is a real-time decision support system that expands and enhances the data and information available to USACE staff members who must make decisions about the operation of Federal water management facilities or who must monitor and approve such decisions made by operation partners. The data and information made available through CWMS includes precipitation data and forecasts as well as data and information about the current state of watersheds, likely future state of watersheds, and consequences of management actions. The data and information help water managers and others make informed operation decisions.

The Hydrologic Engineering Center's (HEC) Real Time Simulation (HEC-RTS) software is based on the CWMS software for use by non-USACE offices. HEC-RTS still provides the same data and information that CWMS does, HEC-RTS just performs these functions in a different manner than CWMS. This document is the User's Manual for Version 3.1.1 of the HEC-RTS software. This manual describes the capabilities of the software and illustrates its graphical user interface (GUI). The manual details how HEC-RTS is configured to use analysis software that is part of the system. The manual explains the flow of data and information between analysis software and the data components used within HEC-RTS.

2.1 Data and Information Needed by Water Managers

To make informed operation decisions, water managers need:

- Current and potential future scenarios of precipitation.
- Data that describes the current state of watersheds, channels, and water management facilities, including reservoirs, diversions, and other controllable features of the system.

Tip	An HEC-RTS watershed is a set of data, information, models, and images that represent watershed lands and the channels, gages, and water control features within the watershed.

- Information about the likely future state (e.g., one hour to two weeks) of the watersheds, channels, and management facilities.
- Information about the consequences of management actions that alter future states of the natural and managed systems.

Data that describes the current state of the system comes from a network of environmental sensors. These sensors, which are owned and operated by Federal, state, and local government agencies, utility companies, and commercial enterprises, measure:

- Weather conditions, including air temperature, precipitation depths and rates, and evaporation depths and rates.
- Watershed states, including snow accumulation.
- Depth, velocity, and other conditions in streams, rivers, canals, and other waterways.
- Lake or reservoir level (from which storage volume may be inferred), rates of release of water through outlets, settings of spillway gates, and other conditions of lakes, reservoirs, and diversions.

Data from sensors are transmitted by radio, satellite, telephone, the Internet, and other media to receiving sites, and then to water managers. There, the data are decoded, transformed, checked for quality (validated), and stored in databases. With this data, water managers have near-real-time reports on the current state of the watersheds, channels, and management features.
Using the environmental data from the databases, as well as forecasts, as inputs to models of watershed and channel processes, water managers can forecast future availability of water. A water manager can predict the runoff from a watershed, hours or even days into the future because of observed and forecasted precipitation values within the watershed. To do so, the water manager uses a mathematical model that simulates infiltration, overland flow, baseflow, channel flow, and other relevant watershed and channel processes.

Tip	A forecast is a simulation of watershed processes and consequences of flooding based on input data and information and hydrologic, reservoir operation, hydraulic, and impact analysis models. Forecast results include flow and stage in the channel from watershed runoff, reservoir release schedules, floodplain inundation maps, floodplain consequence reports, and reports listing actions for emergency responders to take. These results inform water management decision making.

With models of water control facilities, water managers can simulate and assess the impact of operation alternatives. For example, a water manager can determine which of two operation alternatives will more likely result in higher downstream water levels due to a large storm. The forecast of future inflow, combined with a mathematical model of the behavior of the reservoir and the downstream channel, makes this possible.

One operation alternative could be to release water now from a rapidly filling reservoir to accommodate future inflows. Another alternative could be to delay release in anticipation that inflows will diminish, and large releases will not be required. The manager has, with analysis software, the capability to compare these operation alternatives in a quantitative manner. Information from the simulation permits the manager to assess the economic, environmental, life safety, and other consequences of the operation alternatives. This information will lead to better-informed decisions.

2.2 HEC-RTS Overview

HEC-RTS provides data and information needed to water managers readily through DSS (HEC's Data System Storage) files. The HEC-RTS GUI provides a user with the ability to configure watersheds, view and edit data/information, create, and run forecasts, and view results. The functions of HEC-RTS are organized into four groups (modules) - setup, acquisition, visualization, and modeling.

The **Setup Module** contains commands for watershed setup. These commands include: configuring inputs, models, and outputs that describe a watershed's behavior. A visual representation of the watershed is developed and displayed in the GUI that is map-based.

The **Acquisition Module** contains the commands for data acquisition. These are commands for monitoring receipt of data from DSS files, validating the quality of incoming data, transforming the data (e.g., stage to flow), and editing the data, as required.

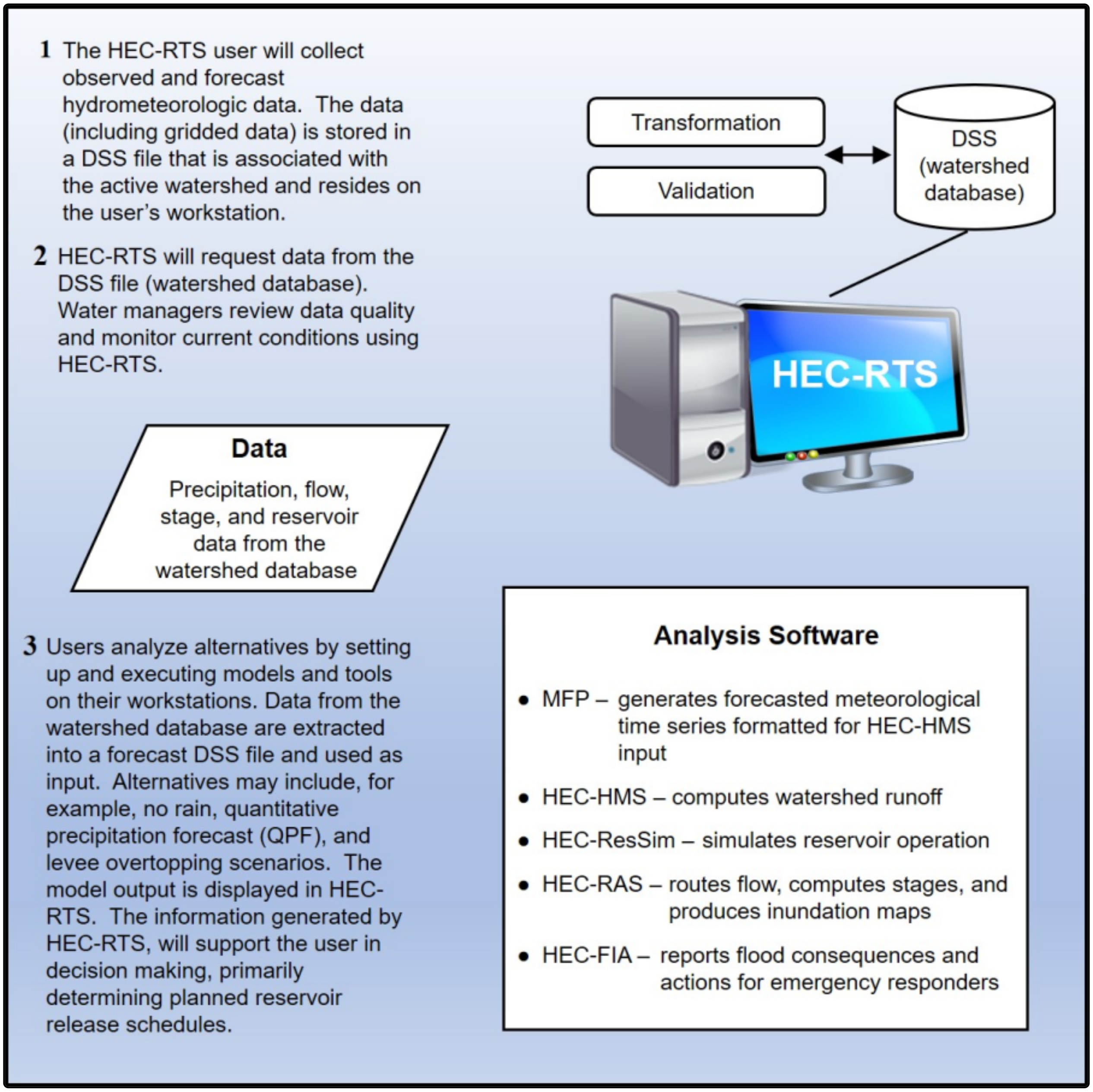
The **Visualization Module** contains the commands for data visualization. These commands include displaying observed and forecast data to evaluate the hydrometeorological state of the watershed. Water managers may need to review hourly data from hundreds of gages as they make decisions. In the visualization module, HEC-RTS provides tools to facilitate review of large amounts of data, including summaries presented as graphs, tables, spreadsheets, charts, river profiles, maps, or sometimes a combination of these. Within the Visualization module, the summaries are linked to a watershed map, so that a user can click on an icon and immediately view the data associated with that location or also view computational results.

The **Modeling Module** contains the commands for model execution. These commands assist with setting up and executing the analysis software for forecasting and viewing results. HEC-RTS links the analysis software so that individual models are executed in an orchestrated manner. Data and other inputs are passed to each piece of software with the DSS data exchange software.

HEC-RTS links the selected analysis software for a forecast, so that the selected software applications are executed in an orchestrated manner. Data and other inputs are passed to each piece of analysis software through the DSS data exchange software. The available analysis software in HEC-RTS meets modern software standards, including an easy-to-use GUI, and executes within operating systems selected by USACE. The analysis software available in HEC-RTS Version 3.1.1 is:

MFP	Meteorologic Forecast Processor - processes meteorological forecasts for input to HEC-HMS. Inputs are forecasted meteorological data such as precipitation and temperature. These forecasts can be entered manually or obtained from external sources such as NWS (National Weather Service). Outputs are forecasted meteorological time series formatted for compatibility with HEC-HMS.
HEC-HMS	Hydrologic Modeling System - simulates watershed response to precipitation. Inputs may include observed or forecasted precipitation, temperature, snowpack, and other environmental conditions. Outputs include flows throughout the watershed, including inflows to reservoirs and local flows below the reservoirs.
HEC-ResSim	Reservoir System Simulation - simulates behavior of reservoirs and linking channels, following user-specified operations for reservoir release decision making. Inputs include flows into reservoirs and unregulated flows downstream of reservoirs (from HEC-HMS). Outputs include reservoir releases, downstream regulated flows, and reservoir storage conditions.
HEC-RAS	River Analysis System - simulates behavior of channels and adjacent floodplains. The simulation of channels is performed one-dimensionally, while the simulation of adjacent floodplains can be performed one- or two-dimensionally. The output from HEC-RAS permits determination of water surface elevations corresponding to flows computed by HEC-HMS or HEC-ResSim. Inputs include flows, and outputs include water surface elevations, depth grids, and inundation maps.
HEC-FIA	Flood Impact Analysis - estimates the consequences of flow or water surface elevations in the system. Inputs include computed or observed flows or water surface elevations throughout the flood plain. Outputs include economic, life loss, or other measures of impact, or optionally, information on actions to be taken in response to flows or water surface elevations that will be experienced.

HEC-RTS ensures that those who need to know the current state of the defined watershed and likely future states have ready access to that information. The capability to access this information is accomplished using information sharing technology, including specially designed websites for display. The flow of data and information through HEC-RTS is displayed in Figure 1.



1 Figure 1 Flow of Data and Information through HEC-RTS

2.3 What's New About HEC-RTS Version 3.1.1

HEC-RTS Version 3.1.1 has been enhanced from previous versions. Version 3.1.1 has the following new features:

- **Updated Analysis Software.** Version 3.1.1 includes the latest version: HEC-HMS - 4.5; HEC-ResSim - 3.4.1 Dev; HEC-RAS - 5.0.7; and, HEC-FIA - 3.1 (modified version).

- **Team Modeling.** Several new features were added to the Team Modeling capabilities, such as: a visual indicator when items are out-of-sync with a master watershed; **Save to Master** item was added when closing a forecast; and, highlighting of forecasts that are not downloaded.
- **HEC-RAS New Capabilities**. HEC-RAS storage areas and two-dimensional display support are now available in HEC-RTS Version 3.1.1. Also, HEC-RTS Version 3.1.1 supports an HEC-RAS restart file.

2.4 Organization of User's Manual

This manual is generally organized in parallel with the HEC-RTS workflow and could be separated into four sections. The section focused on using HEC-RTS precedes the section focused on configuration because using HEC-RTS is a routine task while setting up watersheds is an infrequent task. Supplemental materials are also included in the manual for additional topics and where more detail is needed.

Section I: The Basics • Useful for all staff. • Overview of the basic HEC-RTS "infrastructure". • Walkthrough of HEC-RTS and introduction layers.	• HEC-RTS Interface • Understanding Layers • Primary Layers • Map Layers
Section II: Using HEC-RTS • Targeted toward operations staff. • How to view and monitor data. • How to validate and edit data • How to run a forecast. • How to view, manage, and post results	• Using HEC-RTS – Overview • Acquisition Module • Data Status Lists • Data Validation Editor • Visualization Module • Modeling Module • HEC-RTS Forecasts • HEC-RTS Results
Section III: Configuring HEC-RTS • Targeted toward modeling staff. • How to configure a watershed. • How to configure model alternatives. • How to configure forecast alternatives. • How to edit model parameters.	• Configuring HEC-RTS • Setup Module • Program Order • Time Series Icons • Time Series Icons Layers • Model Alternatives and Forecast Runs • Model Linking Editor • Data Extract and Post Editors • Team Modeling

Section IV: Supplemental Materials • Supplemental information not covered in the preceding documentation. • More in-depth explanation of features.	• Glossary • HEC-RTS Application Settings • Setting Up the Coordinate System • Using Map Editors • Using the Color Chooser • Using HEC-DSSVue • Scripting • Plots • Printing and Copying HEC-RTS Data • Stream Alignment • WISKI Extract and Post • KiWIS Write Extension

Screenshots were captured using example watersheds. In the GUI, some dialog titles and menu options correspond with the names of items that a user might have specified in the watershed created by a user. Thus, dialog titles and menu options shown in the user's watershed may vary from what is shown here.

2.5 Additional Documentation

This user's manual is the first volume in a series of HEC-RTS documentation. In addition to this manual, other HEC-RTS documentation includes:

HEC-RTS Quick Start Guide	Steps the user through the development of an HEC-RTS watershed.
HEC-RTS Applications Manual	This document is a combined engineering applications/technical reference manual intended to assist engineers in the development of models and the implementation of HEC-RTS.
Analysis Software Documentation	Documentation on the analysis software that is part of the system, such as HEC-HMS, HEC-ResSim, HEC-RAS, and HEC-FIA, are available on HEC's website: http://www.hec.usace.army.mil/. However, Version 3.1.1 may include versions of the analysis software that have not been released to the public. For those, version-specific documentation is available from HEC upon request.

3 HEC-RTS Interface

Chapter 1 describes how HEC-RTS creates data and information that is readily available to water managers and introduces the analysis software. The GUI is the steering wheel of HEC-RTS, and features graphical menus and tools organized by function, or modules. The four modules include:

- **Setup** for configuring a watershed (coordinate system, map layers, program sequence, model alternatives, forecast runs, extracts, posts).
- **Acquisition** for data acquisition, monitoring, validation, and editing.
- **Visualization** for viewing data.
- **Modeling** for setting up and executing analysis software for forecasting and viewing results.

This chapter provides an overview of the GUI and other basic functions of the HEC-RTS software. Here are three navigation tips:

- For simplicity, the GUI "hides" commands that are not applicable to the module or element the user is working on. Thus, if an option appears to be missing, make sure that the correct module or element has been selected and that the option is applicable for that selection.
- Directory trees list elements configured in the watershed. When a user configures a new element, it will be added to the directory tree.
- Commands related to a selected element are typically accessible by right-clicking on the element.

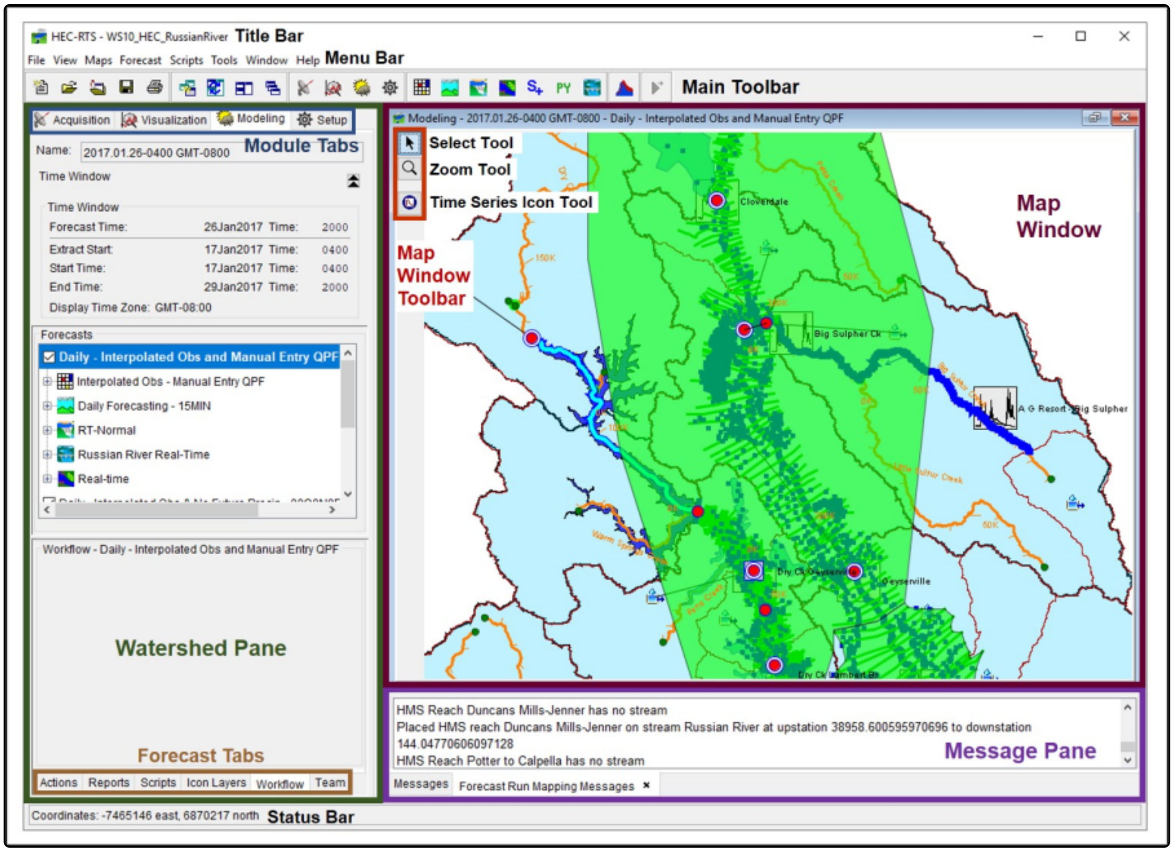
3.1 Starting HEC-RTS

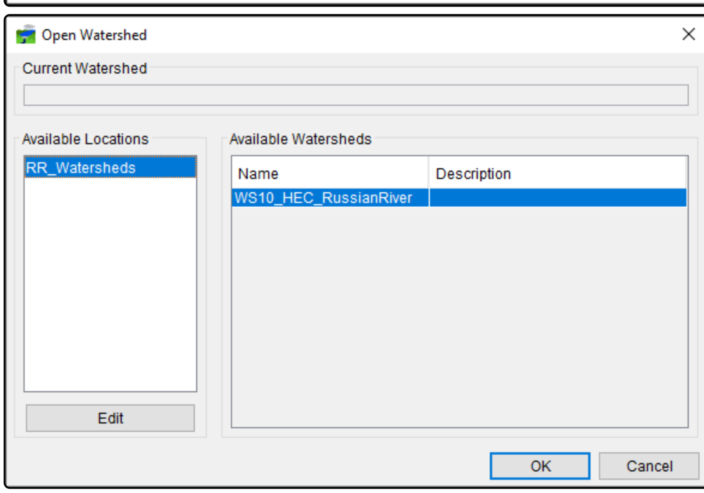
When starting HEC-RTS, double-click the HEC-RTS icon on a user's desktop, or from the taskbar click **Start**, point to **All Programs**, point to **HEC**, point to **HEC-RTS**, and click **HEC-RTS**. The HEC-RTS splash dialog box appears for a few seconds, and then the main window of HEC-RTS displays. The user is now ready to start using HEC-RTS.

3.2 Opening an Existing Watershed

In HEC-RTS, a **watershed** is a set of data, information, models, and images that represent watershed lands and the channels, gages, and water control features within the watershed. A user may open an existing watershed from any of the modules. (If the user needs to create a new watershed, refer to Creating a Watershed.) There are two methods available to open a watershed. First, if the user has opened the watershed before, the user can select the watershed from the **File** menu, point to **Recent Watersheds,** and click on the watershed name. The second way to open a watershed follows:

1. From the HEC-RTS main window (Figure 1), from the **File** menu, click **Open Watershed**, the **Open Watershed** dialog will open (Figure 2).





2. From the **Available Watersheds** box, select the watershed, click **OK**. The **Open Watershed** dialog will close (Figure 2), and the watershed will display in the HEC-RTS main window (Figure 1)

Tip	There can only be one watershed open at a time. Therefore, if a watershed is open and an attempt is made to open another watershed, HEC-RTS will close the initially opened watershed. HEC-RTS first prompts the user to save the watershed.

3.3 HEC-RTS Main Window

From the HEC-RTS main window, each module - **Setup**, **Acquisition**, **Visualization**, and **Modeling** - has menus, tools, and a map window specific to each module. Following are the basic elements of the HEC-RTS main window:

Title Bar	Displays the name of the active watershed.
Menu Bar	Contains general and module-specific menus.
Main Toolbar	Contains shortcut icons for menu commands.
Module Tabs	Each tab represents an HEC-RTS module. Clicking a module tab opens that module in the HEC-RTS main window.
Watershed Pane	The area where the module specific commands and items are located.
Map Window	A map window displays the watershed representation, which is geographically-referenced (geo-referenced).
Map Window Toolbar	Contains tools to navigate within the map window, and setup time series icons.
Message Pane	Displays a scrolling list of messages. System output log information appears in this window from the time you start HEC-RTS until you exit. Messages related to incoming data are colored coded based on the quality of the data. Messages may also report a problem with a system component; problems with field equipment; a caution, warning, or flood event; or other critical situations.
Status Bar	Displays map coordinates when the select tool is hovering over a location in the map window.

3.3.1 Menu Bar

The **File**, **View**, **Maps**, **Scripts, Tools**, **Window**, and **Help** menus (HEC-RTS Main Window) contain commands you can access from all four modules. Some of these menus also contain module-specific commands that appear only within individual modules. The following are descriptions of the commands and options that are common to all four modules.

File	From this menu, you can create, open, or close a watershed, save data associated with the watershed, and exit HEC-RTS. In addition, your most recently used watersheds are located after the ability to open a watershed in the **File** menu (Open Watershed Dialog). Available commands are: **New Watershed, Open Watershed, Recent Watersheds, Save Watershed, Close Watershed,** and **Exit**.
View	This menu is used to add or remove screen items (toolbars, module tabs, message pane, status bar); display watershed units; display time zone; zoom map window to the defined maximum extents; and save, restore and manage layouts. Available commands are: **Zoom to All, Displayed Units, Displayed Time Zone, Toolbars, Messages, Status, Module Tabs, Save Layout, Restore Layout**, and **Layout Manager**.
Maps	This menu is used to adjust the how the map panel of the HEC-RTS main window is displayed including map layers, map background, and whether coordinates are displayed. In addition, you can save the image in the map window as a graphics file, and you can print the image in the map window. Available commands are: **Map Layers**, **Default Map Properties**, **Add Internet Map**, **Display Map Coordinates**, **Save Map Image**, and **Print Map**.
Scripts	This menu allows you to execute existing scripts, create, test new scripts and run scripts. Available commands are: **Editor**, **Schedule Script Job**, **Script Job Status**, and **Run.**
Tools	This menu allows you to access data (HEC-DSSVue), set options for the HEC-RTS software, specify plug-in editor locations to use, view the *console.log* file, view the log file for the software applications being used (HMS, ResSim, RAS, FIA) and the log file for DSS processes, and view memory usage. Available commands are: **HEC-DSSVue**, **Applications, Options**, **Model Version Editor, Console Output**, **Logs**, and **Memory Monitor**.
Window	This menu allows you to select how you want to view and select multiple windows, which modules you want to view. Available commands are: **Duplicate Window**, **Detach Window**, **New Map Window**, **Map Window Settings**, **Sync Map Windows**, **Tile**, **Cascade**, **Next Window**, and **Previous Window**.
Help	Displays current version information about HEC-RTS and accesses the HEC-RTS User's Manual. Available commands are **User's Manual** and **About**.

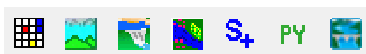
3.3.2 Main Toolbar

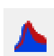
The main toolbar of the HEC-RTS interface provides quick access to the most frequently used option from the HEC-RTS menu bar and consists of the following six toolbars:

Standard Toolbar 
From this toolbar you can create a new watershed; open a watershed; close a watershed; save a watershed; and, print what is displayed in the Map Window.

Map Toolbar 
From this toolbar you can open a new map window; set the zoom level for all open map windows to the selected map window's zoom level; in the **Map Panel**, tile all open map windows; and, in the **Map Panel**, stack all open map windows.

Modules Toolbar 
From this toolbar you can select the individual Module Tabs – **Acquisition, Visualization, Modeling**, and, **Setup**.

Modeling Toolbar 
From this toolbar you can open MFP (need to have an MFP alternative selected); open HEC-HMS; open HEC-ResSim; open HEC-FIA; open the **Supplemental Program** dialog; open the **Scripting Program** dialog; and, open HEC-RAS.

Tools Toolbar 
From this toolbar you can open HEC-DSSVue.

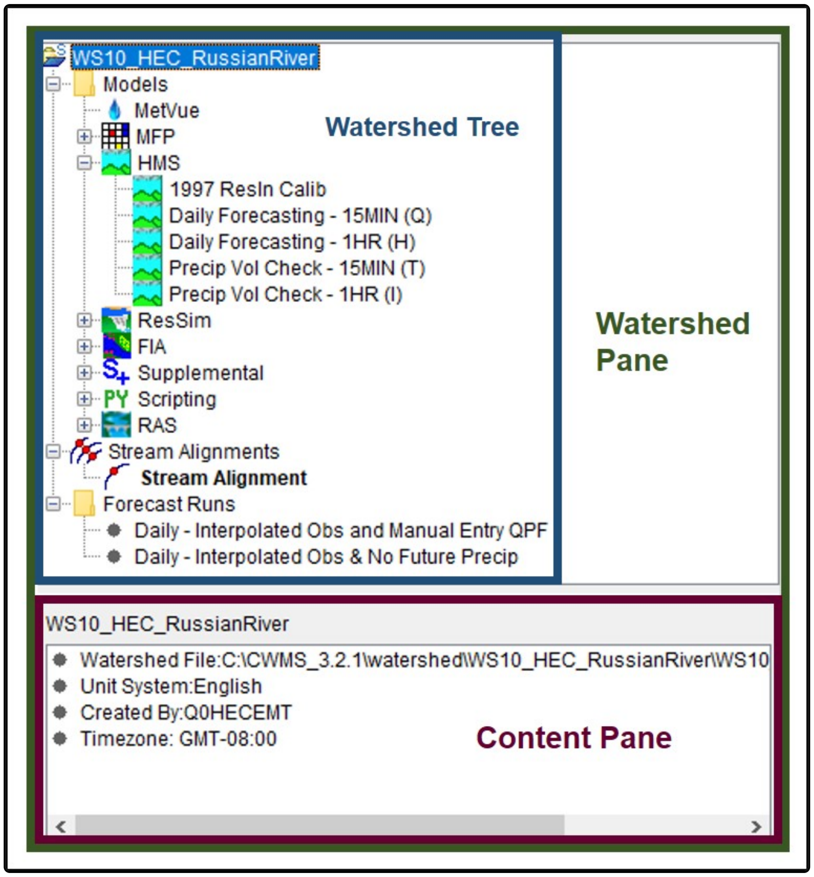
Animation Toolbar 
This toolbar is only available in the **Visualization** module, and when you have created a grid set. The **Animate** button will open the **Animate** dialog and the grid set will display in the map window.

3.4 Module Tabs

There are four modules in HEC-RTS, each module has its own unique set of functions. The four modules are – **Acquisition**, **Visualization**, **Modeling**, and **Setup**.

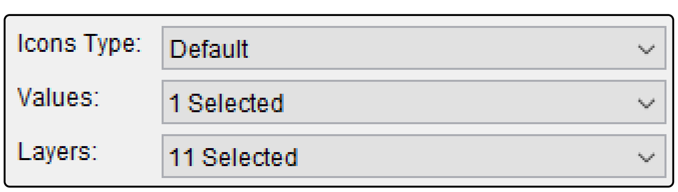
3.5 Watershed Pane

The **Watershed Pane** contains information that is pertinent to each of the HEC-RTS modules. For example, when you click the **Setup** tab (Figure 1), the **Watershed Pane**, will contain the **Watershed Tree**. Depending on what you have selected in the **Watershed Tree**, a **Content Pane** in the lower portion of the **Watershed Pane**, will display information about selected items in the **Watershed Tree**.



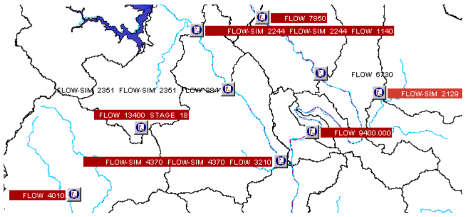
3.6 Time Series Icon Controls

Time Series Icon Controls allows you to select the attributes for the time series layers that are displayed within the watershed. The **Time Series Icon Controls** box displays in the **Watershed Pane** when the **Acquisition** and **Visualization** tabs have been selected. For the **Modeling** tab (HEC-RTS Main Window), from the **Forecast Tabs** (HEC-RTS Main Window), click the **Icon Layers** tab, the **Time Series Icon Controls** box will display in the **Watershed Pane**. If nothing displays, then you need to create a time series layer and data must be available in at least one of the time series icons associated with the time series layer. There are three attributes available - **Icons Type, Values**, and **Layers**.



Icons forces all icons within the map window to display as the selected **Icon Type** in the **Type** list. Selecting the **Default** option in the list will display the icons as they were set in the Time Series Icon Editor dialog.

Values allows you to display the time series data as labels. You will select what data to display from the **Values** list (Figure 1) and the information will display next to the associated time series icons. Multiple values can be selected at once. Setting up the color associated with a value is detailed in Time Series Icons.

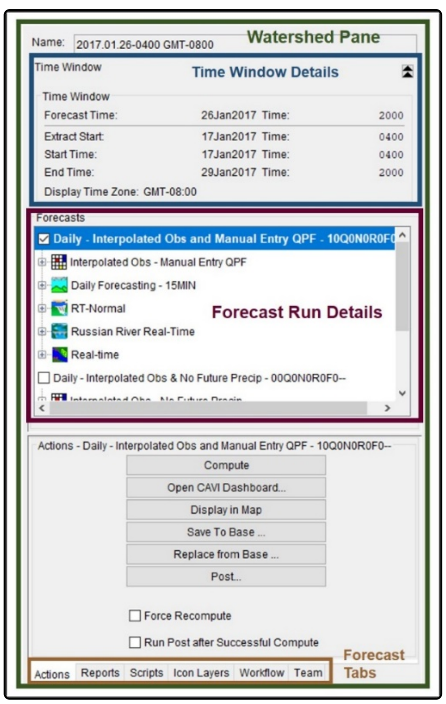


2 Figure 2 Value Labels

Layers allows you to select which layer or layers are visible in the map window. Multiple layers can be selected at once.

3.7 Forecast Tabs

When you are in the **Modeling** module, at the bottom of the **Watershed Pane** are the **Forecast Tabs**. There are six tabs - **Actions, Reports, Scripts, Icon Layers, Workflow**, and **Team**. Each tab provides you with different commands that will allow you to compute a forecast; view extract results; run scripts; select the attributes for the time series layers that are displayed within the watershed; view the workflow of the selected forecast runs; and, provides information on team modeling for the selected watershed. For example, click the **Icon Layers** tab, the **Time Series Icon Controls** display, which are also available from the **Acquisition** and **Visualization** modules.



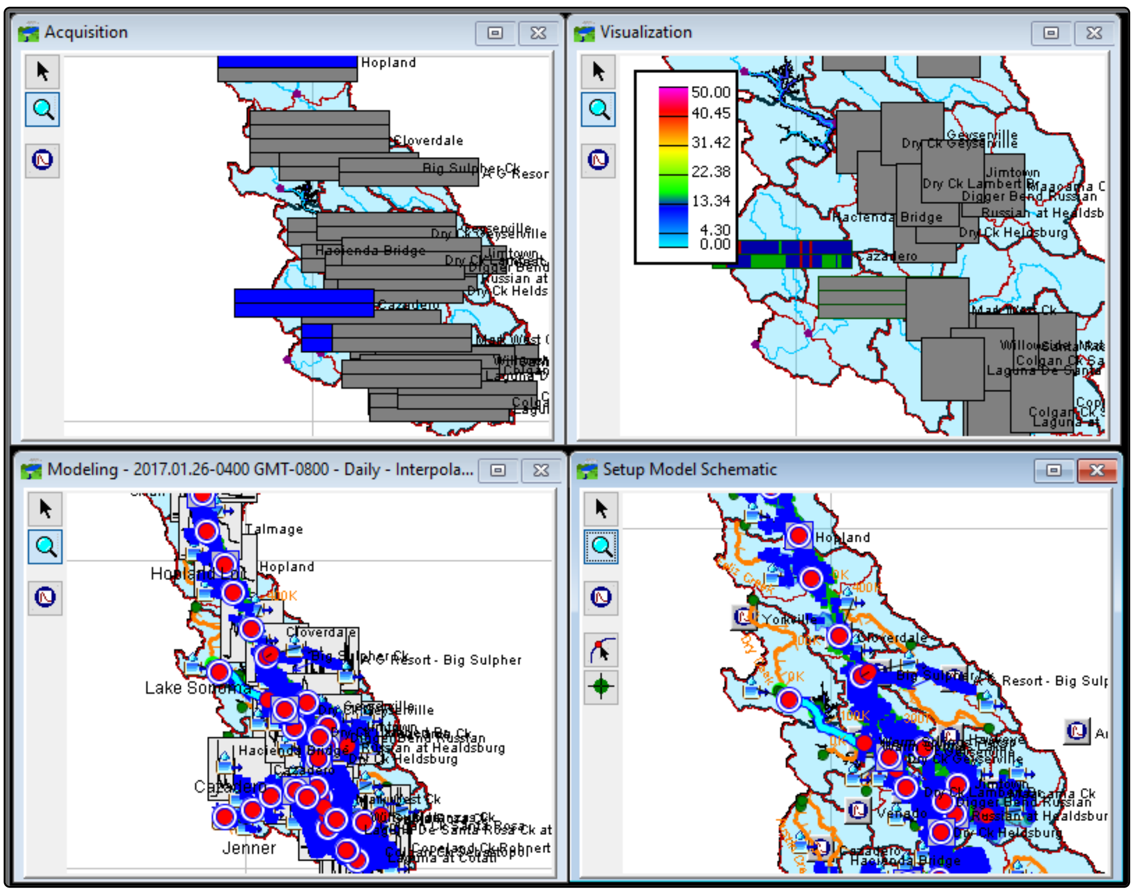
3 Figure 1 Modeling Module - Watershed Pane

3.8 Message Pane

The **Message Pane** (HEC-RTS Main Window) is a record of CWMS software activities, records the opening of a watershed, opening map windows, accessing DSS files, and many other HEC-RTS activities. System output log information displays in the **Message Pane** from the time you start HEC-RTS until you exit. Messages related to incoming data are colored coded based on the quality of the data. Messages may also report a problem with a system component; problems with field equipment; a caution, warning, or flood event; or other critical situations.

3.9 Map Window

Map windows are used to graphically display a watershed, map layers, model alternative schematics, time series icons, visualize data and grid sets, track the acquisition of data, and forecast information. Each module has its own map window, for example, in Figure 1, for the **Setup** module you have chosen to display the stream alignment and the schematic for a forecast run. From the map window for the **Setup** module you can edit elements from each of the model alternatives that are part of the selected forecast run.



4 Figure 1 Map Windows for Each Module

From the map window for the **Modeling** module (Figure 1), you are viewing the schematic for the selected forecast run, and from the map window you can view results for each of the model alternatives that are part of the forecast run.

3.9.1 Map Window Toolbar

The primary set of tools in the map window toolbar includes the **Select Tool, Zoom Tool**, and **Time Series Icon Tool**. The tools change the appearance and functionality of the mouse. Other tools appear in the toolbar depending on which module has been selected. Each tool's functions are described in context in other chapters of the manual.

Select Tool

The **Select Tool** allows you to select elements in the map window. Also, after zooming in, the user can use the **Select Tool** to pan.

Zoom Tool

The **Zoom Tool** allows you to zoom in and out of the map window. To zoom in, hold the mouse button down and outline the area that needs to be enlarged. Right-click to zoom out.

Time Series Icon Tool

The **Time Series Icon Tool** allows you to select time series icons (Time Series Icons and Time Series Icons Layers) on the map window. To select multiple icons, hold down the **SHIFT** key, and click on the time series icons that need to be selected. You can access commands related to a time series icon by right-clicking on the time series icon.

When you request to display the stream alignment in a **Setup** module map window (Figure 1), two additional tools become available that are pertinent to only the stream alignment:

Stream Alignment Tool

The **Stream Alignment Tool** allows you to create, delete, and edit stream elements that make up the stream alignment. A stream alignment is a representation of the stream network in the watershed.

Stream Node Tool

The **Stream Node Tool** allows you to create and edit stream nodes and junctions that make up a stream alignment. Stream nodes are the beginning and ending points for each stream element. Stream nodes are points where stream elements meet (stream junctions).

3.9.2 Map Window Elements

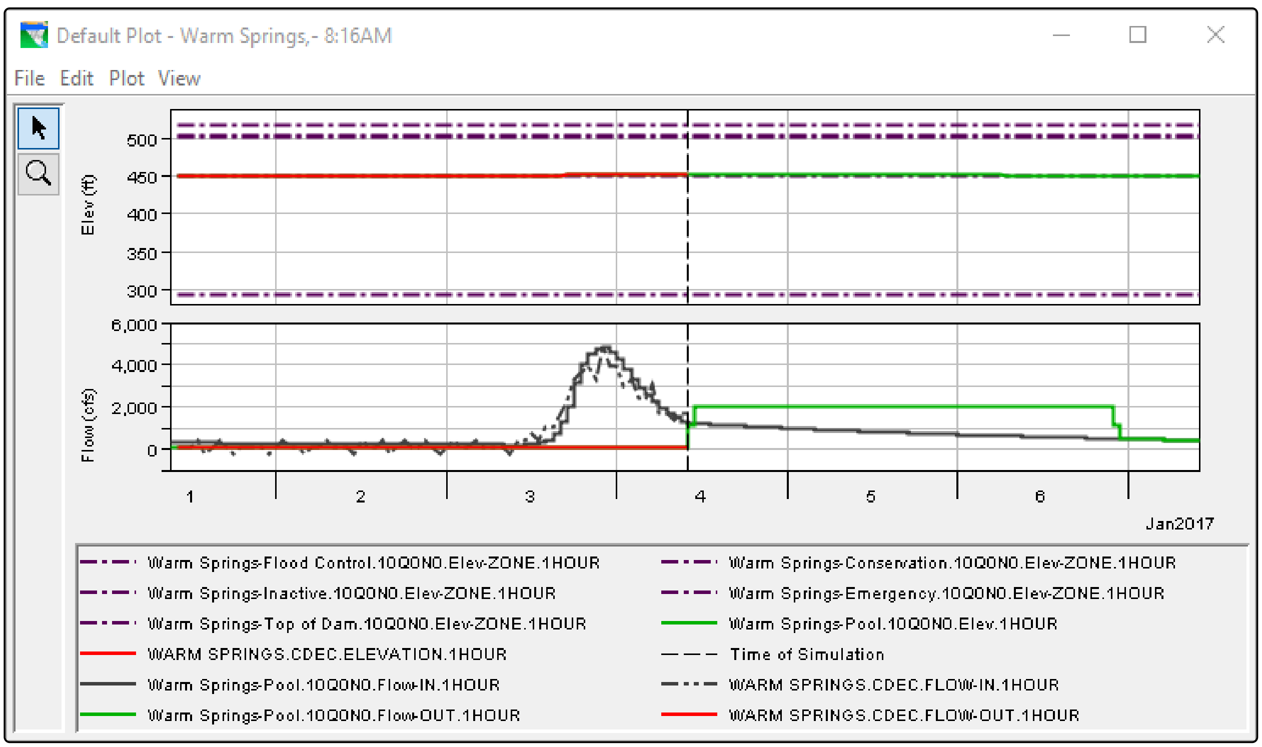
The elements of a map window represent the watershed data visually in a geo-referenced context and interact with associated data. You can create elements (stream alignment, time series icons) and edit those element's properties in the **Setup** module. Also, in the **Setup** module, if you have displayed a model alternative in the map window, you can edit elements specific for that model alternative. In the **Acquisition** and **Visualization** modules, you can plot, tabulate, and view the data associate with time series icons. For the **Modeling** module, you can plot, tabulate, and view results at time series icons and at elements that represent the model alternatives associated with a forecast run. Additional commands are available according to module and are discussed throughout this manual.

3.10 Viewing HEC-RTS Data and Results

Data can be viewed from the Acquisition and Visualization modules, from plots or tables. Results are viewed from the **Modeling** module through plots and reports (detailed information is provided in HEC-RTS Results)

3.10.1 Plots

HEC-RTS plots offer data and results specific to the active module. Data you can view was defined in the **Setup** module at time series icon locations (Time Series Icons). Modeling results can also be viewed at time series icons or at individual modeling elements (e.g., reservoirs). For example, in the **Acquisition** module, the plot for one time series icon might open a two-dimensional graph of incoming data at a specific location, while, in the **Modeling** module, the same time series icon might plot reservoir modeling results at that location (Figure 1).



5 Figure 1 Sample Plot - Reservoir Modeling Results

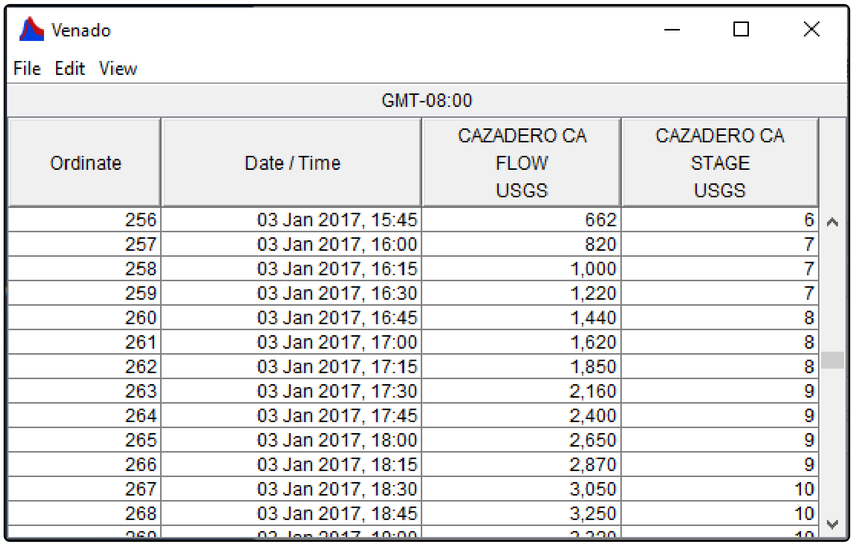
The plot displays the location name in the title bar, has axis labels, and a color-coded legend for the data contained in the plot. When a plot depicts results of a model alternative, as in Figure 1, a dashed vertical line represents the time of forecast. The **Zoom Tool** from the plot window operates the same as described in Map Window.

You can customize the appearance of plots through the use of several editors. To access these editors, use the **Select Tool** and right-click on different elements of the plot (e.g., lines, axis). You can also access these editors through the **Edit** and **View** menus in the **Plot** dialog (Figure 1). The following gives an overview of the different editors (Working with Plots provides further details on customizing plots):

Curve Properties	Right-clicking on a plot curve or point allows you to open a curve properties editor where you can edit curve colors, styles, and weights as well as labels and quality symbols.
Viewport Properties	Right-clicking on the viewport of a plot allows you to open an editor where you can customize the border, background, and gridlines of the plot.
Title Properties	Right-clicking on a plot axis allows you to open a title properties editor where you can customize the title of a plot.
Axis Properties	Right-clicking on a plot axis allows you to open an axis properties editor where you can customize the axis scale and tic marks.
Legend Properties	Right-clicking on the legend of a plot allows you to open an editor where you can add a title and add icons and text to the left and right blocks of a plot.
Label Properties	Right-clicking on an axis label or plot legend allows you to open a label properties editor where you can add or change the background color of labels or add a border to the labels.
Spacer Properties	If you have multiple plots in your plot window, you can right-click on the space between the plots to access this editor. The spacer properties editor lets you adjust the space between plots.
Polygon Properties	Right-clicking on a polygon allows you to open a polygon properties editor where you can customize borders and backgrounds.

3.10.2 Tables

The same data and information viewed from a plot can also be viewed in tabular form (Figure 2). As with plots, the type of data and results displayed depends on the properties you have defined for each time series icon.
You can customize the appearance of tables through the use of commands available from the menu bar of the table dialog. You can add commas, separate date/time into two separate columns, make the year four digits, specify the decimal precision of the data, show the quality of the data, show missing data as a value (e.g., -901.0), copy selected cells or the whole table, print, plot, resize the columns of the table, and export the table to an ASCII file.

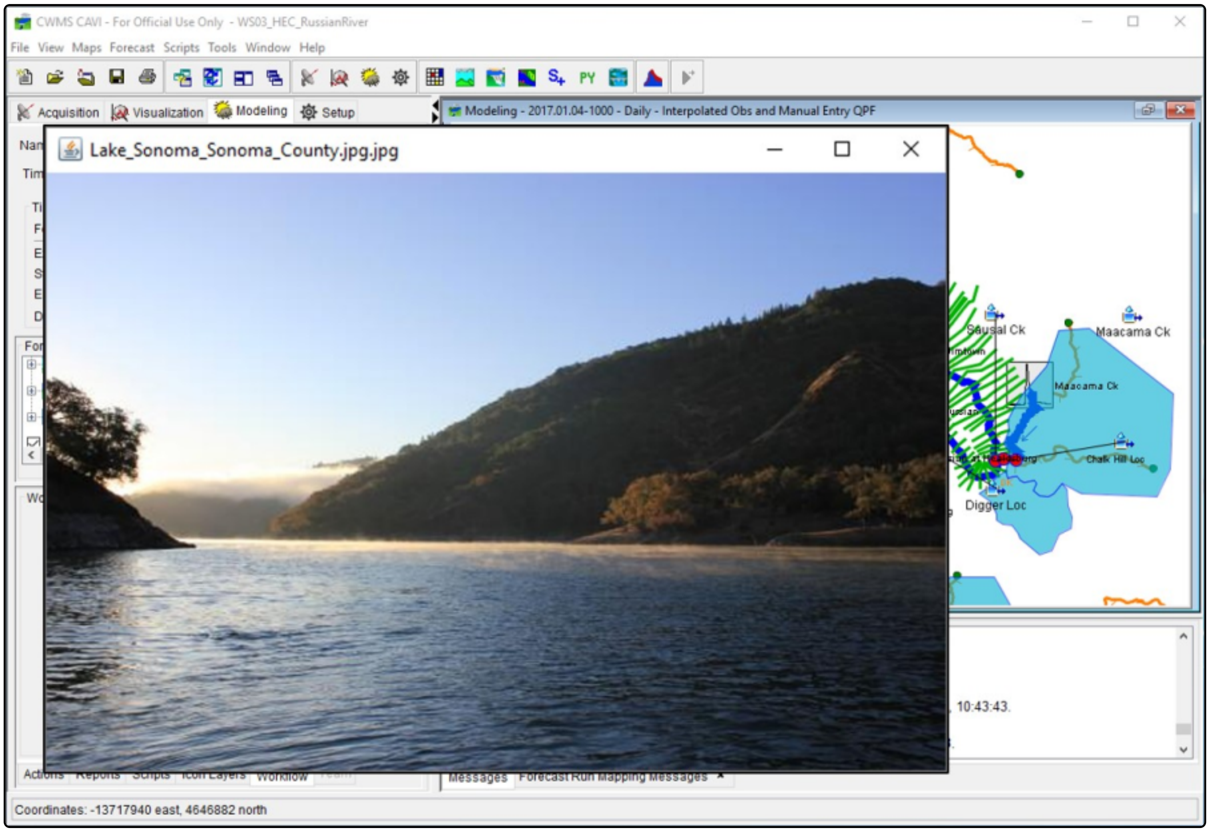


Ordinate	Date / Time	CAZADERO CA FLOW USGS	CAZADERO CA STAGE USGS
256	03 Jan 2017, 15:45	662	6
257	03 Jan 2017, 16:00	820	7
258	03 Jan 2017, 16:15	1,000	7
259	03 Jan 2017, 16:30	1,220	7
260	03 Jan 2017, 16:45	1,440	8
261	03 Jan 2017, 17:00	1,620	8
262	03 Jan 2017, 17:15	1,850	8
263	03 Jan 2017, 17:30	2,160	9
264	03 Jan 2017, 17:45	2,400	9
265	03 Jan 2017, 18:00	2,650	9
266	03 Jan 2017, 18:15	2,870	9
267	03 Jan 2017, 18:30	3,050	10
268	03 Jan 2017, 18:45	3,250	10

6 Figure 2 Data in Tabular Form

3.10.3 Photos and Webcam Images

Photos, webcam images, web pages, or documents can be assigned to time series icons. By adding them to an icon in the **Setup** module, you can access them in the **Visualization, Acquisition**, and **Modeling** modules through the icon shortcut menu. Images and webcams can be viewed in their own separate dialog or as icons on the map window. Figure 3 displays an image that can be viewed from a time series icon. If a time series icon represents a dam, links can be assigned to the icon that could be a webcam that displays a real-time video of an actual outlet on that dam. The time series icon could also have access to pictures of the dam and reports providing details about the dam. For more information on how to set these features up see Time Series Icons.



7 Figure 3 Viewing an Image from a Time Series Icon (https://www.sonomacounty.com/outdoor-activities/lake-sonoma-recreation-area)

3.10.4 Scripts

Scripts can be assigned to individual time series icons. The scripts can be used to update the data at given intervals, compute other information from the data, or complete other tasks. Scripts are assigned to the icons in the **Setup** module, through the **Time Series Icon Editor**. They may also be created in the **Script Editor**. For more detailed information on assigning and using scripts, see Scripting.

4 Understanding Layers

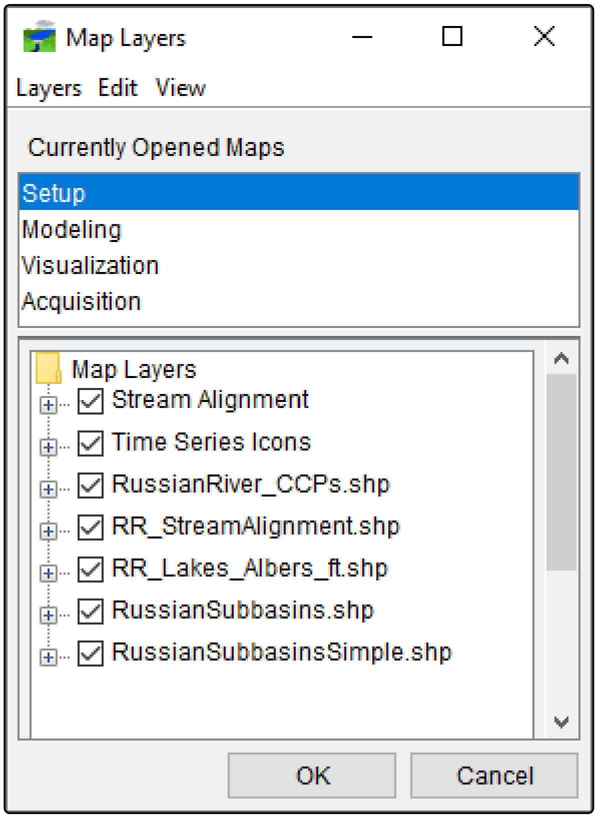
Layers in the HEC-RTS interface are like transparencies laid one on top of the other. Each includes representations of physical images such as roads, county and state boundaries, rivers, and subbasins. These are layered in the map window as color pictures. Each of these images, along with its associated data, is a layer. This chapter describes how to create and manage layers so that required data and information are easily viewed.

There are two types of layers: primary layers and map layers. Primary layers are automatically generated when a new watershed is created and are present in all HEC-RTS watersheds (although some primary layers may be empty in some watersheds) and are module specific. HEC-RTS creates primary layers when a stream alignment, time series icons, map window elements, and gridded precipitation data are created. (A time series icon is a symbol in the map window that provides quick access to time-series data and information that has been assigned to the icon.) HEC-RTS also creates a primary layer when a forecast alternative has been selected for computation. Primary layers can contain sub-layers, for example, the Time Series Icons Layer can have several user-defined sub-layers that represent various types of data and information such as flow and precipitation.

Map layers are optional and are loaded into the watershed by the user. Map layers are physical images, which help visualize a watershed. In HEC-RTS there are several formats for map layers, and include maps from the Internet, such as a Google street map. With the **Display Map Coordinates** function in the **Maps** menu, the user can set which coordinates the **Map Window** will use (watershed or loaded Internet map).

4.1 Map Layers Dialog

The **Map Layers** dialog allows the user to add map layers, organize layers, view all the layers associated with the watershed, and add sub-layers to primary layers. The **Map Layers** dialog can be accessed from any of the modules. From the HEC-RTS main window, from the **Maps Menu**, click **Map Layers**. The **Map Layers** dialog will open. The following sections describe the commands available in the **Map Layers** dialog.



8 FIgure 1 Map Layers Dialog

4.1.1 Menu Bar

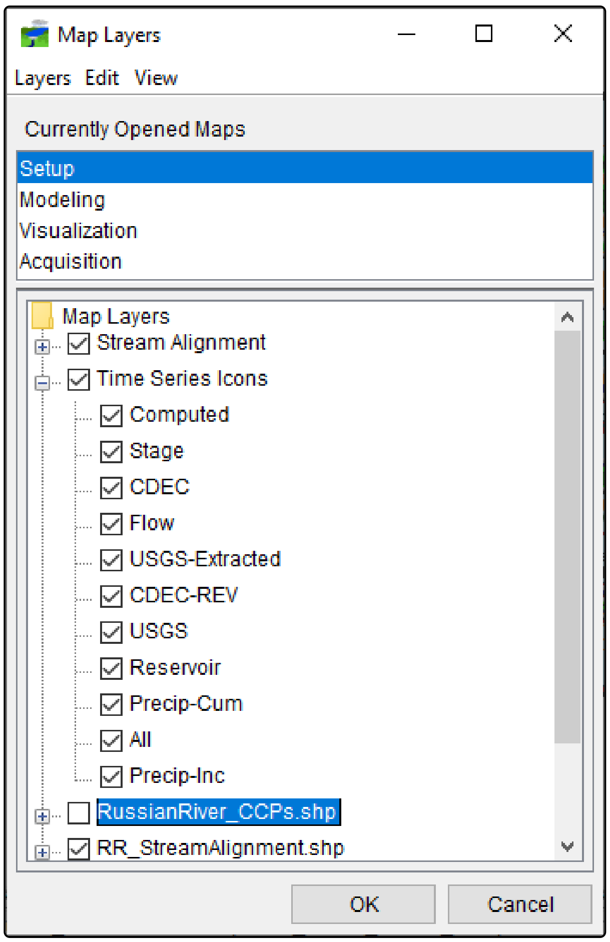
The menu bar of the **Map Layers** dialog contains commands for managing watershed layers. The following is a summary of the available menus and the commands associated with each one.

Layers	From this menu, you can add map layers; import images; remove map layers; and, close the **Map Layers** dialog.
Edit	From this menu, based on your selection in the **Layer Tree** (Figure 1), you can rearrange layers, access each layer's properties, and, other items specific to the selected layer.
View	This menu has one command, **Always On Top**, which keeps the **Map Layers** dialog on top of the HEC-RTS main window.

4.1.2 Layer Tree

The **Map Layers** dialog contains a tree (**Layer Tree**) for controlling the display of layers in a watershed. The **Layer Tree** represents the hierarchical arrangement of the available layers (primary, map). With the top level of the tree being the **Map Layers** folder (Figure 1) which contains all layers in the watershed. Beneath the **Map Layers** folder is a branch for each of the identified layers in a watershed - primary or map layer. To expand a branch, click the plus (+) sign, depending on the layer type you might see sub-layers appear or a legend for the layer appears (Figure 2). When a minus sign (collapses) is displayed, clicking the minus sign collapses the branch. By double-clicking on a layer, the user can expand/collapse the branch and open the selected layer's **Properties Editor**.

From the **Layer Tree** (Figure 2), the checkbox turns the display of a layer or sub-layer on or off. If a layer is unchecked, any sub-layers associated with that layer will not display, even if the sub-layers are checked.



9 Figure 2 Map Layers Dialog - Setup Module - Expanded Layer

4.1.3 Layer Organization

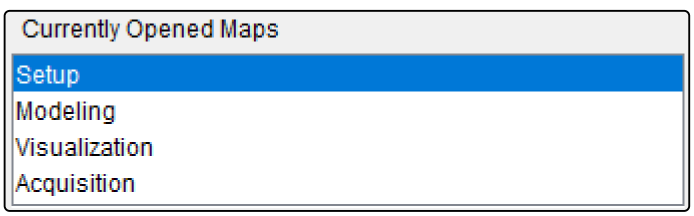
Layers in a **Map Window** can be organized with the **Map Layers** dialog. By default, the primary layers are at the top of the layer tree, while map layers are added to the bottom of the layer tree. Typically background images (map layers) are on the bottom, and layers containing smaller, more-detailed items such as time series icons, gages, and rivers are on top. Layers are drawn in the order that they are displayed in the **Map Layers** dialog. You might want to rearrange the order of layers, because either you cannot view a layer you want to display, or you might need to "hide" a layer.

To rearrange the order of layers:

1. From the **Map Layers** dialog (Figure 2), from the **Layer Tree**, select the layer that you wish to rearrange. From the **Edit** menu or by right-clicking on the selected layer, a shortcut menu will display. From either menu, choose **Move Up**, **Move Down**, **Move to Top** (moves the selected layer to the top of the layer tree), or **Move to Bottom** (moves the selected layer to the bottom of the tree).
2. Continue to change the position of the selected layer until the layer is in the desired position. Click **OK,** and the **Map Layers** dialog will close.

4.2 Module Map Windows

For each HEC-RTS module, there is an associated map window, and for each map window, the layers available are different. When you click on a module tab, in the HEC-RTS main window, a **Map Window** will display. From the **Map Layers** dialog, from the **Currently Opened Maps** box, the currently opened module map windows are listed. Selecting a module, will display the associated layers with that module in the **Layer Tree**.



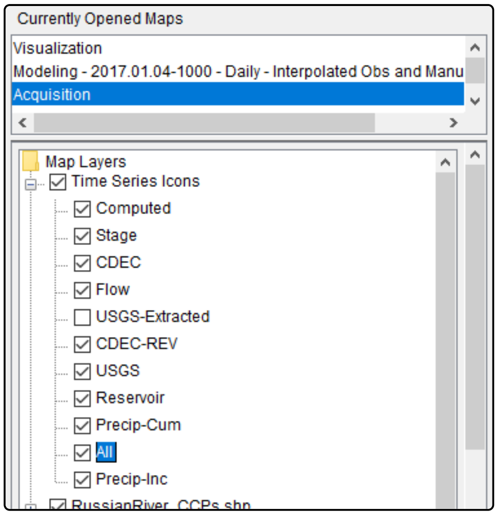
10 Figure 1 Map Layers Dialog - Currently Opened Maps Box

4.3 Layer Visualization Scaling

Layers may be set to become visible or invisible as zooming in or out occurs in a map window (Primary Layers and Map Layers). This allows you to display finer levels of detail as you zoom in on regions in a map window and avoid clutter when displaying the full area. This might include time series icons not appearing for less significant gages, detailed map layers becoming visible, and raster or other images appearing that are not as distinguishable at a coarser zoom level.

4.4 Time Series Icons Layer

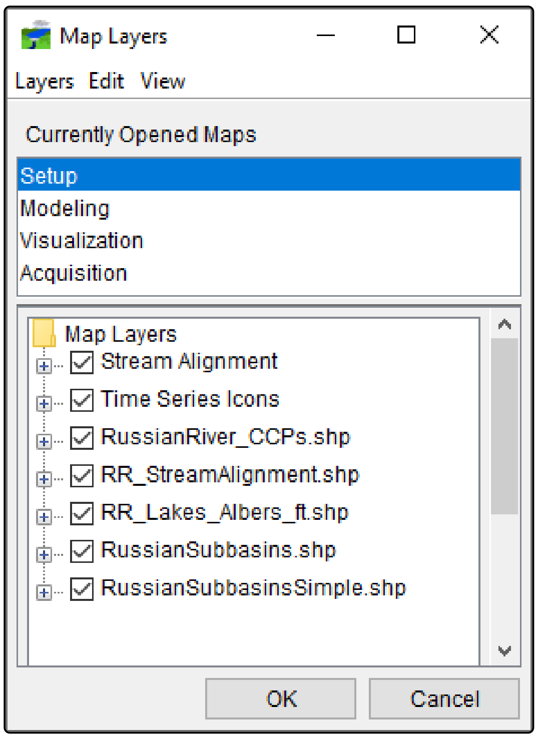
For the **Visualization**, **Acquisition**, and **Modeling** modules, you can change the order of the sub-layers in the primary layer **Time Series Icons** (Figure 1). Expand the **Time Series Icons** layer, all sub-layers are now visible within the map window quickly. You can set specific sub-layers to display, edit the properties of a sub-layer, and set visualization scaling. Further details on setting up sub-layers under the **Time Series Icons** layer is available in Time Series Icons Layers.



11 Figure 1 Map Layers Editor - Acquisition Module - Time Series Icons Layer

5 Primary Layers

Primary Layers are created when a watershed is created (**Time Series Icons** layer); when a stream alignment is created (**Stream Alignment** layer); when a grid set is created (**Gridded Data** layer); and, when forecast alternatives and forecasts are created (**Model Schematic** layer). Primary layers are present in all HEC-RTS watersheds (although some layers may be empty in some watersheds) and are foundational to HEC-RTS (whereas map layers are optional, see Map Layers). This chapter describes how primary layers are configured and managed. The primary layers display in the **Map Layers** dialog (Figure 1) and are usually listed at the top of the **Layer Tree** (Understanding Layers contains further information on the **Map Layers** dialog).



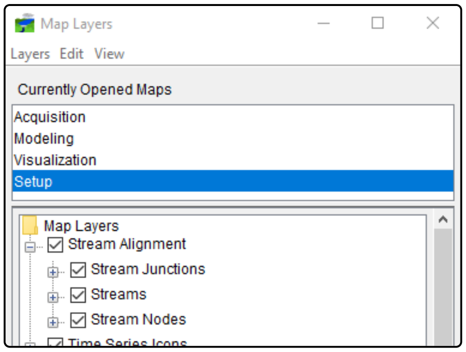
12 Figure 1 Map Layers Dialog

5.1 Time Series Icon Layer

The **Time Series Icons** layer is created for the **Acquisition**, **Visualization**, and **Setup** modules when an HEC-RTS watershed is created. At first, this layer is blank, but as you create time series icons, the **Time Series Icons** layer includes all of the time series icons that you have defined for your watershed. Time series icons are available in the modules **Acquisition**, **Visualization**, and **Setup** modules, once forecast alternatives and forecasts have been created then the **Time Series Icons** layer will display in the **Modeling** module.

5.2 Stream Alignment Layer

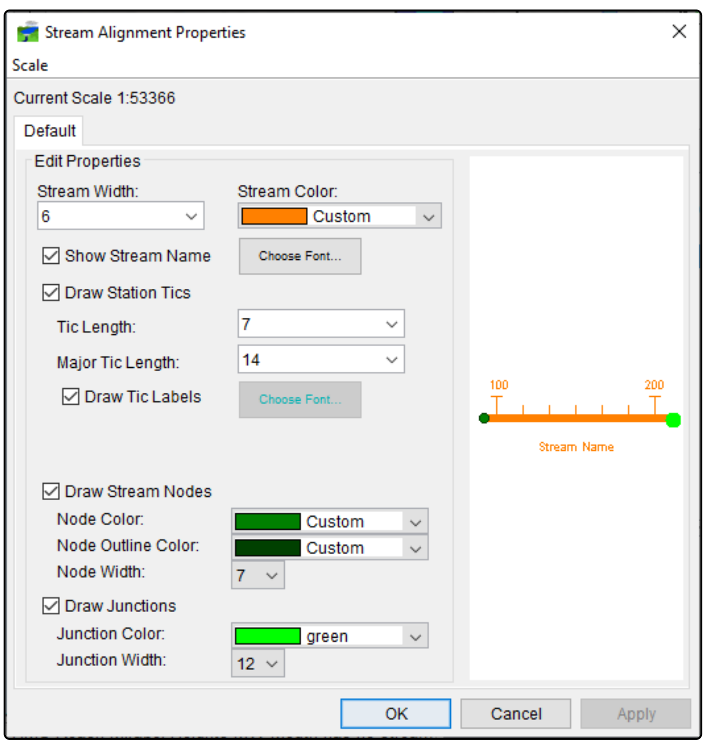
The **Stream Alignment** layer is available in the **Setup** module (also is available in the **Modeling** module when a forecast is open) once a stream alignment has been defined for a watershed (Appendix J). This layer has sub-layers for each stream alignment element. Expanding the **Stream Alignment** layer node in the **Map Layers** dialog (Figure 1) displays its sub-layers (**Stream Junctions**, **Streams**, and **Stream Nodes**).



13 Figure 1 Map Layer Dialog - Stream Alignment Layer

5.2.1 Stream Alignment Layer Properties

You can change the appearance of streams and their labels; change the appearance of stream junctions and stream nodes; and, set the visualization scale for the **Stream Alignment** layer. From the **Map Layer** dialog (Figure 1), right-click on **Stream Alignment**, from the shortcut menu, click **Properties**. The **Stream Alignment Properties** dialog will open (Figure 2).

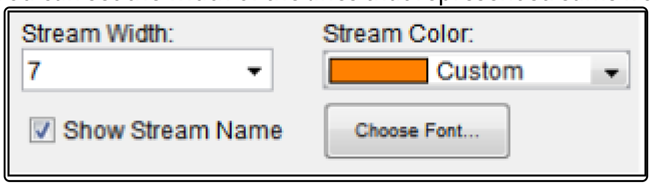


14 FIgure 2 Stream Alignment Properties Dialog

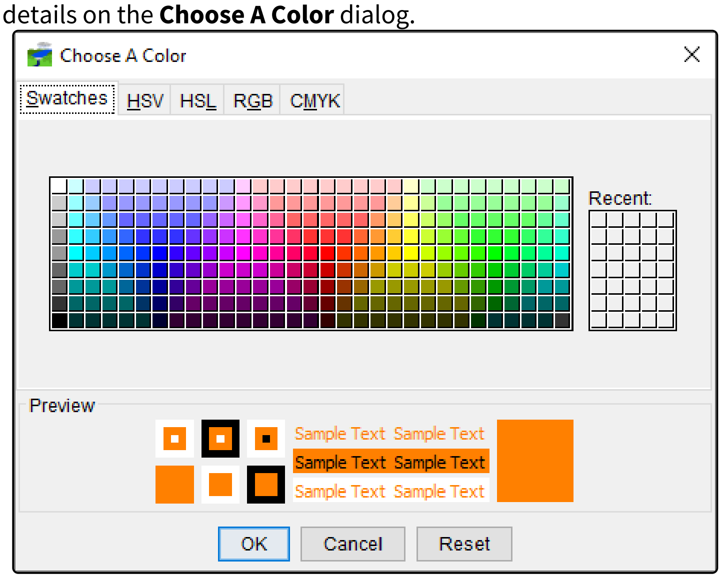
5.2.2 Stream Alignment – Streams

The following are instructions for changing the appearance of **Streams** in the stream alignment:

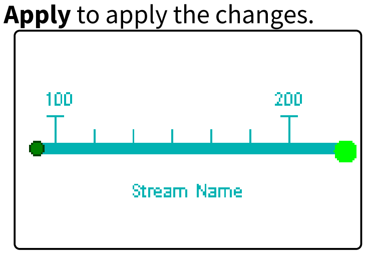
1. You can set the width of the lines that represent streams from the **Stream Width** list (Figure 3).



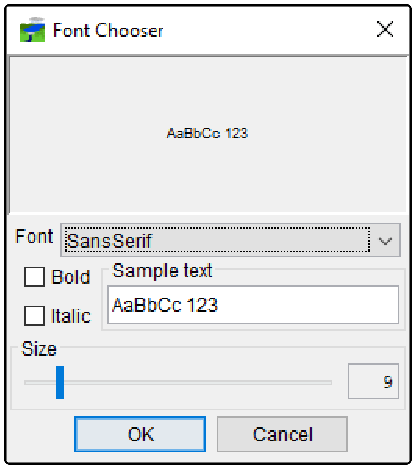
2. You can select the color of the lines that represent the streams from the **Stream Color** list (Figure 2). The default color is orange. There are several colors to choose from, and you can also create custom colors. From the **Stream Color** list, click **Custom**. The **Choose A Color** dialog will open (Figure 4). See Appendix E for further details on the **Choose A Color** dialog.



3. A preview of the stream color change is displayed (Figure 5) in the **Stream Alignment Properties** dialog. Click **Apply** to apply the changes.

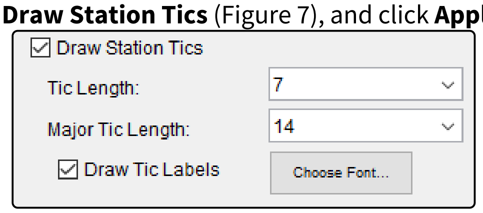


4. By default, stream labels appear in the map window. If you do not want the labels displayed, from the **Stream Alignment Properties** dialog (Figure 2), uncheck **Show Stream Name**, and click **Apply**.
5. To change the font of the stream labels, from the **Stream Alignment Properties** dialog (Figure 2), click **Choose Font**. The **Font Chooser** dialog will open (Figure 6). Select the font type from the **Font** list, and select a size using the **Size** slider. Click **OK**, the **Font Chooser** dialog will close. Click **Apply** on the **Stream Alignment Properties** dialog (Figure 2). All the stream labels will appear in the font and size you choose.



Tic marks appear on a stream representing the stream stationing. There are two types of tic marks, minor and major. The following are instructions for changing the appearance of tic marks:

6. If you do not want to display the tic marks, from the **Stream Alignment Properties** dialog (Figure 2), uncheck **Draw Station Tics** (Figure 7), and click **Apply**.

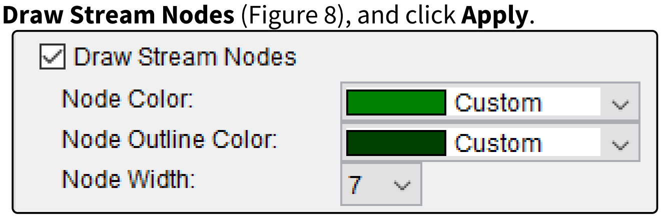


7. From the **Tic Length** list (Figure 7), you can set the length of the minor tic marks. To change the length of the major tic marks, use the **Major Tic Length** list. Click **Apply**, and the map window displaying the stream alignment will update.
8. To change the font of the tic mark labels, from the **Stream Alignment Properties** dialog (Figure 2), click **Choose**. The **Font Chooser** dialog will open (Figure 6). Select the font type from the **Type** list, and select a size using the **Size** slider. Click **OK**, the **Font Chooser** dialog closes (Figure 6). From the **Stream Alignment Properties** dialog (Figure 2), click **Apply**. All the tic mark labels will appear in the font and size you chose.

5.2.3 Stream Alignment – Stream Nodes

The following are instructions for changing the appearance of all **Stream Nodes** in the stream alignment:

1. If you do not want to display stream nodes, from the **Stream Alignment Properties** dialog (Figure 2), uncheck **Draw Stream Nodes** (Figure 8), and click **Apply**.

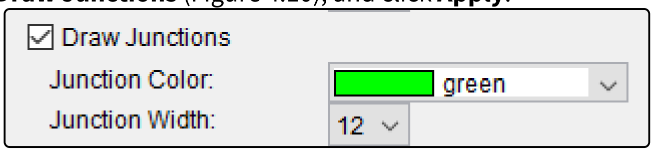


2. You can select the color for stream nodes from the **Node Color** list (Figure 8). The default color is green. Similar to streams, you can also create a custom color.
3. You can select the outline color of stream nodes from the **Node Outline Color** list (Figure 8). The default color is dark green. Like streams, you can also create a custom color.

4. You can set the width of stream nodes from the **Node Width** list (Figure 8).

5.2.4 Stream Alignment – Junctions

The following are instructions to change the appearance of all **Stream Junctions** in the stream alignment:
1. If you do not want to display stream junctions, from the **Stream Alignment Properties** dialog (Figure 2), uncheck **Draw Junctions** (Figure 4.10), and click **Apply**.

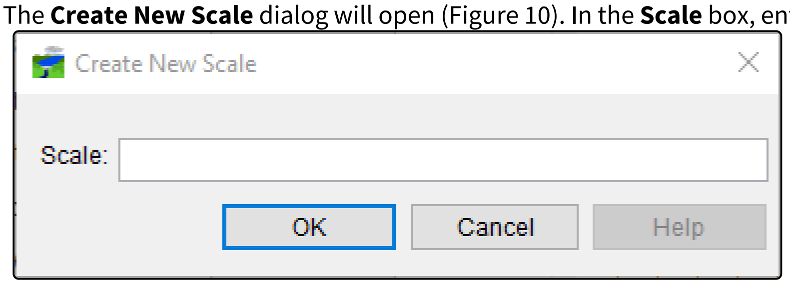


2. You can select the color of stream junctions from the **Junction Color** list (Figure 9). The default color is dark green. Like streams, you can also create a custom color.
3. You can set the width of Stream Junctions from the **Junction Width** list (Figure 9).

5.2.5 Stream Alignment - Scale

The visualization **Scale** feature allows you to customize stream alignment display settings as you are zooming in and out on the map window. The following are instructions to set the visualization scale for the **Stream Alignment** layer:

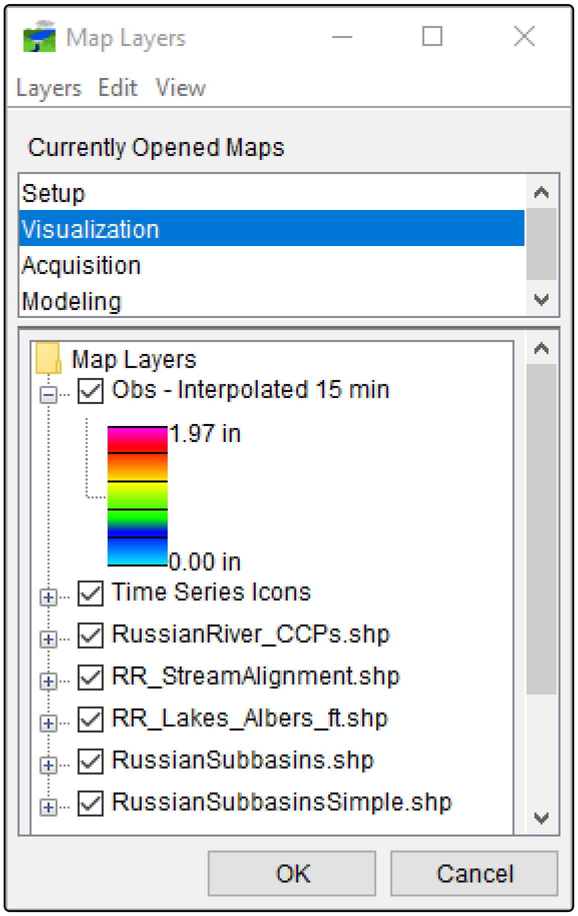
1. To set the visualization scale for the stream alignment to a default, from the **Stream Alignment Properties** dialog (Figure 2), click **Scale**, click **Set to Defaults**. The visualization scale is set automatically.
2. To enter your own scale factor, from the **Stream Alignment Properties** dialog (Figure 2), click **Scale**, click **New**. The **Create New Scale** dialog will open (Figure 10). In the **Scale** box, enter values for the scale factor.



3. Click **OK**. This creates a new tab in the **Stream Alignment Properties** dialog (Figure 2), from the new tab, you can configure the stream alignment's appearance.

5.3 Gridded Data Layer

The **Gridded Data** layer (Figure 1) is available in the **Visualization** and **Modeling** modules, only when gridsets are included or computed for the watershed. A **gridset** is a sequence or time series of precipitation grids. The **Gridded Data** layer contains the gridsets for the watershed that have been created outside of HEC-RTS using the HEC-GridUtil or HEC-MetVue software.

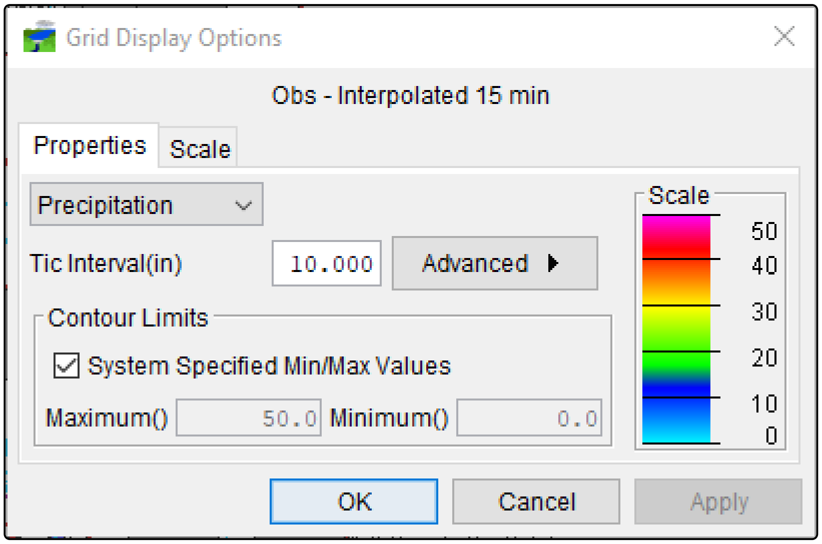


15 Figure 1 Gridded Data Layer

5.3.1 Gridded Data Layer - Properties

You can change the appearance of a girdset and set the visualization scale for the **Gridded Data** layer. From the **Map Layer** dialog (Figure 1), right-click on the name of the gridset (*Obs – Interpolated 15 min*), from the shortcut menu, click **Properties**. The **Grid Display Options** dialog will open (Figure 2).

From the **Grid Display Options** dialog (Figure 2) you can choose a color contour scheme (default is **Precipitation**), tic interval, and override the minimum/maximum values for the contour limits. There are several advanced features. These features are accessed by clicking **Advanced**, which expands the **Grid Display Options** dialog (Figure 2). The advanced options are:



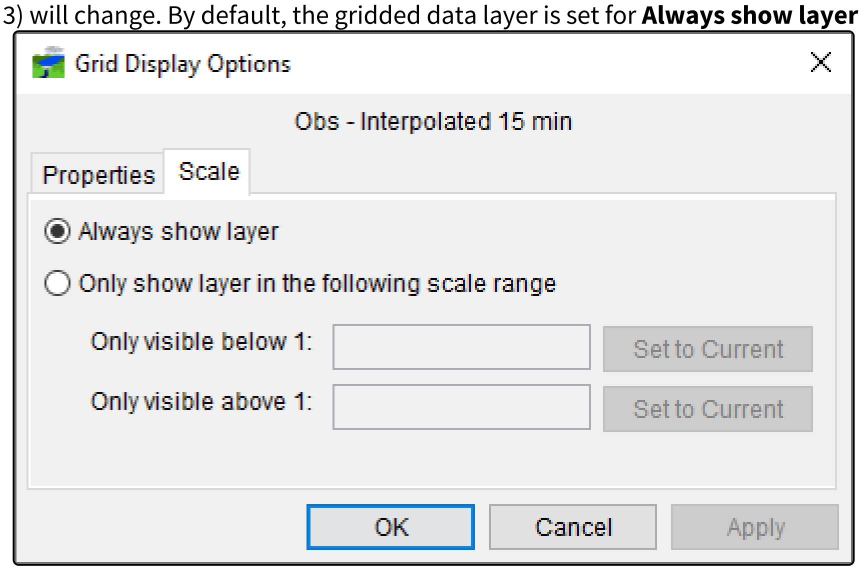
16 Figure 2 Grid Display Options Dialog

- **Brightness**: The relative lightness or darkness of a color, from full color to black.
- **Saturation**: The relative intensity of the color, from white to full color.
- **Transparency**: The relative opacity of the color from opaque to translucent to transparent.
- **Aspect Shading**: Enabling this feature causes the display color to change based on the viewing angle.
- **Adjust Color Scale to Clipping Area**: Enabling this feature causes the displayed color ramp to rescale to the maximum and minimum clipping thresholds.

5.3.2 Gridded Data Layer - Scale

To set the visualization scale for the **Gridded Data** layer:

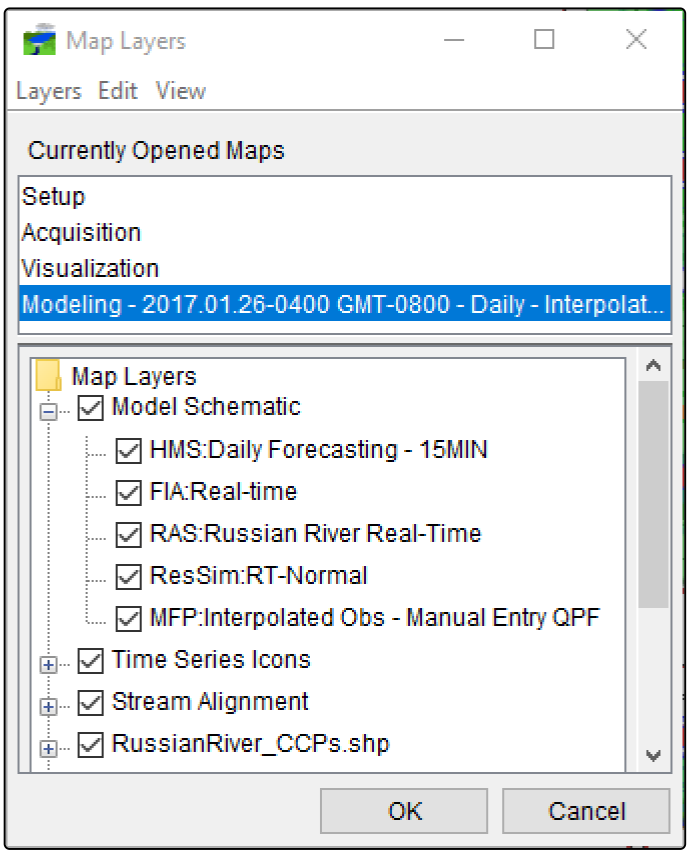
1. From the **Grid Display Options** dialog (Figure 2), click on the **Scale** tab, the **Grid Display Options** dialog (Figure 3) will change. By default, the gridded data layer is set for **Always show layer** (Figure 3).



2. To set visualization scales click **show layer in the following scale range** (Figure 3). To set the scale so that the gridded data layer becomes visible as you zoom-in, you can set the scale to the current scale by clicking **Set to Current**. If you wish to enter a value, in the **Only visible below 1:** box (Figure 3) enter the scale factor.
3. To set the scale so that the **Gridded Data** layer becomes visible as you zoom-out, you can set the scale to the current scale by clicking **Set to Current**. If you wish to enter a value, in the **Only visible above 1:** box (Figure 3) enter the scale factor.
4. Click **OK**, the **Grid Display Options** dialog closes (Figure 3), and the visualization scale is set for the gridded data layer.

5.4 Model Schematic Layer

Once you have created forecast alternatives, created a forecast, and have made the forecast active, the **Model Schematic** layer (Figure 1) will display in the **Map Layers** dialog for the **Modeling** module. The **Model Schematic** layer contains graphic representations of computational models for the active forecast alternative. HEC-RTS Forecasts provides detailed information on forecast alternatives, forecasts, and the different computational models represented in the **Model Schematic** layer.



17 Figure 1 Model Schematic Layer

When you expand the tree, you can see the different sub-layers for some of the models that are part of a forecast. For example, if you did not want to see the impacts for the HEC-FIA model, from the **Map Layers** dialog, from the **Map Layers** tree, expand **Model Schematic** (Figure 1). Clear (uncheck) *FIA:Real-time*, and the impact areas defined for the

HEC-FIA model will no longer display in the map window. To view the impact areas, click *FIA:Real-time*. This procedure applies to all of the sub-layers.

5.5 Shortcut Menu for Primary Layers

This shortcut menu is accessed by a right-click on primary layers in the **Map Layers** dialog (Map Layers Dialog). The shortcut menu provides several ways to manipulate the primary layers. The available commands are:

Move to Top	Move the layer to the top of the tree.
Move Up	Move the layer up the tree one position.
Move Down	Move the layer down the tree one position.
Move to Bottom	Move the layer to the bottom of the tree.
Properties	Opens a properties dialog associated with the selected layer. Depending on the selected layer, the properties dialog may be editable or not. See Appendix D for further information on the available map editors.
Import	This command is only available for the **Stream Alignment** layer. **Import** allows you to import a stream alignment from a shapefile (**.shp*); opens the **Import Stream Alignment** dialog.
Export	This command is only available for the **Stream Alignment** layer. **Export** allows you to export the stream alignment for a watershed to a shapefile (**.shp*); opens a Save browser.
Add Icon Layer	This command is only available for the **Time Series Icons** layer. **Add Icon Layer** allows you to create a time series icon layer; opens the **New Time Series Icon Layer** dialog.
Show Legend	This command is only available for the **Gridded Data** layer. **Show Legend** allows you to display the legend for a gridded dataset in the **Map Layers** dialog.

6 Map Layers

To help visualize a watershed, HEC-RTS has the capability to display various maps of watershed features. These various maps are called **map layers**, which are displayed in HEC-RTS as color pictures, layered one over the other like transparencies. The map layers (optional) also provide a geographical reference for the watershed. This chapter describes the formats of map layers and shows how the layers are configured and managed.

6.1 Map Layer Formats

Digital maps (GIS layers) are referred to as **map layers** in HEC-RTS. When a map layer is included in a watershed, the software displays the map layer as GIS features, points, lines, and polygons. The map layer formats that HEC-RTS can display are listed in Table 1.

Table 1 Map Layer Formats Supported by HEC-RTS

Description	**Common Filename Extension**
Arc shapefile	.shp
AutoCAD DXF	.dxf
Raster Image (*.img)	.png, .gif, .jpg, .bmp
Virtual Format	.vrt
GeoTIFF	.tif
Mr SID	.sid
Open Street Maps	.osm
USGS DLG	.dlg
USGS DEM	.dem, .flt
SMRMP	.smrmp
ASCII NetTIN	.net

ArcInfo® DEM	.asc

6.1.1 Arc Shapefiles (.shp)

This file type is the native data structure for the ArcGIS® program. Shapefiles store non-topological geometry and attribute information for the spatial features of a data set in a particular format. Usually there are three component files associated with a shapefile: *.dbf*, *.shp*, and *.shx*. The *.shp* file contains the shapes (points, polylines, or polygons) that are displayed in the layer.

Each shape has a record in the *.dbf* file containing descriptive data called "attributes". The *.shx* file contains an index that links each shape to its record in the *.dbf* file. For more information on shapefiles, see the *ESRI Shapefile Technical Description* available here: http://www.esri.com/library/whitepapers/pdfs/shapefile.pdf

6.1.2 AutoCAD® DXF (.dxf)

A Drawing Interchange File (*.dxf*) is created by a Computer-Aided Design (CAD) package, AutoCAD®. This format is a tagged-data representation of an AutoCad® drawing file. Tagged data refers to each data element being preceded by a group code. A group code is an integer indicating the type of data element. The file created is a vector-based graphic. The DXF format is commonly used for data exported from other CAD and GIS programs.

6.1.3 Raster Image

The raster image formats supported by HEC-RTS are: JPEG format (*.jpg* or *.jpeg*), Graphics Interchange Format (*.gif*), Portable Network Graphic (*.png*), bitmap (*.bmp*), and Multi-resolution Seamless Image Database (MrSID or *.sid*). This *.img* file should not be confused with the format for raster images used by the ERDAS Imagine image processing program. HEC-RTS cannot display files in the ERDAS Imagine *.img* format.

6.1.4 Virtual Format (.vrt)

The **.vrt* format is a format driver for GDAL (Geospatial Data Abstraction Library) and allows for a virtual GDAL dataset to be composed of other GDAL datasets. The primary use of this format is to group together a series of grids that should be associated together. This file format allows multiple depth grids for the same event to create a mosaic single grid. Relative file paths are saved in an XML format, as well as a histogram describing the frequency of cell values across all grids within the **.vrt*.

6.1.5 GeoTIFF (.tif)

A metadata standard which allows geo-referencing information to be embedded into a TIF (Tagged Image File Format) file. The added metadata could be map projection datums, coordinate systems, and any other information that will establish spatial references. A TIF file stores raster graphic images and data within a single file.

6.1.6 Mr SID (.sid)

The MrSID® (multi-resolution seamless image database) is a propriety geo-referenced image file format developed by LizzardTech® for use with. This format compresses large raster images files like aerial photographs or satellite imagery for easier viewing. Some MrSID images contain geo-referencing and positioning information and can be placed in the watershed (from the **Map Layers** dialog, from **Layers**, click **Import Image(s)**).

6.1.7 OpenStreetMap (.osm)

This is an XML file created in the OpenStreetMap (OSM) file format, which is used for saving street map information. This file is generated by HEC-RTS in the watershed's *map* folder and should not be placed or edited by the user.

6.1.8 USGS Digital Line Graph (.dlg)

U.S. Geological Survey (USGS) Digital Line Graphs (DLGs) are created from two sources using manual and automated digitizing methods. The two sources are aerial photographs or cartographic sources. DLG files are vector representations of the data. When HEC-RTS interacts with a *.dlg* file it automatically creates a *.dlgbin* file for use.

6.1.9 USGS DEM (.dem)

USGS publishes the file format USGS DEM. A raster file format, *.dem* files usually contain terrain elevations, but the file format can be used for representing any surface. The format allows for internal documentation of the coordinate system, date of publication, etc. These values are referenced horizontally either to a Universal Transverse Mercator (UTM) projection or to a geographic coordinate system.

6.1.10 SMRMP (.smrmp)

Segmented Multi-Resolution Multi-Parameter (SMRMP) file (HEC-FIA) created to operate gridded data (namely, digital terrain grids, inundation depth grids, and arrival time grids) more efficiently.

6.1.11 ASCII NetTIN (.net)

This is an ASCII file format used to represent a triangulated irregular network (TIN). TIN data are stored in a simple format that defines the nodes, edges, and triangles contained in the TIN. This format is common for transferring TIN data between software.

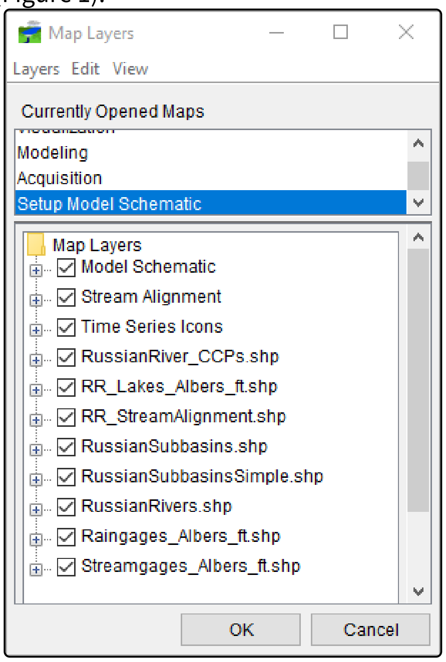
6.2 Adding Map Layers

You can add as many types of map layers as needed. In HEC-RTS Version 3.1.1, HEC-RTS can transform coordinates "on-the-fly" as long as a map layer's coordinate system is defined properly. For example, for an Arc shapefile there is an associated projection (*.prj*) file. The projection file contains the mathematical formulas that relate to the spherical coordinates on the globe to flat, planar coordinates.

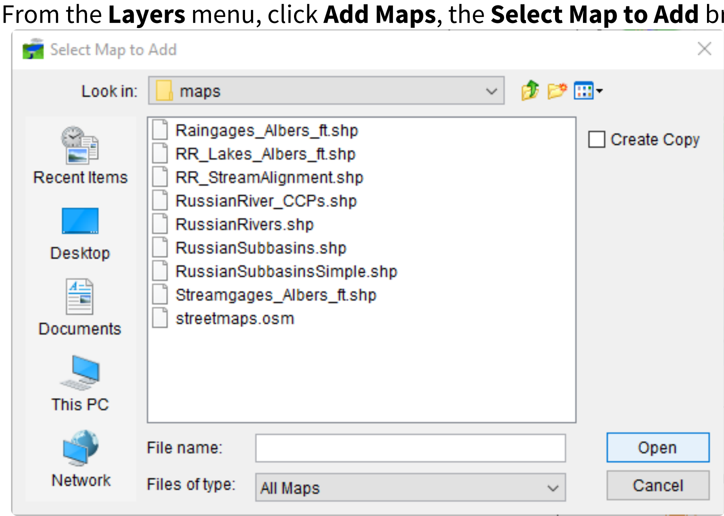
Map layers are added from the **Maps** menu in any of the HEC-RTS modules. When a map layer is added to the watershed, HEC-RTS reads the coordinates of points, lines, and polygons on the map layer and resets the geographic extents of the watershed equal to the smallest rectangle that will contain all objects in the map layer. When adding more than one map layer, you must ensure that each map layer is projected in the same coordinate system (Setting Up the Coordinate System). For example, if an Arc shapefile (.*shp*) is added that displays the major streams in a watershed area, the geographic extents will change according to the projection of that shapefile. If the shapefile is projected in *State Plane Coordinates* in feet, HEC-RTS will set the geographic extents accordingly, and the units of the stream alignment will be in feet.

To add map layers:

1. From the HEC-RTS main window, from the **Maps** menu, click **Map Layers,** the **Map Layers** dialog will open (Figure 1).



2. From the **Layers** menu, click **Add Maps**, the **Select Map to Add** browser (Figure 2) will open.



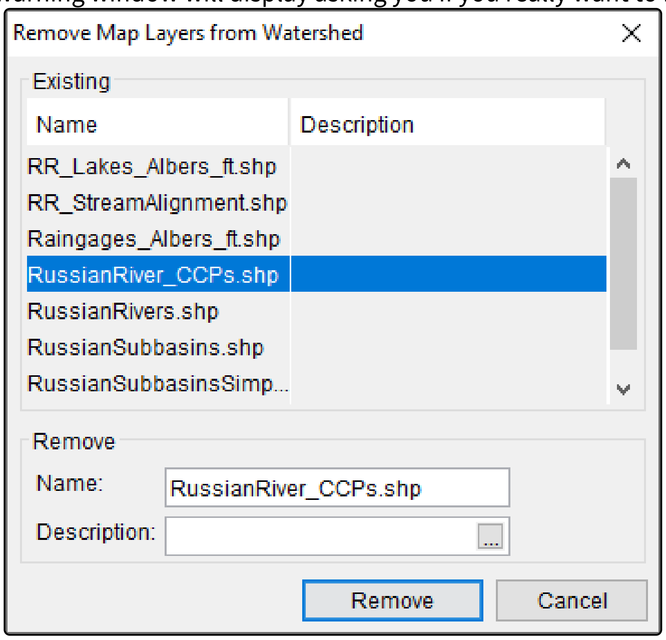
3. Select the desired map layer format from the **Files of type** list. Select the filename, and click **Open**. The **Select Map to Add** browser will close (Figure 2). The selected map layer will display in the map window, and in the **Map Layers** dialog (Figure 1), the map layer will display under the **Map Layers** tree.

When enabled, the **Create Copy** checkbox (default) on the **Select Map to Add** browser (Figure 2) will copy the selected map layer to the active watershed's directory structure under the *maps* folder.

6.3 Removing Map Layers

To remove map layers:

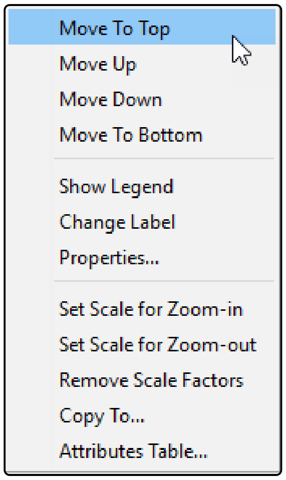
1. From the **Map Layers** dialog, from the **Layers** menu, click **Remove Map Layer**. The **Remove Map Layers from Watershed** dialog will open (Figure 1). Select the map layer to remove from the watershed, click **Remove**. A warning window will display asking you if you really want to remove the selected map layer, click **Yes**.



2. The **Remove Map Layers from Watershed** dialog will close (Figure 1), and the selected map layer will no longer display in the **Map Layers** dialog.

6.4 Shortcut Menu for Map Layers

Right-click on a map layer (non-primary) in the **Map Layers** dialog, a shortcut menu displays (Figure 1). The shortcut menu provides several ways to manipulate a map layer. This ability to manipulate depends on the type of map layer (Figure 1), since HEC-RTS does not provide the same capabilities to all recognized map layer formats. The available commands are:



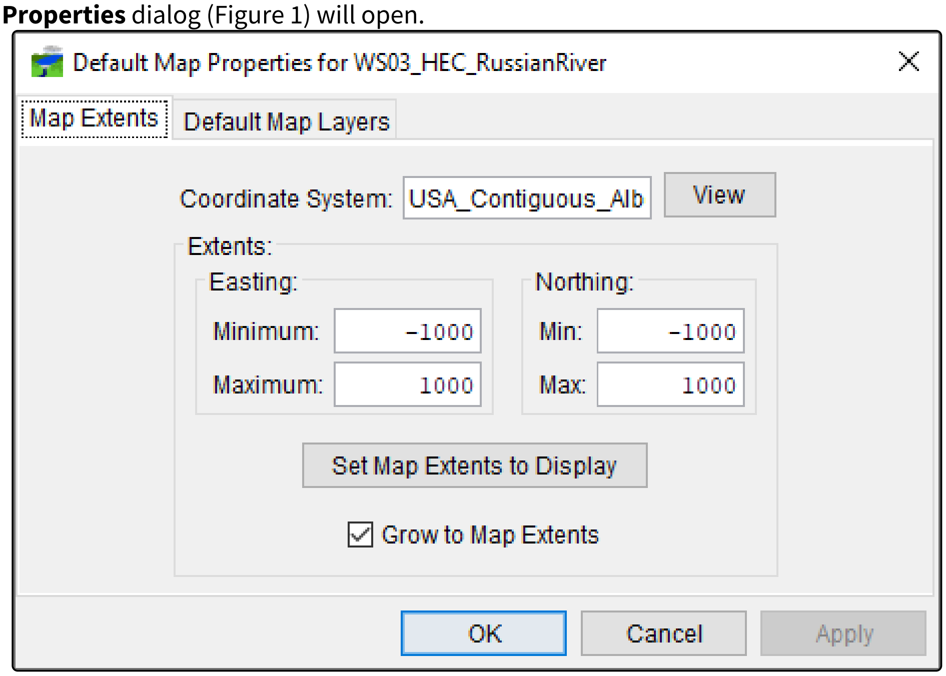
Move to Top	Move the layer to the top of the tree.
Move Up	Move the layer up the tree one position.
Move Down	Move the layer down the tree one position.
Move to Bottom	Move the layer to the bottom of the tree.
Show Legend	Show the legend for the layer.
Hide Legend	Hide the legend for the layer.
Change Label	Change the label of the layer in the **Map Layers** dialog. This does not change the filename of the layer.
Properties	Displays the properties editor specific to the selected layer. Depending on the format of the layer, this dialog may be editable or not. See Using Map Editors for further information on the available map editors.
Set Scale for Zoom-in	Set the zoom-in visualization scale for the layer as detailed in Setting the Visualization Scale.

Set Scale for Zoom-out	Set the zoom-out visualization scale for the layer as detailed in Setting the Visualization Scale.
Remove Scale Factors	Disable or clear all scale settings for the layer as detailed in Setting the Visualization Scale.
Copy To	Copy the layer to another folder.
Attributes Table	Opens the **Attributes Table** window, which displays the attributes of the selected layer.

6.5 Geographic Reference for Map Layers

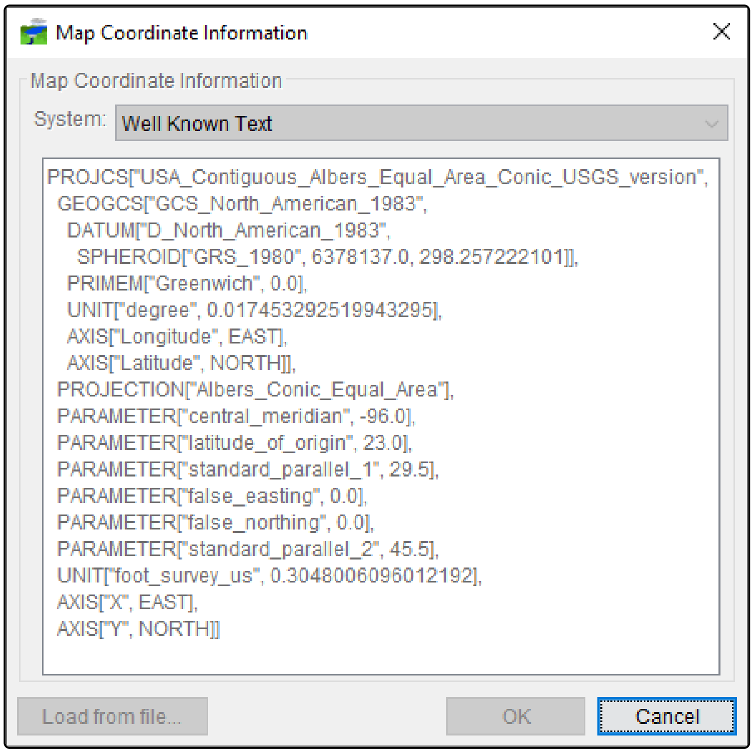
To maintain a geographic reference (also called a geo-reference), HEC-RTS uses a user-selected and customizable coordinate system, called the World Coordinate System (WCS). This superimposes a grid on layer features to establish x and y coordinates in WCS for each point on the layer. The x-coordinate is referred to as "easting", and the y-coordinate is referred to as "northing". You can select the extent of this grid, the dimensions of the cells of the grid, the units of measurement of the grid, and the location of the origin of the grid. Because of this flexibility, the user can specify and use virtually any grid coordinates convenient for HEC-RTS model integration. To establish geographic reference:

1. From the HEC-RTS main window, from the **Maps** menu, click **Default Map Properties**. The **Default Map Properties** dialog (Figure 1) will open.



2. The **Default Map Properties** dialog contains two tabs: **Map Extents** and **Default Map Layers**. The **Map Extents** tab includes:

<table>
<tr><td>Coordinate System</td><td>This field identifies the established coordinate system for the watershed. To view the coordinate system, click View. The Map Coordinate Information dialog (Figure 2) will open, providing you with information about the coordinate system that was established when the watershed was created. You are not able to edit this information.</td></tr>
<tr><td>Extents</td><td>Easting Minimum and Maximum; Northing Minimum and Maximum. These values (Figure 1) indicate the location of the left, right, bottom, and top borders (respectively) of the grid in the map window.



18 Figure 2 Map Coordinate Information Dialog</td></tr>
<tr><td>Set Map Extents to Display</td><td>This will set the limits of the map window. If you zoom in on an area and click Set Map Extents to Display, the extents on the Default Map Properties dialog (Figure 1) will change to the zoomed area.</td></tr>
<tr><td>Grow to Map Extents</td><td>When selected, HEC-RTS automatically sets the geographic extents to define the smallest rectangle that encompasses all the objects in the watershed.</td></tr>
</table>

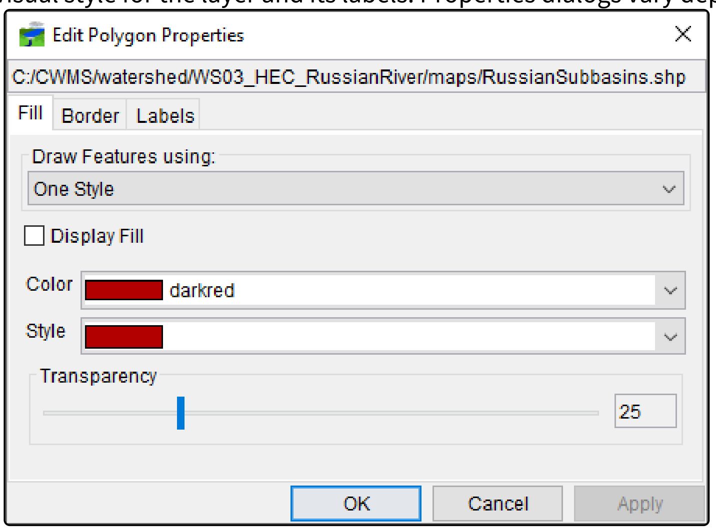
The geographic extents of map layers must be selected carefully to ensure that the entire watershed is included. HEC-RTS can transform map layer coordinates "on-the-fly" if coordinate systems are defined properly in ways that HEC-RTS can determine the coordinate system. For example, for an Arc shapefile there is an associated projection (*.prj*) file. The projection file contains the mathematical formulas to relate spherical coordinates on the globe to flat, planar coordinates. Therefore, you may need to use GIS tools to identify or transform the layers from one coordinate system to another before using them with HEC-RTS.

3. The **Default Map Layers** tab (Figure 1) includes a list of map layers that are being displayed in a map window. You use check boxes to display or hide map layers.

6.6 Map Layer Properties

As described in Map Layer Formats, HEC-RTS supports several map layer formats, each with its own properties. To edit those properties:

1. From the **Map Layers** dialog, right-click on a layer to edit, from the shortcut menu, click **Properties**.
2. The appropriate properties editor will display (Figure 1). For example, in Figure 1, you have selected a map layer that is a polygon shapefile. The **Edit Polygon Properties** editor will open (Figure 1). Here, you can specify the visual style for the layer and its labels. Properties dialogs vary depending on the type of layer you are editing.



6.7 Setting the Visualization Scale

The easy way to set the visualization scale for map layers is outlined below:

1. From a map window, zoom in to a point at which you want a map layer to become visible.
2. From the **Map Layers** dialog, right-click on that map layer to set its visualization scales. From the shortcut menu, click **Set Scale for Zoom-in**. This sets the visualization scale for the map layer. If you go to the map window and zoom out, the layer will no longer appear.
3. Inversely, if you want the layer to become visible as you zoom out, from the shortcut menu, click **Set Scale for Zoom-out**.
4. Visualization scale settings can be cleared or disabled. From the shortcut menu, click **Remove Scale Factors**. The visualization scale for the layer is cleared.

7 Using HEC-RTS - Overview

This chapter describes the typical workflow involved in the routine use of HEC-RTS to support the water management process. This chapter will provide information on running models, analyzing results, and, providing information to the user on how to run models when an extreme event occurs.

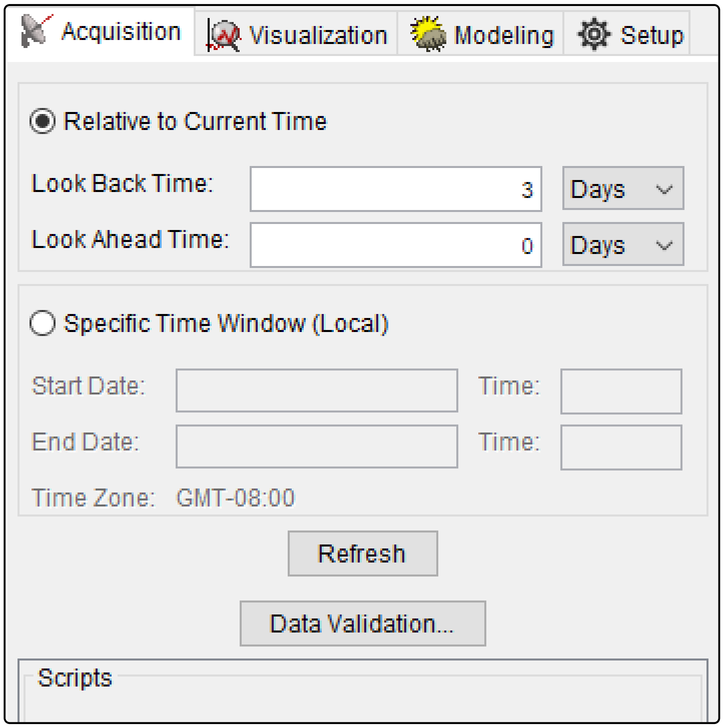
At this point, you have the ability to configure a watershed, now you are ready for the routine tasks of running forecast models; viewing and analyzing results; revising model data, as needed; and posting results to HEC-RTS DSS files to prepare for the next forecast.

7.1 Managing Data for HEC-RTS

Data in the watershed that needs to be monitored will be selected from the time series icon layers, which have already been configured (Time Series Icons and Time Series Icons Layers). The time series icons will be placed at locations in the watershed that correspond with data that needs to be monitored, and will be organized into layers (groups), which will make the data readily visible. Time series icons are available in the **Acquisition, Visualization**, and **Setup** modules in HEC-RTS and provide access to functions related to data. This includes acquiring, monitoring, validating, and editing data by using the **Acquisition** module. The **Visualization** module provides a flexible set of tools that allows visualization of data (i.e., gridded precipitation data). These visualization tools help you evaluate the current state of the watershed by comparing the data to threshold values that define normal conditions. For example, let's say an upper threshold value for precipitation data of three inches has been defined. If a precipitation value is four inches, then the time series icon will alert you using a visual indicator.

7.1.1 Acquiring Data

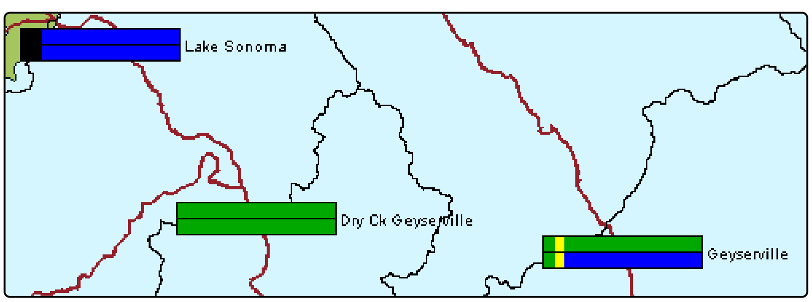
In the **Acquisition** module, you will retrieve time-series data from gages. This data will typically include precipitation; river flows and stages; temperature; and, reservoir releases and elevations. One way you can view this data, is by defining a time window that is relative to the current time by setting look back and look ahead times (in days or hours). Another way to view data is historically, you specify a starting and ending date and time. An example of defining time window relative to current time is displayed in Figure 1.

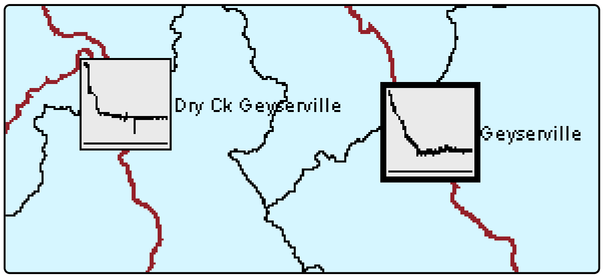


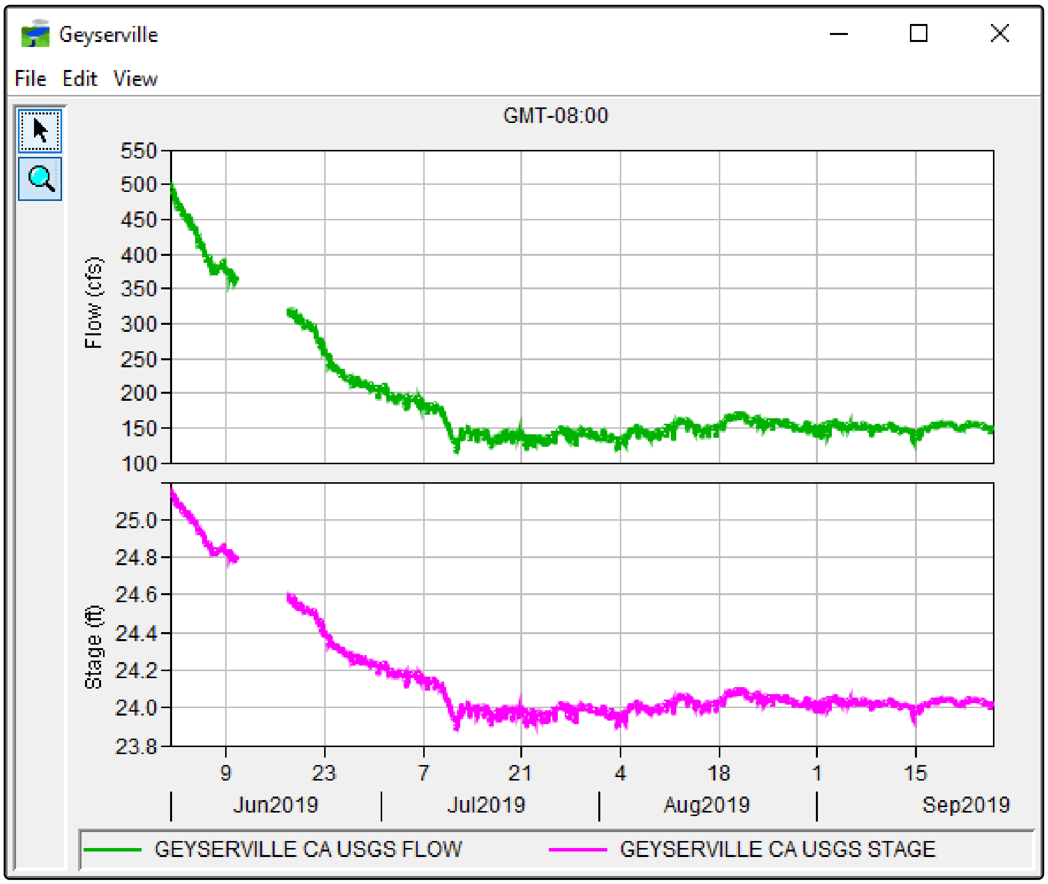
19 Figure 1 Acquisition Tab - Time Window

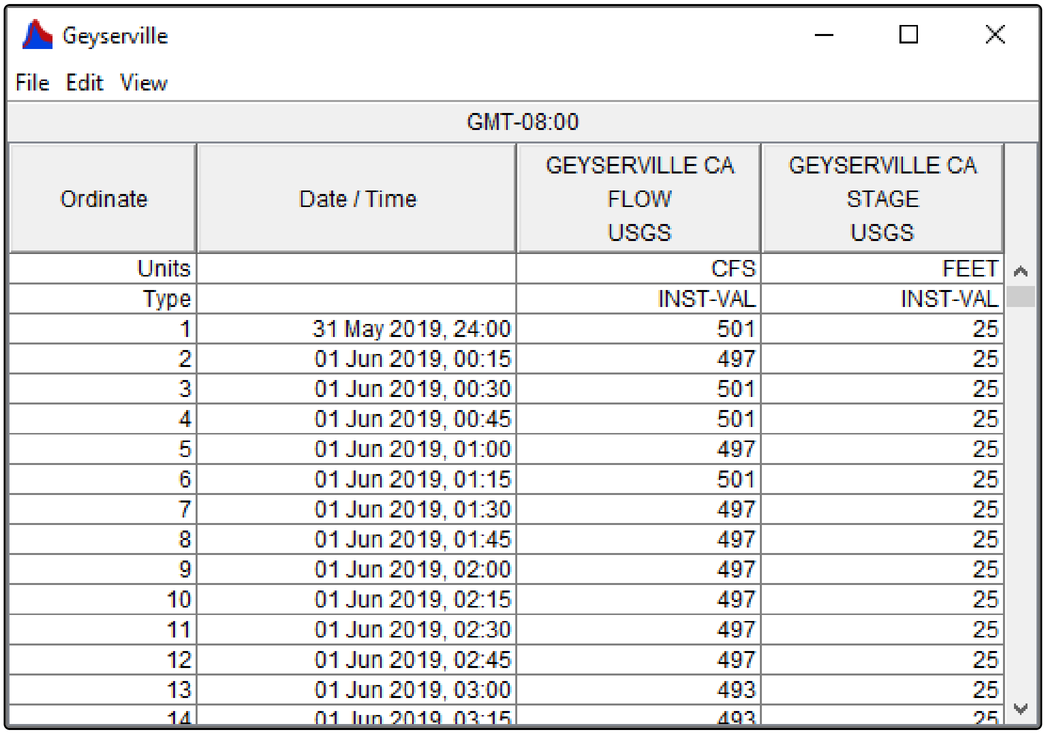
7.1.2 Monitoring and Validating Data

In the **Acquisition** module, you will monitor and validate your time-series data. Several tools are available to show the quality and quantity of the data. Quality color bars (Figure 2) and thumbnail plots (Figure 3) provide you with a quick overview of the acquisition processes by showing you the quality (missing values needing to be filled-in) and quantity (magnitude of data and possible data outliers) of your data. If the quality color bar or thumbnail plot for a location indicates that the data are questionable, then you can check the data using plots (Figure 4) and tables (Figure 5).

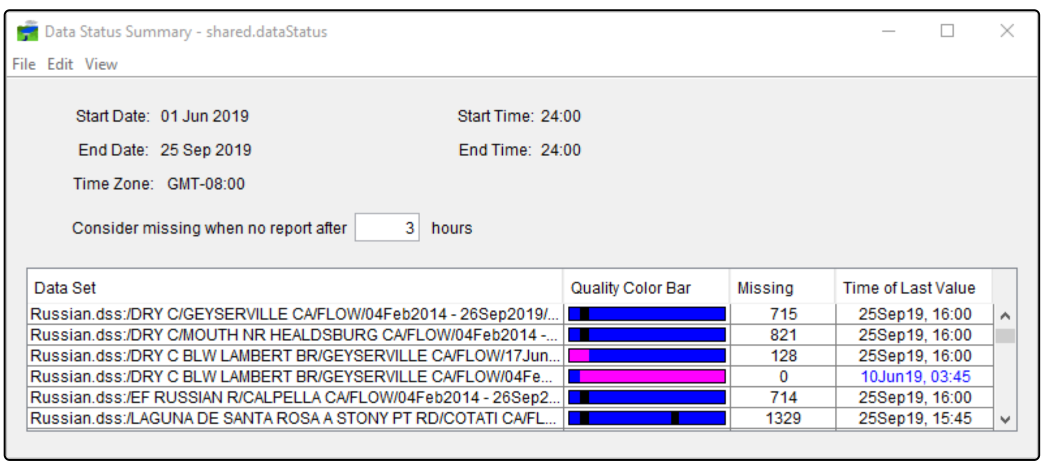



Dry Ck Geyserville
Geyserville


Geyserville
File Edit View
GMT-08:00
Flow (cfs)
550
500
450
400
350
300
250
200
150
100
Stage (ft)
25.0
24.8
24.6
24.4
24.2
24.0
23.8
9
23
7
21
4
18
1
15
Jun2019
Jul2019
Aug2019
Sep2019
GEYSERVILLE CA USGS FLOW
GEYSERVILLE CA USGS STAGE



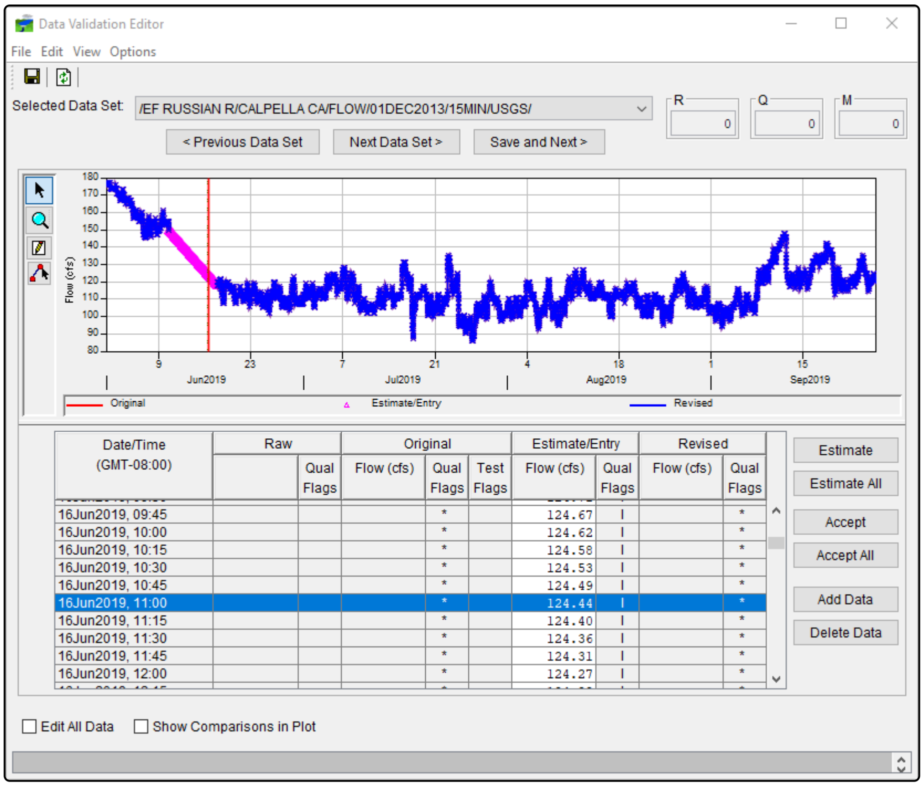
You can use the **Data Status Summary** (Figure 6) to view groups of data (e.g., all precipitation data, all flow data). Refer to Acquisition Module and Data Status Lists for additional details for monitoring and validating data in HEC-RTS.



20 Figure 6 Data Status Summary

7.1.3 Editing Data

When you are checking the validity of data to verify its accuracy, you might need to edit questionable data and fill-in missing data. You use the **Data Validation Editor** (Figure 7) in the **Acquisition** module to edit time series data. Refer to Data Validation Editor for further details.

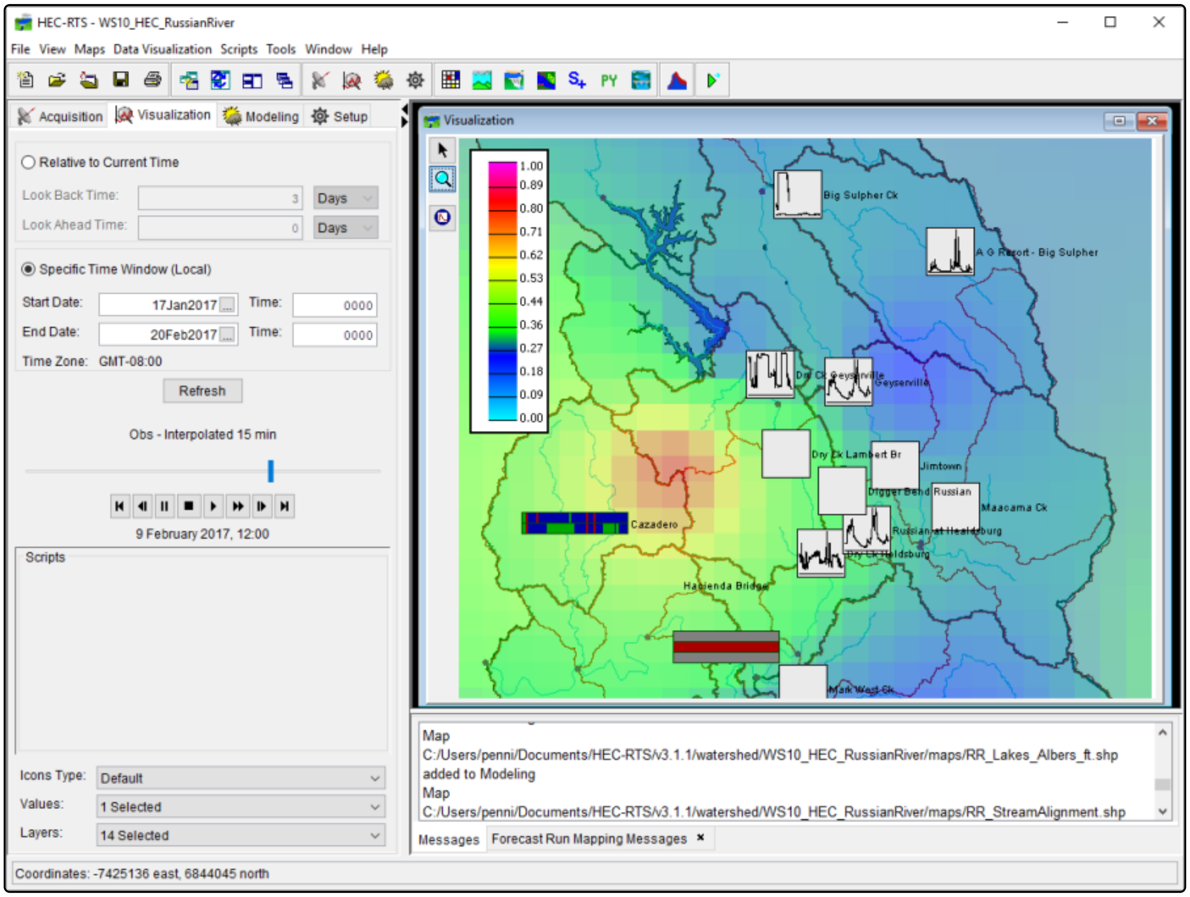


21 Figure 7 Data Validation Editor

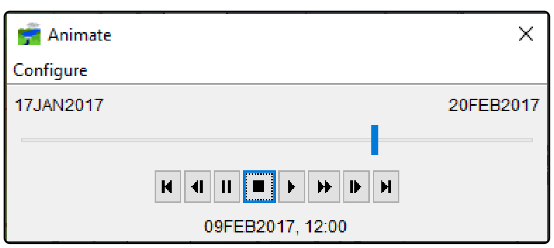
You can use text-entry editing or graphical editing to make data revisions. When there are data gaps (i.e., missing data), this tool is also convenient for estimating values using the graphical editor or the table of data values. After editing the data, you can choose to accept the changes and store the revised data values back to a DSS file. Refer to Data Validation Editor for additional details for editing data.

7.1.4 Visualizing Data

The **Visualization** module allows you to view current conditions in your watershed and the associated hydrometeorological data in a georeferenced context (e.g., gridded precipitation), as shown in Figure 8. Grid animation controls allow you to view precipitation grids for the time window you have chosen in the **Visualization** module. These controls (Figure 9) function as you would expect the controls to work for a video player.

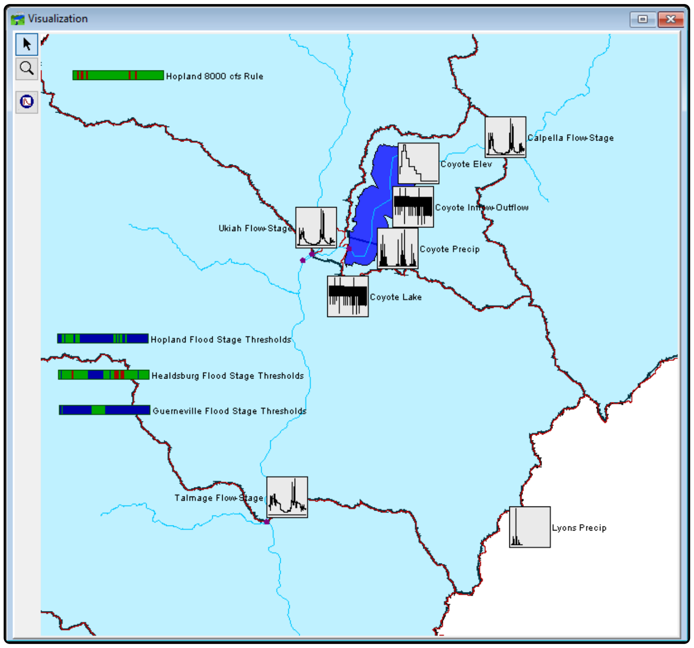


22 Figure 8 Visualization Module - Gridded Precipitation Displayed



23 Figure 9 Visualization Module - Grid Animation Controls

Like the quality color bars used in the **Acquisition** module, you can use threshold color bars (Figure 10) in the **Visualization** module to compare current data against threshold values that you set for a location. For example, in the **Acquisition** module you may wish to display quality color bars to identify missing data, while in the **Visualization** module, you may wish to use threshold color bars or thumbnail plots to understand your watershed's hydrometeorological or operational conditions.



24 Figure 10 Visualization Module -Threshold Color Bars

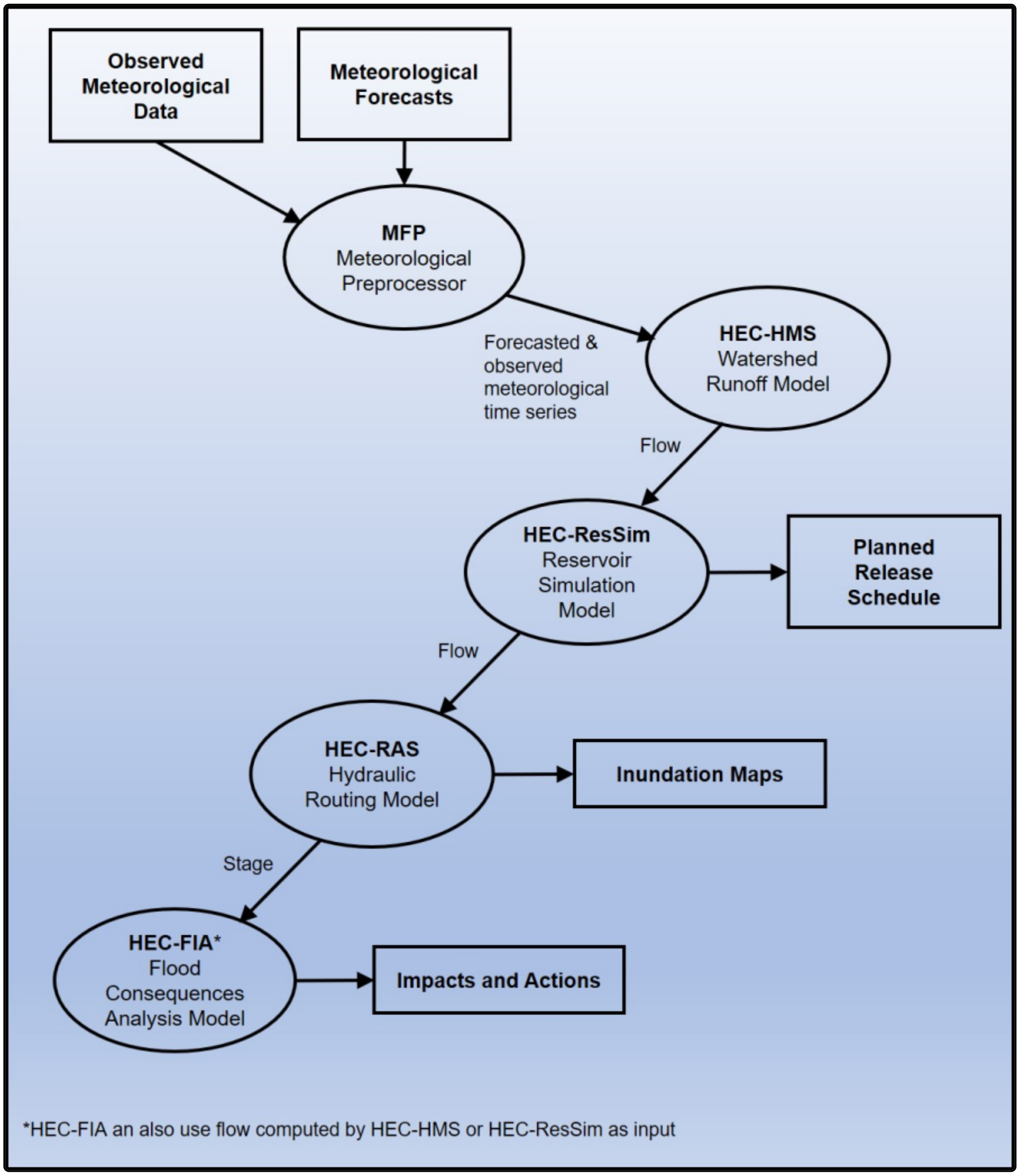
For example, in Figure 10, the upper left threshold color bar displays red for the parts of the time window when a gage downstream of the reservoir exceeds an operational flow threshold. The other three threshold color bars (Figure 10) display the times when key downstream gages exceed the NWS (National Weather Service) "monitor" stage (blue) or flood stage (red). You can also use thumbnail plots and tables to visualize your time-series data. For example, in Figure 10, thumbnail plots display hydrographs at gages upstream and downstream of the reservoir, and on a lateral branch of the river. Refer to Visualization Module for additional details for visualizing data.

7.2 Running Forecast Models

Now that the quality of the time-series data has been validated, you are ready to create an HEC-RTS forecast. An HEC-RTS **forecast** is a simulation of watershed processes and consequences of potential flooding from various software applications configured in HEC-RTS. The meteorological data processor, MFP, combines observed and forecasted meteorological data into gridded datasets for use in HEC-HMS. The hydrologic software application, HEC-HMS, computes flow from forecasted and observed meteorological time-series. The reservoir operation software application, HEC-ResSim, computes flow and release schedules. The hydraulic software application, HEC-RAS, computes stage and inundation maps. The impact analysis software application, HEC-FIA, computes flood consequences and action reports. Prior to creating an HEC-RTS forecast, you will have already configured an HEC-RTS watershed. Specifically, **Model Alternatives** will have been configured, **Model Alternative Keys** (Model Alternatives and Forecast Runs) assigned, Program Order and **Forecast Runs** defined, and **Model Linking** (Model Linking Editor) configured.

An HEC-RTS forecast might be configured using a typical forecast run that includes something similar to the following model alternatives: Winter Nextrad QPF (for MFP), normal conditions (for HEC-HMS), normal reservoir operations (for HEC-ResSim), normal river conditions (for HEC-RAS), and no flooding consequences (for HEC-FIA). If a high rainfall event is forecasted, an HEC-RTS forecast might be created using a forecast run with model alternatives similar to the following: HRRR QPF (for MFP), wet basin conditions (for HEC-HMS), restricted downstream channel capacity (for HEC-ResSim), high channel flow (for HEC-RAS), and evacuate with warning (for HEC-FIA).

In the **Modeling** module, the user will specify a forecast time window, create and manage forecast extract and post lists, extract time-series data for the forecast, create and manage forecast runs, edit model parameters, and run the models. Figure 1 displays the data flow concepts through the five standard HEC-RTS analysis applications (MFP, HEC-HMS, HEC-ResSim, HEC-RAS, and HEC-FIA).



25 Figure 1 Data Flow Between Models

7.2.1 Hydrologic Modeling (MFP, HEC-HMS)

The meteorological preprocessor **MFP** will be used to combine observed precipitation data and future precipitation information into a single precipitation dataset that can be used by the watershed runoff model, **HEC-HMS**. Using the observed and future precipitation data, HEC-HMS computes runoff, including uncontrolled local flows that are typically used by HEC-ResSim. The types of data typical that need to be changed for a forecast are loss rates and baseflow.

7.2.2 Reservoir Simulation (HEC-ResSim)

After the upstream boundary flows and uncontrolled local flows have been computed by HEC-HMS, you will run the reservoir simulation application **HEC-ResSim** to use those flows during the operation of the reservoirs to compute regulated flows. Reservoir operations might include zones and rules for day-to-day operations, along with zones and rules for extreme events.
The types of data that might typically be changed for a forecast are the starting conditions for reservoirs. As long as the reservoir starting conditions are linked to real-time data (starting elevations or storages, reservoir or outlet releases, and any other boundary conditions needed by the model), then the user shouldn't have to make any revisions to the HEC-ResSim model in order to run routine HEC-RTS forecasts.

7.2.3 River Analysis (HEC-RAS)

Regulated flows computed by HEC-ResSim are typically used as boundary conditions to the hydraulic routing model, **HEC-RAS**. Computed flows from HEC-HMS or supplemental models can also be used as input to HEC-RAS. In turn, HEC-RAS computes stage, which is input to HEC-FIA. The types of data that might typically be changed for a forecast are *n* values at selected cross sections to match profile information better throughout the lookback period.

7.2.4 Impact Analysis (HEC-FIA)

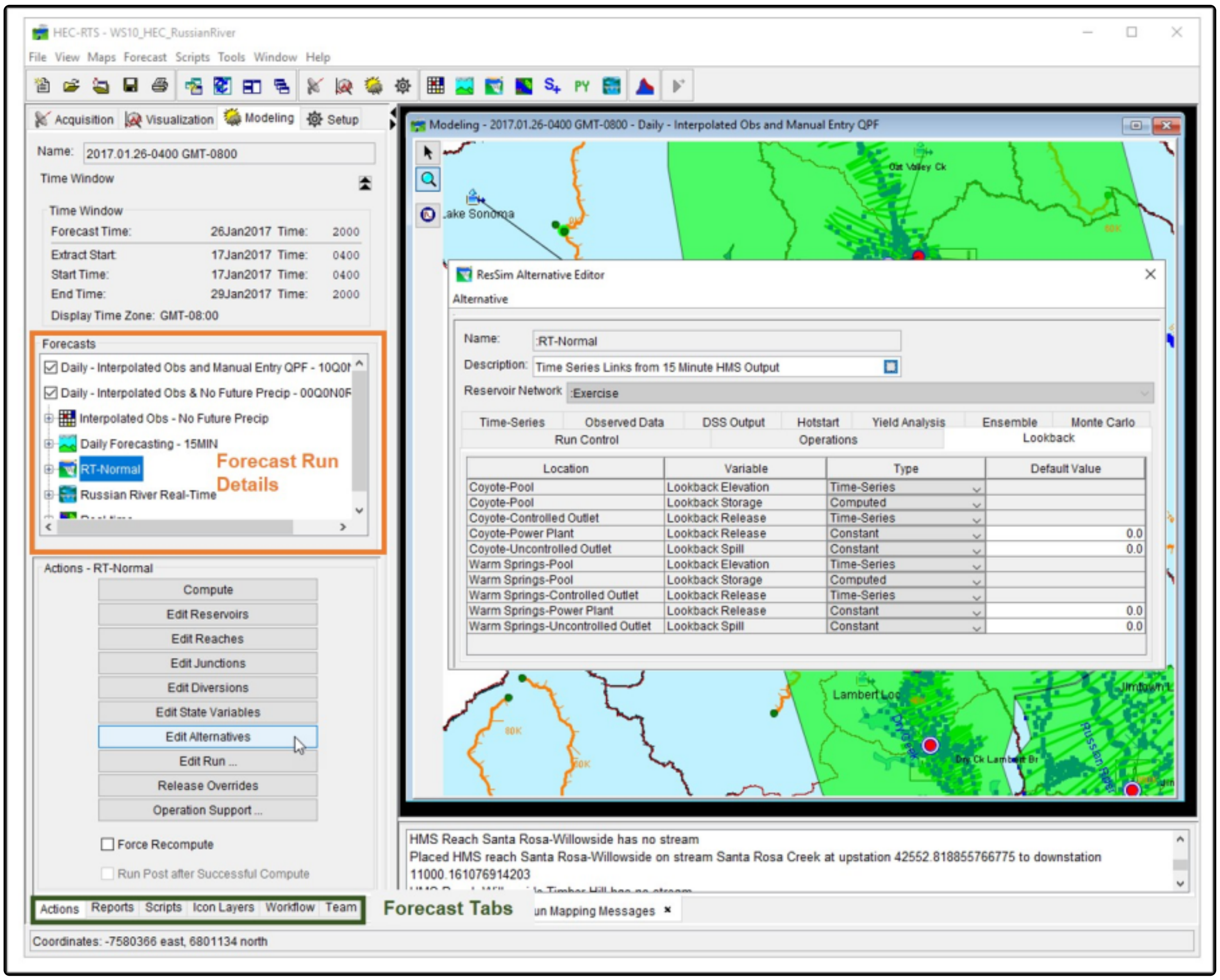
HEC-FIA analyzes impacts within inundated areas based on computed hydrographs from HEC-ResSim and stages from HEC-RAS. HEC-FIA calculates flood damage determined by rising stage in a stream, river, lake, or reservoir. HEC-FIA also computes action reports that tell responders what actions need to done based on the computed stage values. The types of data that might typically be changed for a forecast are adjustments to the levee failure stages to see the associated damage and impacts during an event.

7.3 Viewing and Analyzing Results

Viewing of modeling results occurs from the **Modeling** module by using plots and reports. You can view results at individual modeling elements (e.g., computation points, reservoirs, impact areas, etc.). Typical results include tabulated subbasin precipitation depths, subbasin hyetographs, stage and flow hydrographs, reservoir release decisions, hydraulic plots and tables, floodplain inundation maps, and impact reports. Refer to HEC-RTS Results for more details on results.

7.4 Editing Model Data

After application results have been reviewed, revisions might be needed to model data. The active forecast run in the **Modeling** module, indicates which software applications are being used in the forecast. For each software application, the user can edit some of the data by clicking on the appropriate software application layer in the forecast tree and using the buttons located in the **Actions** tab (Figure 1). Alternately, the user can click on the appropriate application icon in the **Modeling** module's toolbar to open that application's native interface.



26 Figure 1 Example of Editing Model Data (HEC-ResSim Alternative Editor)

For example, if the user wants to change the lookback storage (i.e., starting condition) for a reservoir, the user needs to edit the HEC-ResSim model alternative being used in the forecast run. From the **Modeling** module (Figure 1), from the **Forecast Run Details** section (Figure 1), click on the HEC-ResSim model alternative (*RT-Normal*). From the **Forecast** tabs (Figure 1), click on the **Actions** tab. A list of HEC-ResSim commands will display, you want to edit the HEC-ResSim model alternative, so click **Edit Alternatives** (Figure 1). At this point, you now have access to HEC-ResSim's **Alternative**

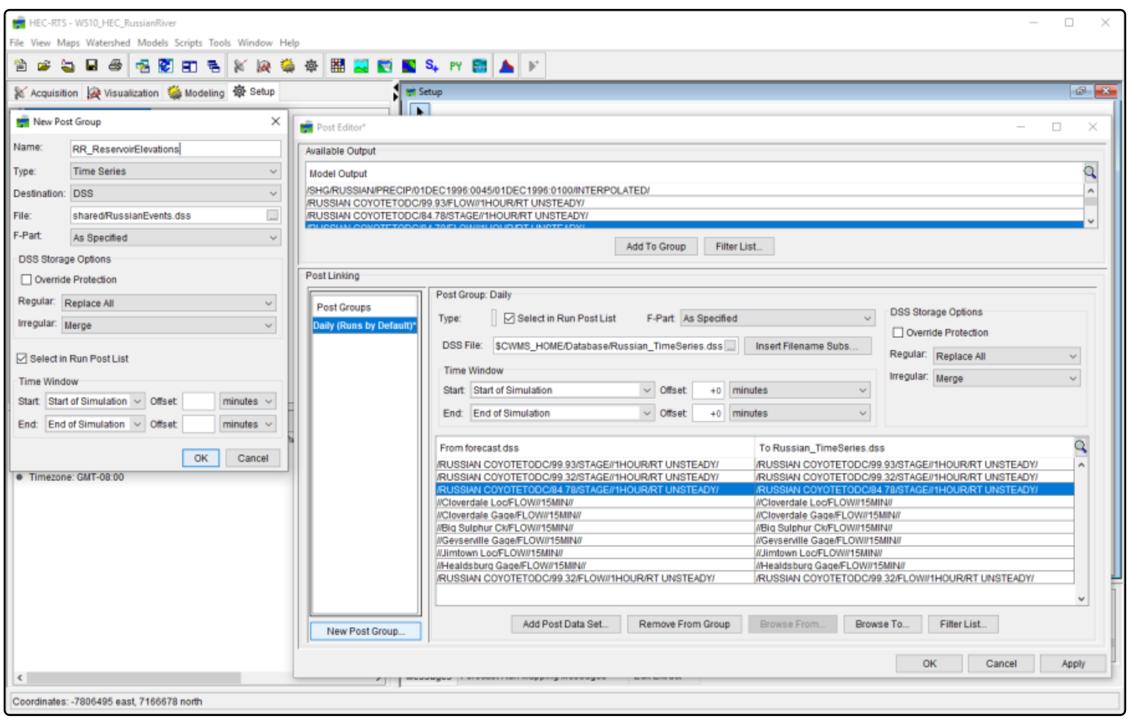
Editor, the **Lookback** tab (Figure 1) contains the reservoir starting storage (and elevation) values (among other initial conditions).

Once you have made your model alternative revisions, be sure to save the watershed. The example, provided you with one way to edit model alternatives, there are several other ways in HEC-RTS to edit model alternatives. If revisions are made to a model alterantive from the **Forecast Run Details** area (Figure 1) of the **Modeling** module, the revisions are only applicable for the forecast run that is part of the selected forecast. Other forecast runs (as well as other forecasts that have been run with the selected forecast run) will not contain the revisions. If the user wants the forcast run revisions to be available for subsequent forecasts, then you will need to save the revisions to the "base" data. To save "base data", select the forecast run (*Daily – Interpolated Obs and Manual Entry QPF*) that contains the revisions, click the **Actions** tab (Figure 1), click **Save To Base** (located on the **Actions** tab).

7.5 Posting or Saving Forecast Results

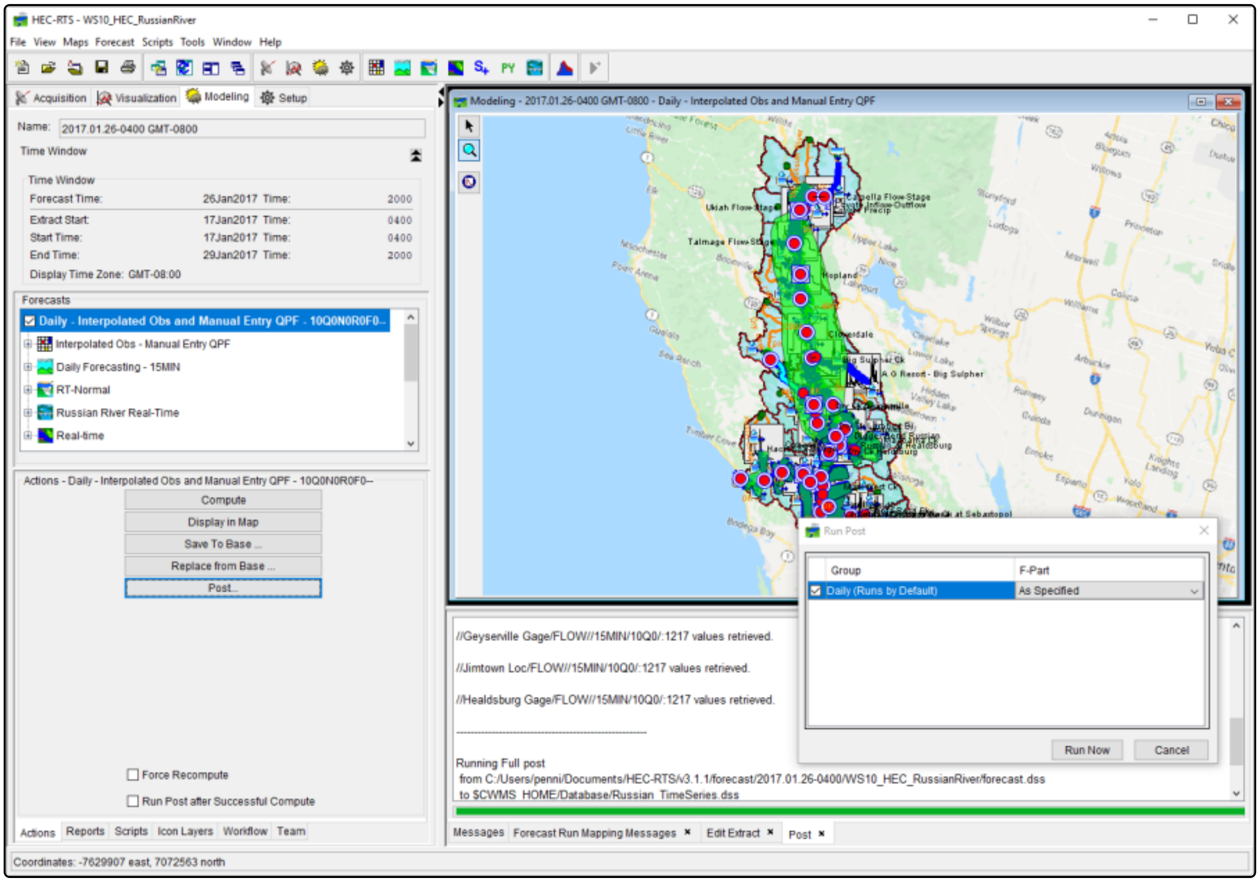
Now that you have run a forecast and are satisfied with the results, you will want to post (save/store) forecast results to a DSS file. HEC-RTS provides many options for posting results - by forecast, post only certain results, post by model alternative, and many other ways. Refer to HEC-RTS Forecasts for additional details for posting forecast results.

As an example, you might want to only store the reservoir elevation and release time-series values (remember, this is just a simple example). From the HEC-RTS main window, click on the **Setup** tab, from **Models** menu, click **Edit Post**. The **Post Editor** (Figure 1) will display, from the **Available Output** box, select the appropriate pathnames for reservoir elevation. Click **Add To Group** (Figure 1), if no other post groups have been defined the **New Post Group** dialog will display (Figure 1). In the **Name** box enter a name, and from the **File** list select a DSS file, click **OK**. The **New Post Group** dialog box will close, and the **Post Editor** will display information about the created post group (Figure 1).



27 Figure 1 Edit Post

After the post group is created, from the HEC-RTS main window, click on the **Modeling** tab (Figure 2), from the **Forecast** tabs, click **Actions**. Click **Post**, the **Run Post** dialog will open (Figure 2). Select a post group, click **Run Now**, a **Post** tab is added to the **Message Pane**, and the results of the post are displayed.



28 Figure 2 Running a Post

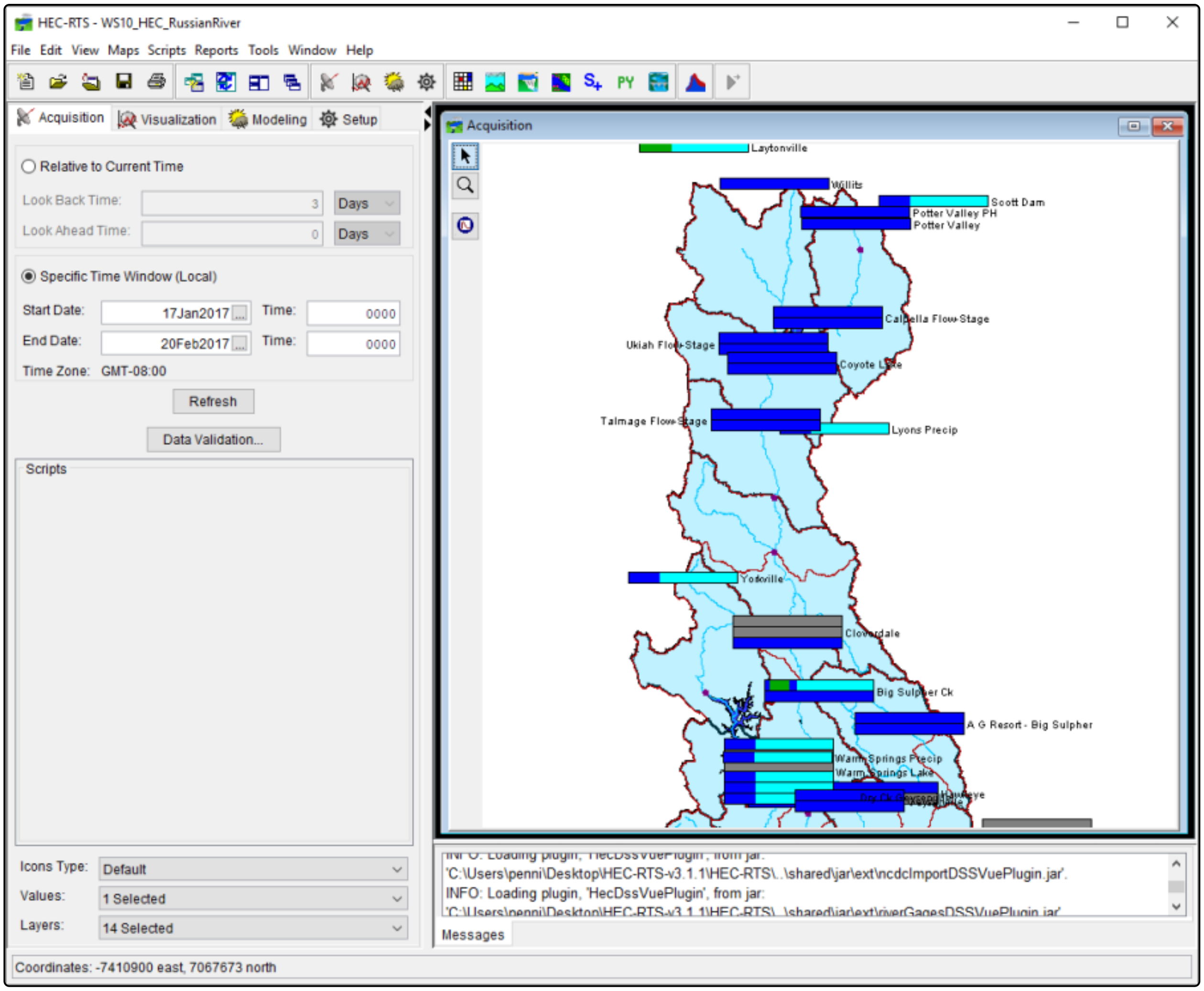
8 Acquisition Module

The **Acquisition** module allows you to examine data, evaluate the quality of data, and quickly identify problem locations for a watershed. HEC-RTS keeps you apprised of the quality of data in DSS files by displaying color bars and thumbnail plots that dynamically convey the quality of data at individual locations; uses plots and tables to provide detailed views of data; and, provides a mechanism to group gages for viewing (such as precipitation or flow), which allows you to view details for more or fewer gages than appear in the geo-referenced map display.

Once issues with data have been identified, the Data Validation Editor can be used to validate the data. The **Data Validation Editor** allows the user to select specific locations for validation (ad hoc mode) or to use pre-defined validation lists (list mode). The **Data Validation Editor** can filter data to identify and display only those data time series with questionable, missing, and/or rejected data. Additionally, the **Data Validation Editor** offers tools for both text-entry editing and graphical editing. With the former, the user can enter corrected values in a table. With the latter, the user can use the mouse to create line segments in a plot to replace questionable or missing data. Also, automated estimates can be used to edit data.

8.1 Acquisition Module Overview

The **Acquisition** module (Figure 1) contains buttons that execute commands specific to data acquisition. These commands include setting up a time window, updating time series icons in the map window, and access to the Data Validation Editor. A detailed discussion of common screen components is presented in HEC-RTS Interface.



29 Figure 1 Acquisition Module

8.1.1 Menu Bar

The following is an overview of the menu bar (Figure 1) for the Acquisition module. The **File**, **View**, **Maps**, **Scripts**, **Tools**, **Windows**, and **Help** menus are discussed in HEC-RTS Interface. The commands available from the menus will facilitate the acquisition and management of time series data for a watershed.

Edit	From this menu the user can edit color bar settings; access the **Data Validation Editor**; create a validation list from selected icon time series; and, delete validation lists. Available commands are: **Icon Quality Colors**, **Data Validation**, **Create Validation List**, and **Delete Validation List**. **Note:** For HEC-RTS, validation lists are not used, and the subject is not discussed in this User's Manual

Reports	From this menu the user can view the quality of data in the form of data status lists. The user can select from existing files of data status lists, or create new data status lists, and save the data status for future use. The only available command is **Data Status Summary**.

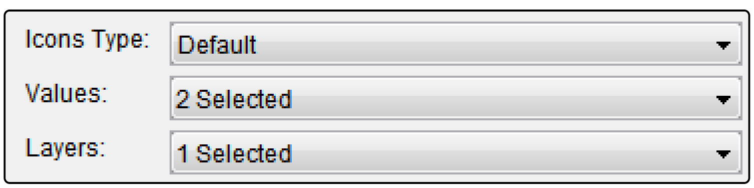
8.1.2 Map Window

The **Map Window** (Figure 1) of the Acquisition module contains map layers and layers of time series icons that allow viewing of data acquisition processes in a geo-referenced context. The user can customize how time series icons appear according to the needs and objectives of the watershed being studied. For example, time series locations that need to be evaluated daily for an individual watershed may appear as quality color bars to facilitate quick review of data. On the other hand, the use of time series icons to monitor data provides space-conserving graphical images on the schematic. Customizing the time series icon layers is done in the **Setup** module (Time Series Icons Layers).

Quality color bars and/or thumbnail plots are normally used in the Acquisition module because these icons allow you to quickly scan the displayed gages and determine where problems may exist. Both icons can be set to update dynamically to reflect the relative quality of the data.

8.1.3 Time Series Icon Controls

The **Time Series Icon Controls** (Figure 2) allow you to select attributes for the time series icon layers that are displayed. The time series icon controls are located on the **Acquisition** tab near the bottom of the pane (Figure 1). There are three attributes available: **Icons Type**, **Values**, and **Layers** (Figure 2).



30 Figure 2 Time Series Icon Controls

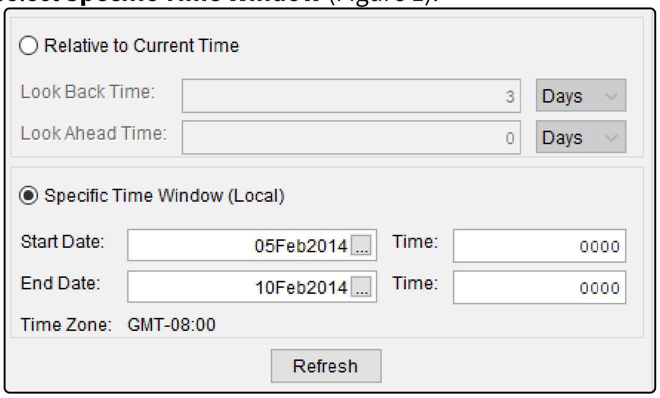
Icons Type	forces all icons within a map window to display as the selected **Icon Type** in the list. Selecting the **Default** option in the list will display the icons as they were set originally in the Setup module (Time Series Icons).
Values	allows you to display the time series icon's data as labels, which display next to or on the location of their associated time series icons. For example, you can choose to display the data's total, minimum, and maximums values as labels. Multiple values can be selected at once.
Layers	allows you to select the sub-layers that are visible in the map window. Multiple layers can be selected at once.

By selecting the attributes, you can change how time series icons are displayed within the Acquisition module. For more detailed information on how to configure the properties that control the threshold color bar icons displayed (Time Series Icons Layers).

8.2 Set Time Window

You can either set a specific (i.e., fixed) time window (for reviewing a historical event) or you can set a time window relative to the current time (the default setting, used for reviewing data). To set or change the time window:

1. From the **Acquisition** module (Figure 1), for the default time window, HEC-RTS assumes that you want to view current data, so **Relative to Current Time** is selected (Figure 1). If you want to view historical data, you must select **Specific Time Window** (Figure 1).

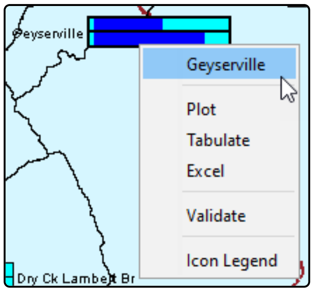


31 Figure 1 Acquisition Module - Time Window Setup

2. If you would like to set the **Time Window** (Figure 1) relative to the current date and time, select the **Relative to Current Time** and enter the **Look Back Time** and select either **Days** or **Hours** as the time unit. Normally you will set the time window to extend backward one to thirty days from the current time. Optionally, you can enter a **Look Ahead Time** and select either days or hours; this allows you to look ahead at data.
3. If you would like to set a historical **Time Window** (Figure 1), select **Specific Time Window** and enter the **Start Date** and **Time** of the time window and the **End Date** and **Time** of the time window for the historic data.
4. Click **Refresh** (Figure 1) to complete the time window change.

8.3 Time Series Icon Shortcut Menu

From the **Map Window**, the time series icons display, by right-clicking on a time series icon, a shortcut menu (Figure 1) will display that allows you to view the data associated with a selected time series icon either in a plot, table, or using Microsoft Excel®. Also, from the shortcut menu you can access the Data Validation Editor to edit the data associated with the selected time series icon. Another option is that you can view the legend of the selected time series icon. The shortcut menu is the same for all available time series icon types (i.e., for the color bars, thumbnail plots, graphics, or dots).



32 Figure 1 Time Series Icon Shortcut Menu

8.4 Reviewing Data Quality

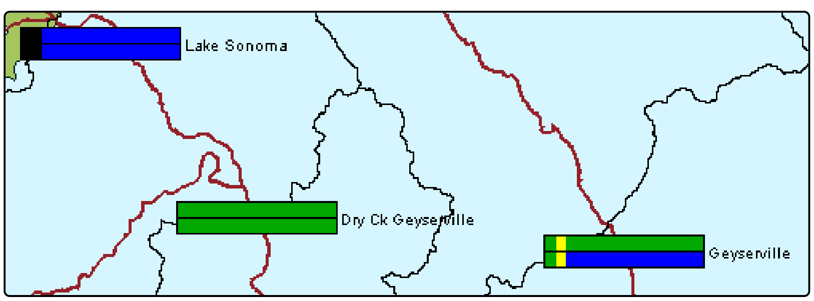
Reviewing the quality of the data is the primary task during the HEC-RTS data acquisition process. There are four primary ways to assess the quality of data, to identify specific locations with data issues.

- **Use of quality color bars and thumbnail plots.** HEC-RTS can display time series icons as quality color bars or thumbnail plots representing gages where data has been collected. Quality color bars are color coded according to the quality of data at a location. This allows you a quick visual overview of data that has been acquired in the watershed for the specified time window.
- **Use of plots and tabular data.** When you have identified locations where issues exist, plots and tables provide detailed views of data, helping you determine the nature of the issue.
- **Use of Data Summary Lists.** Data summary lists allow you to view gages in groups (such as all precipitation or all flow gages) and see details for more or fewer gages than display in the geo-referenced map window (Data Status Lists).

8.4.1 Quality Color Bars

Quality color bars (Figure 1) offer a quick overview of acquired data in the watershed, reflecting the status of data at each location as being acceptable, missing, or questionable.

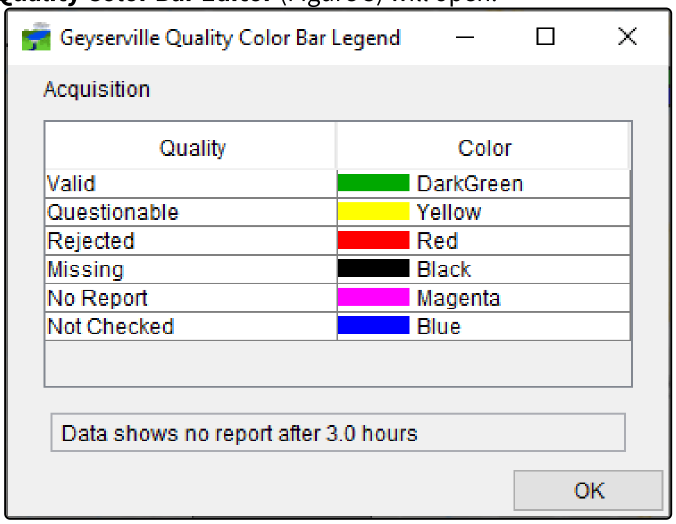
HEC-RTS attaches quality flags to the data based upon thresholds that have been set.

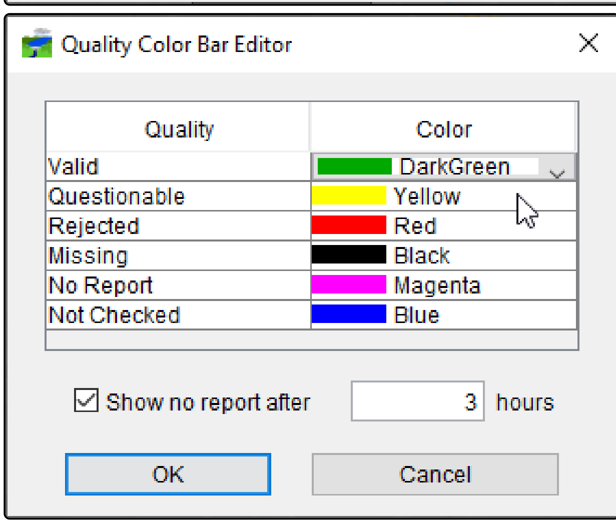


33 Figure 1 Quality Color Bars

A **quality color bar** (Figure 1) is a thick, segmented line representing the history of the data quality at a specific location within the time window you have established. The name of the location appears adjacent to the color bar, and the length of the color bar represents a timeline equal to the duration of the time window. Segments along the color bar reflect time intervals, with colors corresponding to the data quality legend you have defined. The quality color bars are updated dynamically at the time interval that you entered in the Time Window. As part of your daily routine, you should review quality color bars to determine if data is good and which data needs to be validated. To view the legend for quality color bars:

1. Right-click on a time series icon in the **Map Window**, from the shortcut menu, click **Icon Legend**, the **Quality Color Bar Legend** dialog (Figure 2) will open.
2. The legend displays the quality flags for the data and the colors that represent each quality flag. You can change the color associated with the quality flags.
3. From the HEC-RTS main window, from the **Acquisition** tab, from the **Edit** menu, click **Icon Quality Colors**, the **Quality Color Bar Editor** (Figure 3) will open.

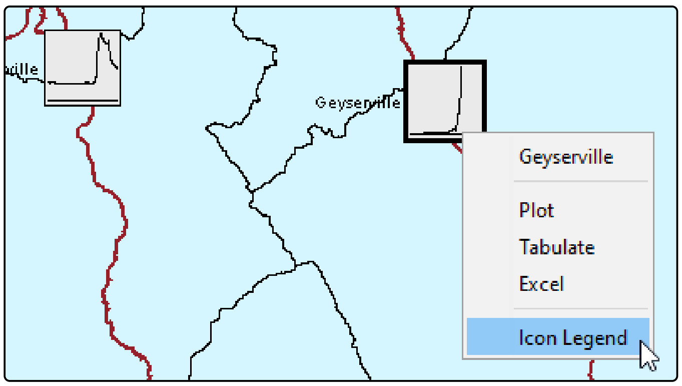




4. To change the color of a quality flag, in the **Color** column of the table click the drop-down list and a list of available colors appears.
5. Click the color you wish to change, and the color appears in the cell of the **Color** column.
6. The **Show no report after** flag (Figure 3) is a quality check for irregular time series datasets only and is not used in HEC-RTS.
7. Click **OK**, the **Quality Color Bar Editor** (Figure 3) closes, and the **Map Window** refreshes all quality color bars.

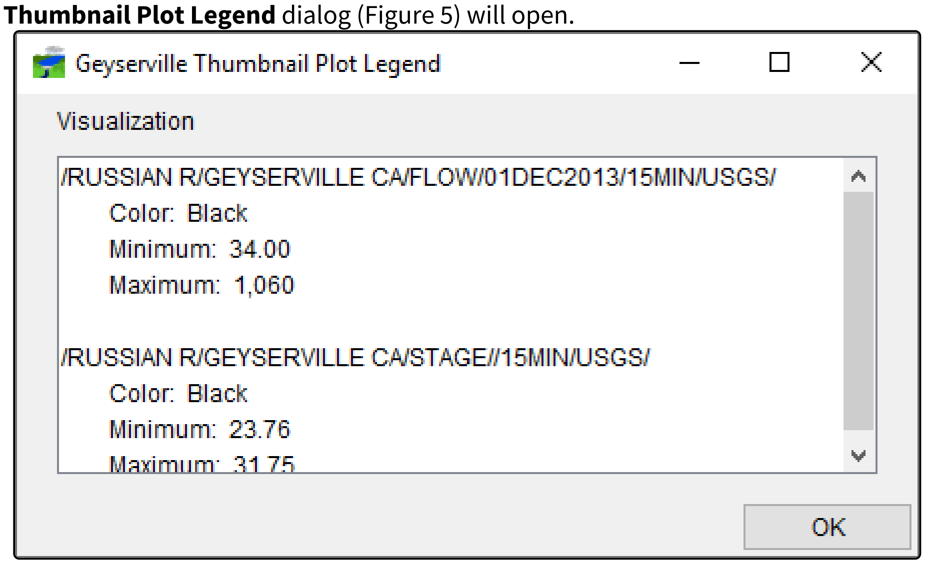
8.4.2 Thumbnail Plots

In some cases, you may wish to use thumbnail plots to represent gages and time series locations. Thumbnail plots (Figure 4) provide a miniature version of the full-size plots available in HEC-RTS, displaying data for the time window. The name of the location appears adjacent to the thumbnail plot. As the time window updates, HEC-RTS refreshes the data and updates the thumbnail plot. The thumbnail plots are updated dynamically at the time interval that you entered in the Time Window.



To view the legend for thumbnail plots:

1. From the **Map Window**, right-click a thumbnail plot, from the shortcut menu (Figure 4), click **Icon Legend**. The **Thumbnail Plot Legend** dialog (Figure 5) will open.

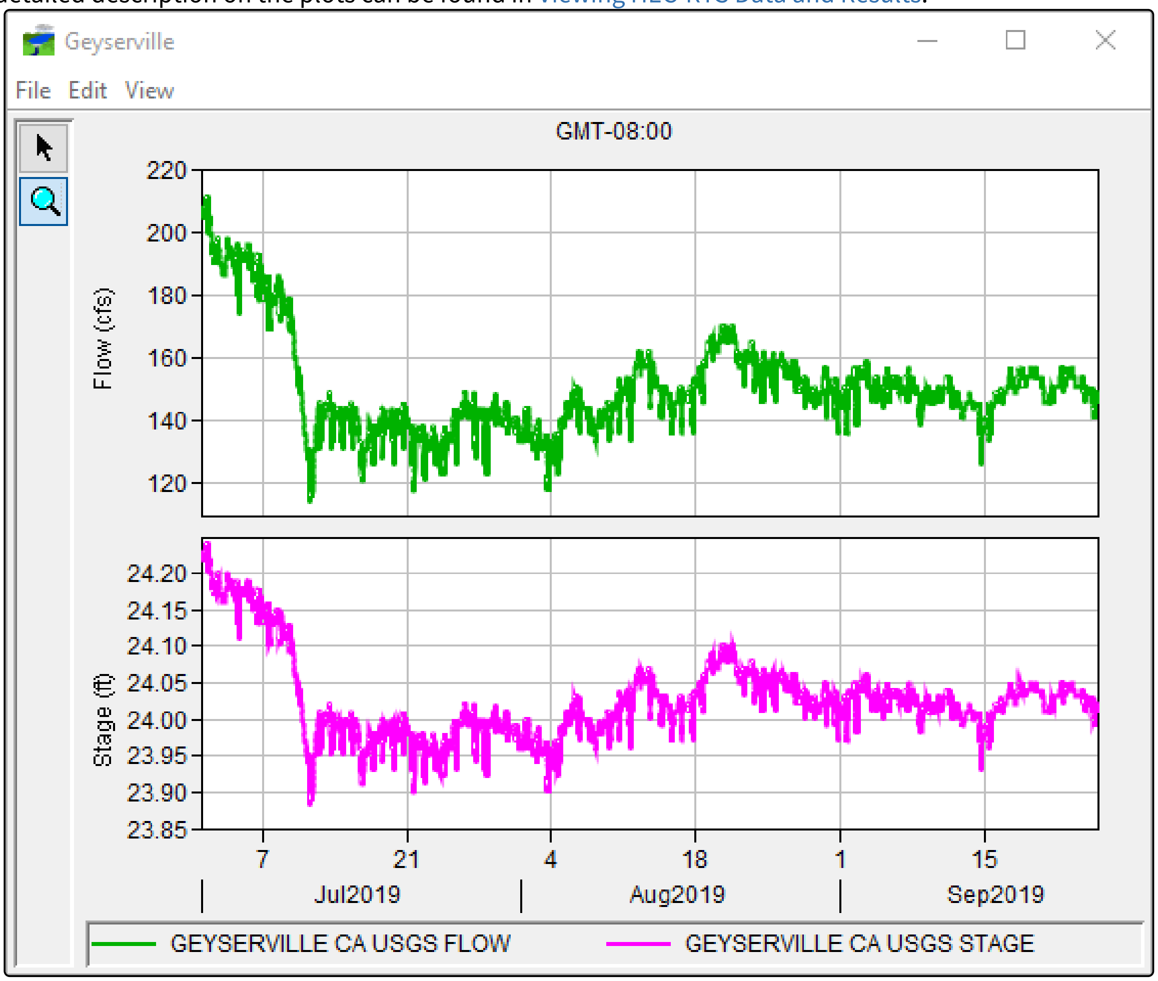


2. The legend displays the pathname, the color of the curve, and the minimum and maximum values of the curve. The **Thumbnail Plot Legend** dialog (Figure 5) provides you with information, to edit the color of the curve shown in the thumbnail plot (Time Series Icons).

8.5 Using Plots

If the quality color bar or thumbnail plot for a location indicates that the data is questionable or bad, viewing the data in a plot can help you to determine the nature of the problem. To access plots for data:

1. From the **Map Window**, right-click a time series icon, from the shortcut menu, click **Plot**.
2. This opens a two-dimensional plot(s) (Figure 1) that contains the data for that location for the current time window. Figure 1 shows sample plots for both flow and stage gages. Use the **Zoom Tool** to zoom in on a specific portion of the plot. See Map Window for a more detailed description of the **Zoom Tool**. In addition, a more detailed description on the plots can be found in Viewing HEC-RTS Data and Results.

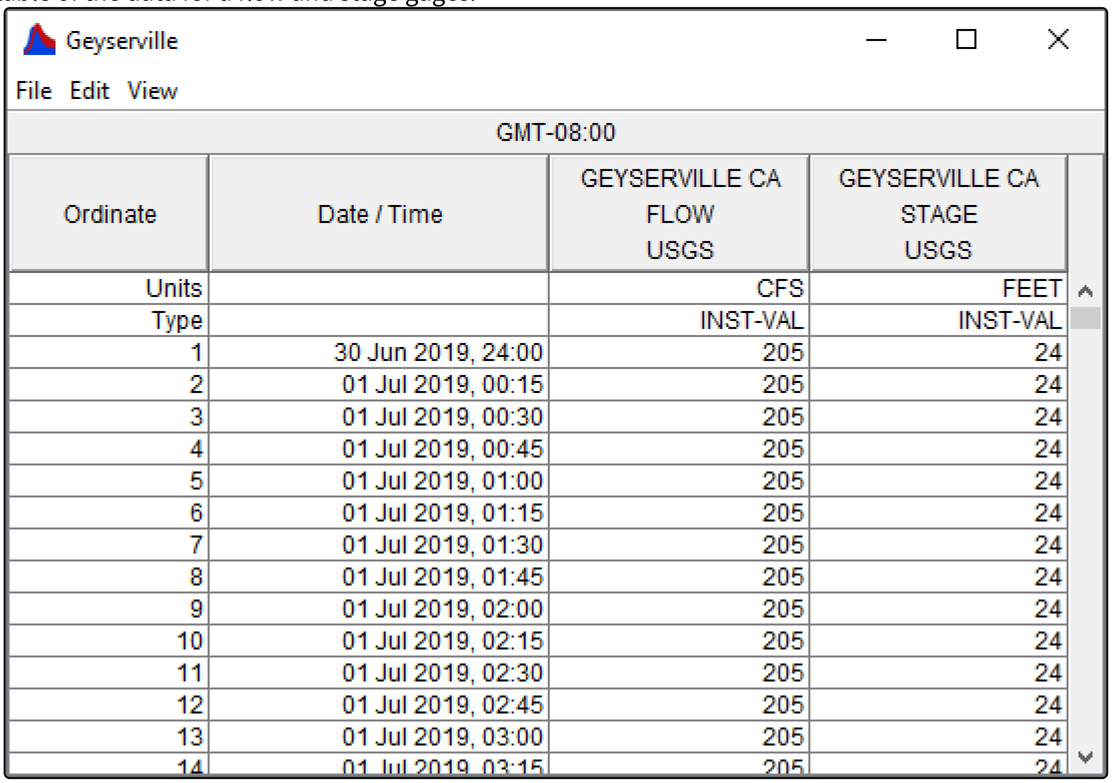


8.6 Using Tables

If the quality color bar or thumbnail plot for a location indicates that the data is questionable or bad, viewing the data in a table can help you to determine the nature of the issue. To access tables of data:

1. From the **Map Window**, right-click a time series icon, from the shortcut menu, click **Tabulate**.

2. This opens a table that contains data for that location for the current time window. Figure 1 shows a sample table of the data for a flow and stage gages.



Ordinate	Date / Time	GEYSERVILLE CA FLOW USGS	GEYSERVILLE CA STAGE USGS
Units		CFS	FEET
Type		INST-VAL	INST-VAL
1	30 Jun 2019, 24:00	205	24
2	01 Jul 2019, 00:15	205	24
3	01 Jul 2019, 00:30	205	24
4	01 Jul 2019, 00:45	205	24
5	01 Jul 2019, 01:00	205	24
6	01 Jul 2019, 01:15	205	24
7	01 Jul 2019, 01:30	205	24
8	01 Jul 2019, 01:45	205	24
9	01 Jul 2019, 02:00	205	24
10	01 Jul 2019, 02:15	205	24
11	01 Jul 2019, 02:30	205	24
12	01 Jul 2019, 02:45	205	24
13	01 Jul 2019, 03:00	205	24
14	01 Jul 2019, 03:15	205	24

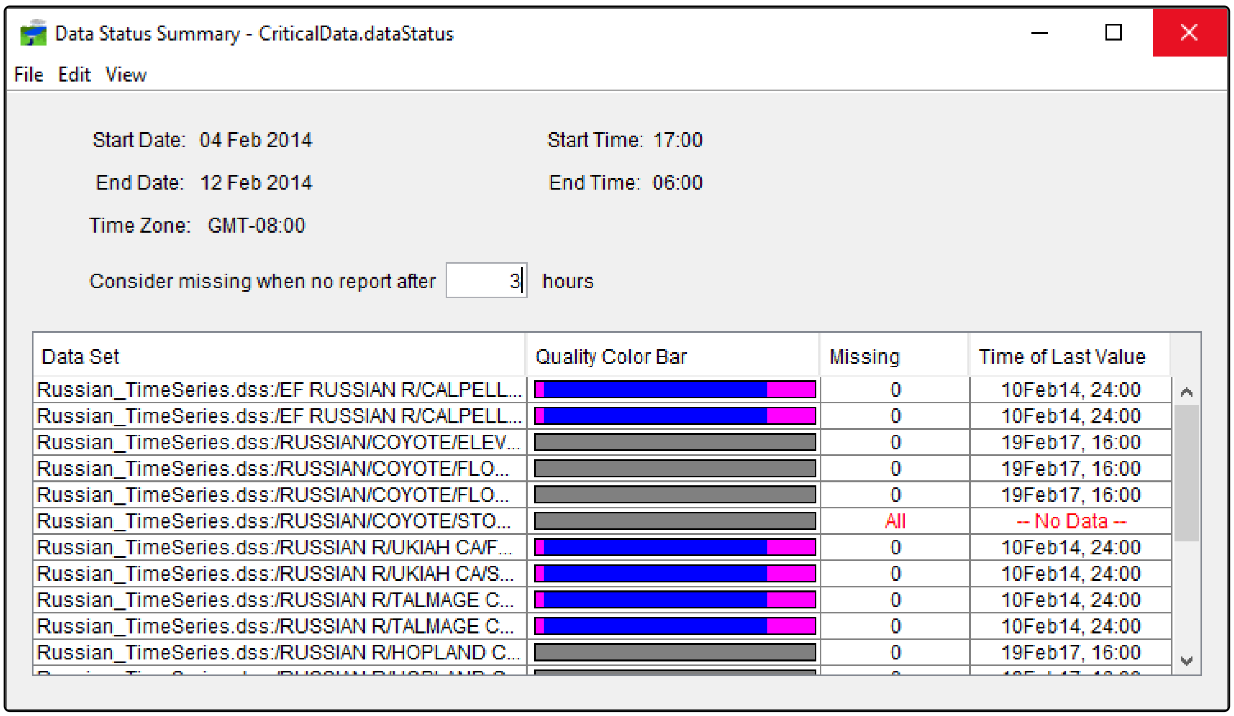
3. If you have already opened a plot, you can also tabulate the data displayed in the plot from the **Plot** dialog (Sample HEC-RTS Plot). From the **File** menu, click **Tabulate**. For more details on tabular data in HEC-RTS, see Visualizing Data. For printing and exporting tabular data, see Printing and Copying HEC-RTS Data.

9 Data Status Lists

Data status lists provide the capability to review time series data by groups (such as all precipitation or all flow gages) and see details for more or fewer time series data than what appears in the geo-referenced map display. These lists can help you assess data and identify specific locations where problems may exist in the acquired data.

9.1 Data Status Summary Dialog

To manipulate data status lists, from the **Acquisition** tab, from the **Reports** menu, click **Data Status Summary**, the **Data Status Summary** dialog (Figure 1) will open. If a data status list has been displayed before, the **Data Status Summary** dialog displays the previously viewed data status list.



34 Figure 1 Data Status Summary Dialog

From the **Data Status Summary** dialog (Figure 1), the user can create data status lists, select data status lists, copy a data status list, review the quality color bars, review how many values are missing, and find the time and date of the last valid data value for that particular time series data.

Data status lists allow the user to organize time series data for specific needs. For example, a data status list might be created that displays all precipitation gages, all stream gages, or a mix of precipitation and stream gages that are needed for a specific forecast. A data status list contains time series data (pathnames) from DSS files.

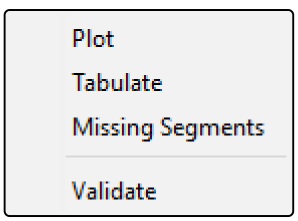
9.1.1 Menu Bar

Following is an overview of the **Menu Bar** for the **Data Status Summary** dialog (Figure 1). The commands available are:

File	From this menu you can create a data status list, open a data status list, save a data status list, or copy one data status list to another. Available commands are: **New**, **Open**, **Save**, **Save As**, and **Close**.
Edit	From this menu you can edit the table of information that represents a data status list. You can insert and delete rows, and get data from DSS files. Available commands are: **Insert Row(s)**, **Delete Row(s)**, and **Browse DSS**.
View	From this menu you can plot and tabulate specific time series data, view the missing data segments of time series data, validate data associated with a time series pathname, view the quality color bar legend, and refresh time series data when the time window for the **Data Status Summary** dialog has changed. Available commands are: **Plot**, **Tabulate**, **Missing Segments**, **Validate**, **Quality Color Legend**, **Time Window**, and **Refresh**.

9.1.2 Shortcut Menu

From the table in the **Data Status Summary** dialog (Figure 1), select a time series pathname (a row in the table), then right-click on the selected time series pathname, a shortcut menu will be displayed (Figure 2). From the shortcut menu the user can plot, tabulate, view missing data, or validate data for the selected time series pathname.



35 Figure 2 Shortcut Menu

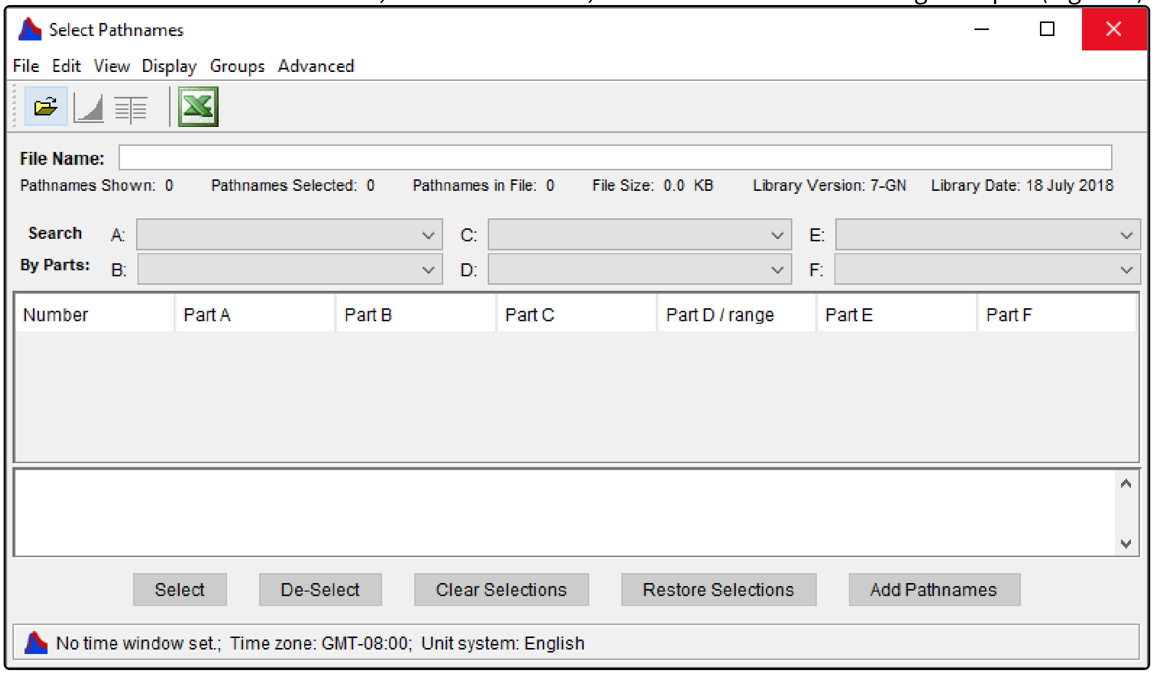
9.2 Data Status List

Data status lists allow you to review time series data by groups (such as all precipitation or all flow gages) and provide details for more or fewer time series data than what appears in the geo-referenced map display. You can create a data status list by retrieving time series data from a DSS file. Once a data status list has been created, review the data to determine the time window for your forecast, add or delete time series pathnames to the data status list, and edit data specific to the time series data.

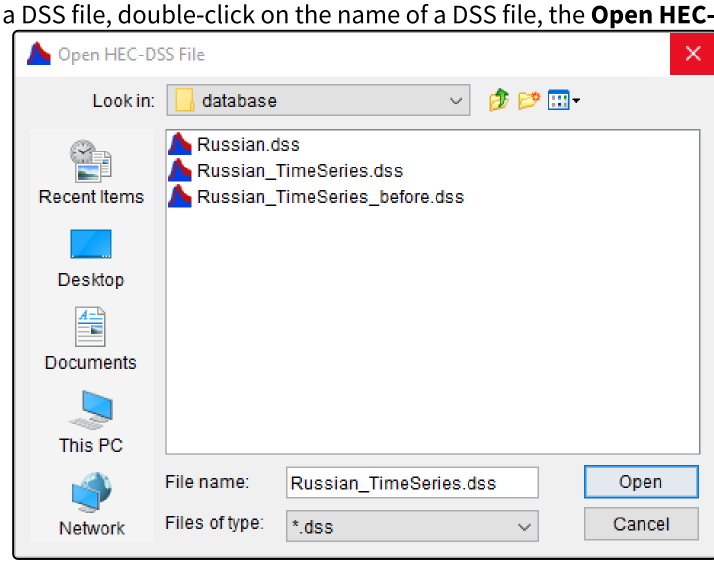
9.2.1 Creating a Data Status List

To create a data status list:

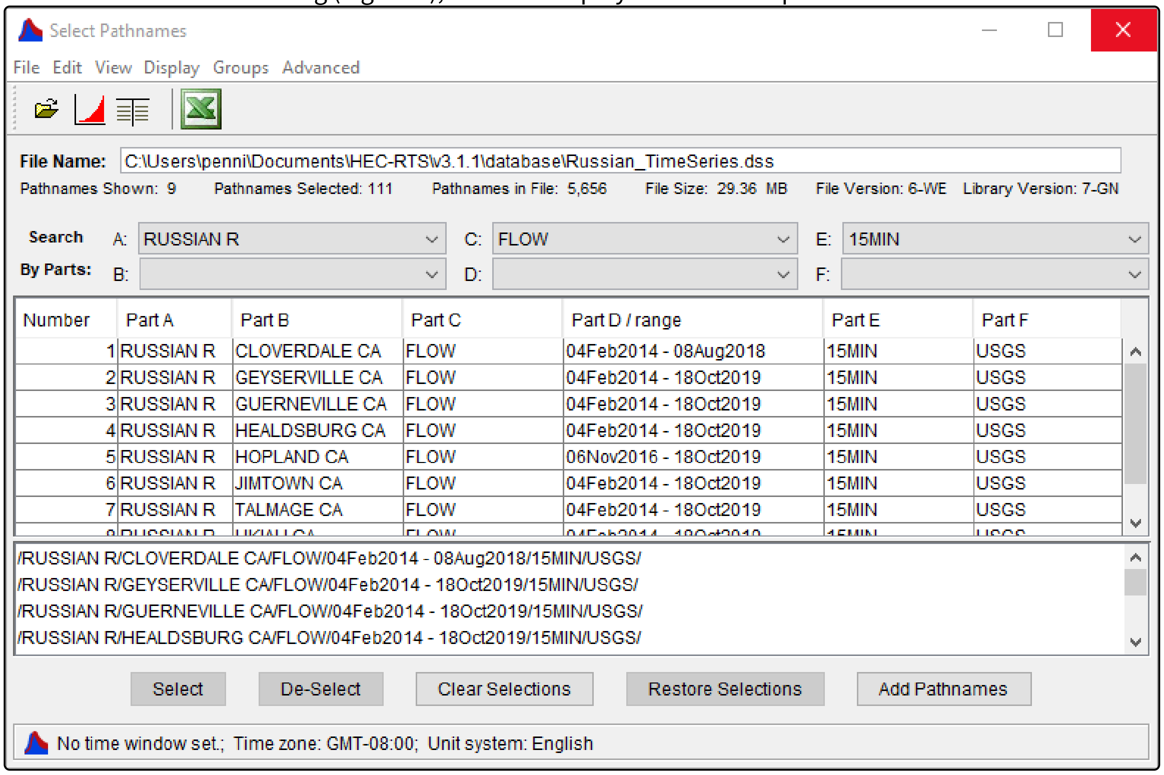
1. From the **Data Status Summary** dialog, from the **File** menu, click **New**. The **Data Status Summary** dialog will now be blank. From the **Edit** menu, click **Browse DSS**, the **Select Pathnames** dialog will open (Figure 1).



2. From the **File** menu, click **Open**, an **Open HEC-DSS File** browser will open (Figure 2). Navigate to the location of a DSS file, double-click on the name of a DSS file, the **Open HEC-DSS File** browser will close (Figure 2).

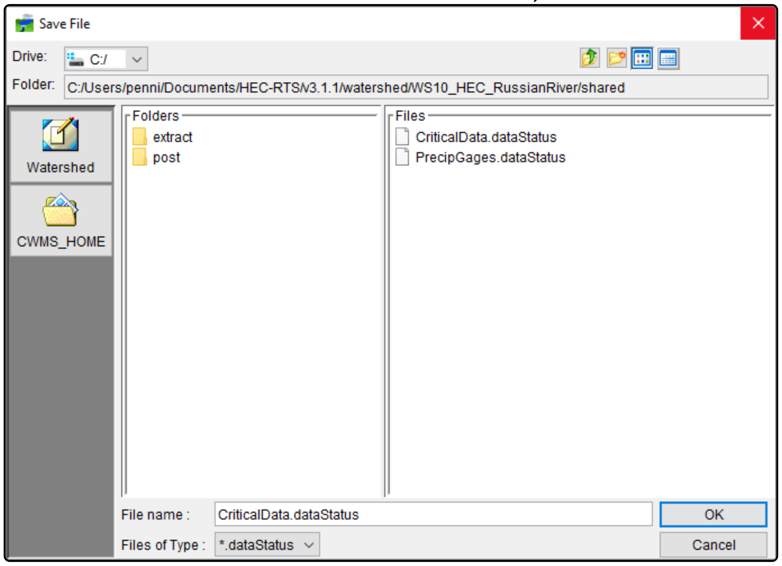


3. The **Select Pathnames** dialog (Figure 3), will now display the available pathnames in the selected DSS file.



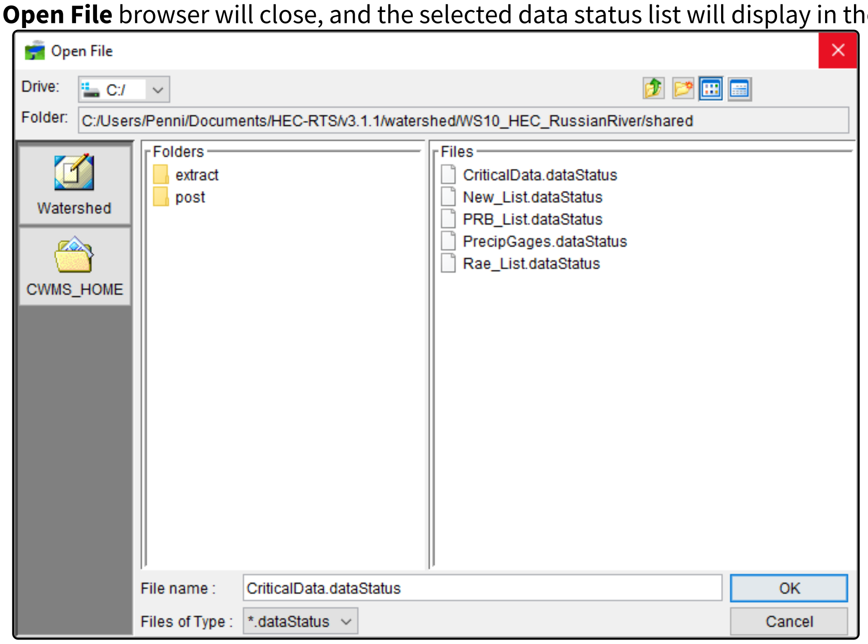
4. To add DSS pathnames to a data status list, highlight a row(s) in the table, click **Select,** your selection(s) will display in the selection box.
5. Since the selected DSS file can contain many pathnames, to assist you in finding the correct pathnames, several filter lists are available in the **Search By Parts** area of the **Select Pathnames** dialog (Figure 3).
6. When you are finished selecting your pathnames, click **Add Pathnames** (Figure 3). From the **File** menu, click **Close**, the **Select Pathnames** dialog (Figure 3) will close. The **Data Status Summary** dialog will now contain the selected pathnames.
7. To save the data status list, from the **File** menu (Data Status Summary Dialog), click **Save**. A **Save File** browser will open (Figure 4). In the **File name** box (Figure 4), by default, HEC-RTS will save to the default data status list (*CriticalData.dataStatus*). If you want to save to the default data status list, click **OK**. The **Save File** browser will close (Figure 4), and the default data status list (*CriticalData.dataStatus*) will now contain your selected pathnames.
8. If you do not want to save to the default data status list, from the **Save File** browser, enter a name in the **File name** box (Figure 4). The **Folder** box (Figure 4) will contain the default location (*C:\Users\xxxx\Documents\HEC-RTS\v3.1.1\watershed\WS10_HEC_RussianRiver\shared*) on your workstation where the new data status list will be saved. Click **OK**, the **Save File** browser will close (Figure 4), an information window will display telling you

that the data status list has been saved. Click **OK**, the information window will close.



9.2.2 Open an Existing Data Status List

To open an existing data status list, from the **Data Status Summary** dialog, from the **File** menu, click **Open**, and an **Open File** browser (Figure 5) will open. From the **Files** box select a data status list from the available files. Click **OK**, the **Open File** browser will close, and the selected data status list will display in the **Data Status Summary** dialog.



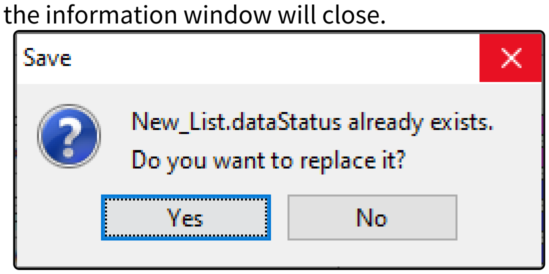
9.2.3 Make a Copy of an Existing Data Status List

To make a copy of an existing data status list, from the **Data Status Summary** dialog, from the **File** menu, click **Save As**, the **Save File** browser (Figure 4) will open. From the **File name** box, enter a filename for the new data status list. Click **OK**, the **Save File** browser will close (Figure 4), an information window will display telling you that the data status list has been saved. Click **OK**, the information window will close.

9.2.4 Adding DSS Pathnames to a Data Status List

To add DSS pathnames to a data status list:

1. From the **Data Status Summary** dialog, from the **File** menu, click **Open**, and an **Open File** browser (Figure 5) will open. From the **Files** box select a data status list from the available files. Click **OK**, the **Open File** browser will close, and the selected data status list will display in the **Data Status Summary** dialog.
2. From the **Edit** menu, click **Browse DSS**, the **Select Pathnames** dialog (Figure 2) will open. From the **File** menu, click **Open**, an **Open HEC-DSS File** browser will open (Figure 2). Navigate to the DSS file that contains the pathnames you want to add to the selected data status list. Select the DSS file, click **Open**, the **Open HEC-DSS File** browser will close.
3. The **Select Pathnames** dialog (Figure 2), will display all the available pathnames in the selected DSS file.
4. To add DSS pathnames, highlight the pathnames in the table by clicking on a row, click **Select**, the selection will display in the selection box (Figure 3). Continue adding pathnames by selecting them in the table and clicking **Select**.
5. When all pathnames have been selected, click **Add Pathnames**. From the **File** menu, click **Close**, the **Select Pathnames** dialog will close (Figure 3).
6. From the **Data Status Summary** dialog, click **Save**. A **Save** window will open (Figure 6). The **Save** window is asking do you want to replace the selected data status list. Click **Yes**, the **Save** window will close, and an information window will open. This window lets you know the selected data status list has been saved. Click **OK**, the information window will close.

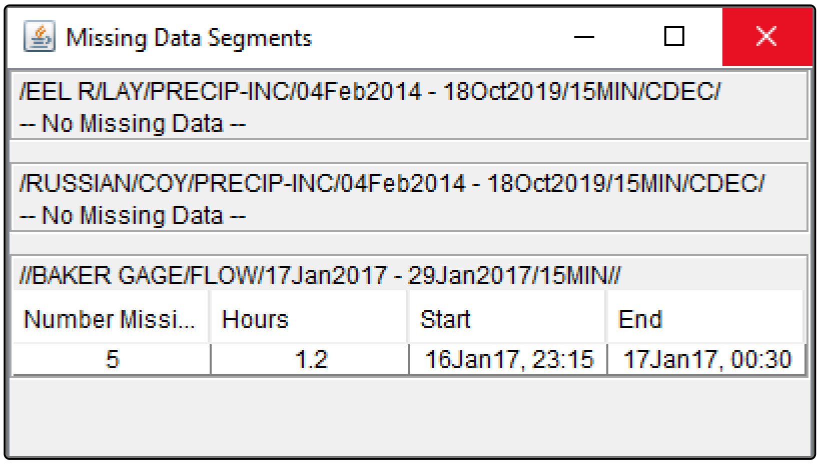


9.2.5 Delete a DSS Pathname from a Data Status List

To delete a DSS pathname from a data status list, from the **Data Status Summary** dialog, from the **Data Set** column of the table, select a pathname. From the **Edit** menu (Data Status Summary Dialog), click **Delete Row(s)**. The selected pathname will no longer display in the table on the **Data Status Summary** dialog. From the **File** menu, click **Save**, a **Save** window will open (Figure 6). The **Save** window is asking do you want to replace the selected data status list. Click **Yes**, the **Save** window will close, and an information window will open. This window lets you know the selected data status list has been saved. Click **OK**, the information window will close.

9.3 View Missing Data Segments

HEC-RTS offers another way to focus in on problem data by filtering missing data segments for individual time series pathnames. To view details about missing data segments for time series data, from the **Data Status Summary** dialog, from the table, select a DSS pathname(s). From the **View** menu, click **Missing Segments**, the **Missing Data Segments** dialog (Figure 1) will open.



36 Figure 1 Missing Data Segments Dialog

Another way to view details about missing data segments for time series data, is from the **Data Status Summary** dialog, from the table, from the **Data Set** column, right-click on a pathname, from the shortcut menu. click **Missing Segments**, the **Missing Data Segment** dialog will open (Figure 1).

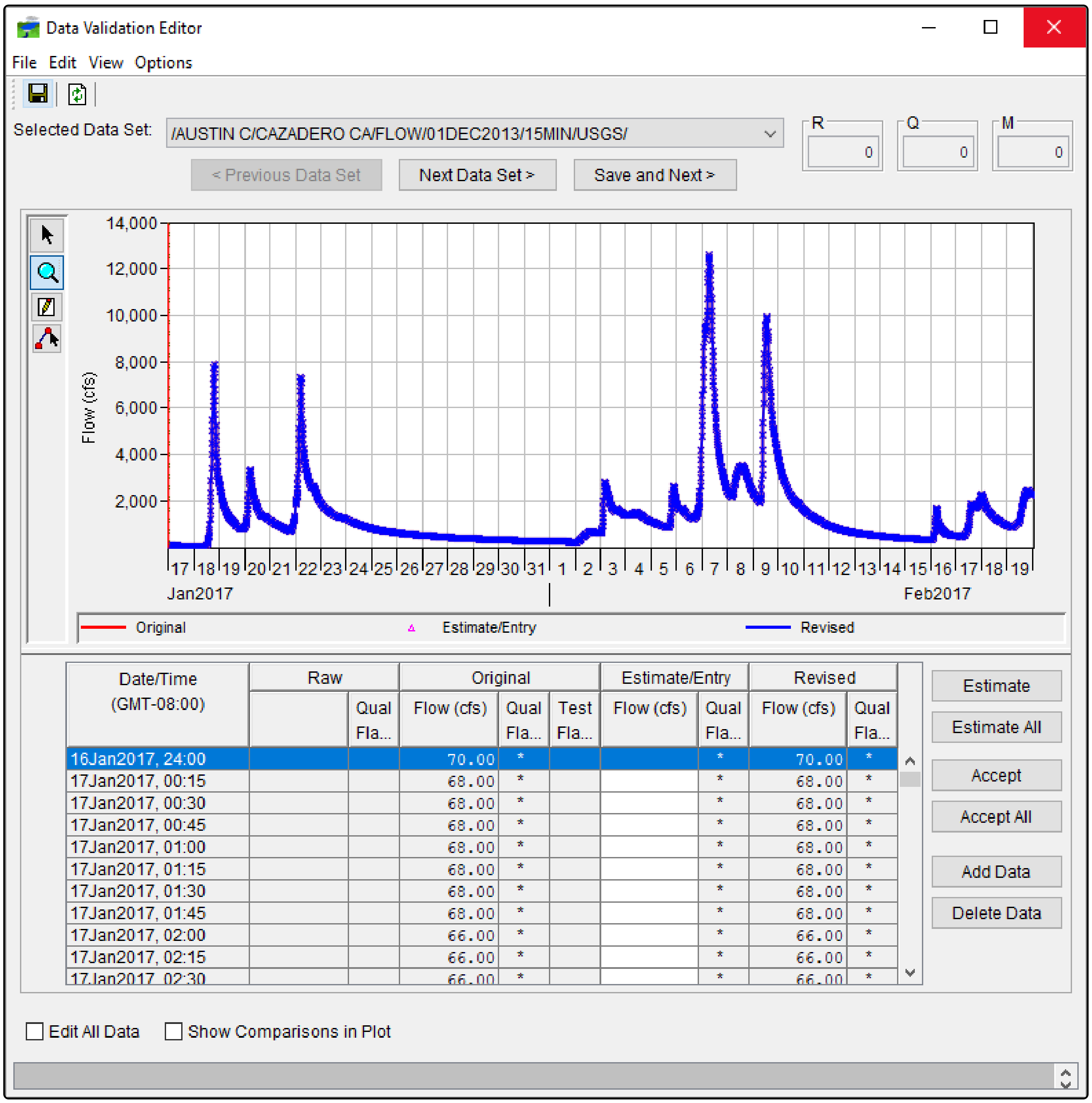
For the selected pathnames, the **Missing Data Segments** dialog (Figure 1) reports internal missing gaps for each selected pathname. The missing gaps are periods in the time series data where data values were expected, but are not present, and data values are present before and after these periods to form each gap. For time series data that are defined to have a regular interval (e.g., *1Hour*) data values are expected at each interval.

For time series data with an undefined or irregular interval (i.e., the time series interval is set to *0*) an interval will need to be defined. To set this interval, from the **Data Status Summary** dialog, enter a value (in hours) in the **Consider missing when no report after hours** box.

Missing data in critical time series data status lists has an impact on HEC-RTS decision support activities. After analyzing the information displayed in the **Missing Data Segments** dialog (Figure 1), the user may decide to open the Data Validation Editor for the analyzed information to appropriately edit/correct the missing data values.

10 Data Validation Editor

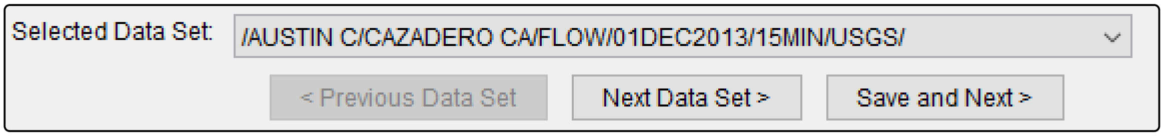
The **Data Validation Editor** (Figure 1) allows the user to select specific time series locations for data validation (see Data Status List for further details). Before manually validating data, a good idea, is to filter selected time series datasets so that only those time series datasets with identified problems will be displayed. Additionally, the **Data Validation Editor** offers tools for both text-entry editing (corrected values in a table) and graphical editing (graphically changing values in a plot).



Date/Time (GMT-08:00)	Raw	Raw Qual Fla...	Original Flow (cfs)	Original Qual Fla...	Test Fla...	Estimate/Entry Flow (cfs)	Estimate/Entry Qual Fla...	Revised Flow (cfs)	Revised Qual Fla...
16Jan2017, 24:00			70.00	*			*	70.00	*
17Jan2017, 00:15			68.00	*			*	68.00	*
17Jan2017, 00:30			68.00	*			*	68.00	*
17Jan2017, 00:45			68.00	*			*	68.00	*
17Jan2017, 01:00			68.00	*			*	68.00	*
17Jan2017, 01:15			68.00	*			*	68.00	*
17Jan2017, 01:30			68.00	*			*	68.00	*
17Jan2017, 01:45			68.00	*			*	68.00	*
17Jan2017, 02:00			66.00	*			*	66.00	*
17Jan2017, 02:15			66.00	*			*	66.00	*
17Jan2017, 02:30			66.00	*			*	66.00	*

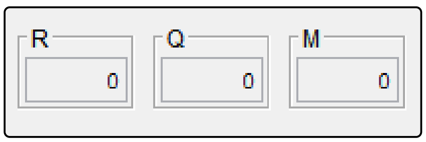
37 Figure 1 Data Validation Editor

From the **Selected Data Set** list (Figure 2), the user can select the specific time series dataset that will be viewed or modified.



38 Figure 2 Selected Data Set List

By default, the **Data Validation Editor** (Figure 1) only allows the editing of data that has been marked as being rejected, questionable, or missing. The **R** (rejected), **Q** (questionable), and **M** (missing) fields (Figure 3) are related to the quality flags of the data and displays counts of the data values in the **Original** column of the tabular data area for the selected dataset and time window. If the user needs to edit the data, from the **Data Validation Editor** (Figure 1), click **Edit All Data**, or from the **Options** menu (Figure 1), click **Edit All Data**.



39 Figure 3 Data Quality Flags

To view comparison time series datasets from the **Data Validation Editor** (Figure 1) click **Show Comparison in Plot**, or from the **View** menu, click **Show Comparison Data**.

To save the revised data values to DSS file(s), click  , or, from the **File** menu (Figure 1), click **Save**. To undo all of the changes made to either the graphical or tabular data areas since the last **Save**, click  . (**Note:** Once you click **Save**, you cannot undo any previous changes.)

10.1 Menu Bar

Following is an overview of the **Menu Bar** for the **Data Validation Editor**. The commands available are:

File	Provides options for printing and saving changes to DSS file(s) and closing the **Data Validation Editor**. Available commands are: **Save**, **Print Table**, **Print Graph**, **Page Setup**, and **Close**.
Edit	During an editing session, you can set flags for restoring original data, and automatically make can estimate for missing data. In addition, you an extend the selected time series dataset before the time window start date and/or after the end data. Available commands are: **Restore Original Data**, **Always Estimate Missing**, and **Add Data**.
View	Allows comparison data if a comparison time series dataset has been included, and you can specify the number of decimal places that will be displayed in the **Data Validation Editor**. Available commands are: **Show Comparison Data** and **Show Decimal Places**.

<table>
<tr>
<td>Options</td>
<td>Provides you with an option for editing any of the displayed data; an option that allows you to continue working while data is being saved; by default, scale plots so that all data values are visible in a plot; and, provides an option that allows you to specify how changes will be saved to DSS file(s). Available commands are: Edit All Data, Save in Background, Scale Plot to Raw Data, and Storage Rule.

From Storage Rule, you have the following commands available:
<table>
<tr><td>Delete Insert</td><td>removes all unprotected data from the data's time window, merges time values between protected data, and updates unprotected data at corresponding times with data. Data is discarded when protected data exists at corresponding times.</td></tr>
<tr><td>Replace All</td><td>merges time values between data and replaces unprotected data at corresponding times with data. Data is discarded when there is protected data at corresponding times.</td></tr>
<tr><td>Do Not Replace</td><td>merges time values between data and discards data when there is existing data at corresponding times.</td></tr>
<tr><td>Replace Missing Values Only</td><td>merges time values between data and replaces unprotected missing data at corresponding times with data. Data is discarded when protected data exists at corresponding times.</td></tr>
<tr><td>Replace with Non Missing</td><td>merges time values between data and replaces unprotected data at corresponding times with non-missing data. Data is discarded when protected data exists at corresponding times.</td></tr>
</table>
</td>
</tr>
</table>

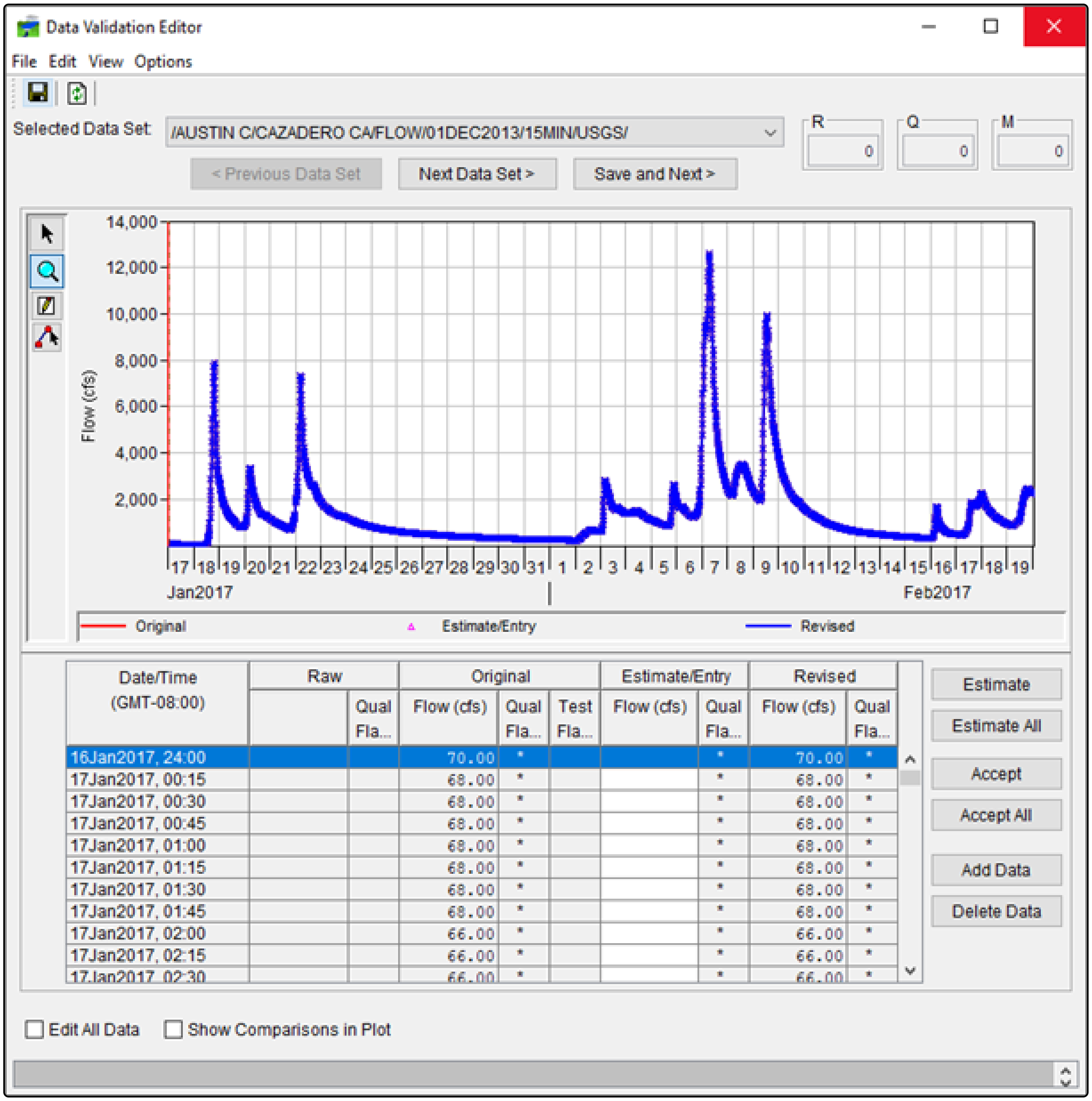
10.2 Time Window

The time window used by the **Data Validation Editor** is the same time window that is set for monitoring data in the **Acquisition** module. See Set Time Window for information for more information.

10.3 Graphical Editing Area

The graphical editing area (Figure 1) displays the selected dataset graphically. By default, a blue line represents data values that have been validated, and a green line represents the original data values in DSS file(s).
By default, the data shown in the plot, scales so that all data values are visible in the plot, this is because the data could

contain erroneously large or small data values. From the **Options** menu (Data Validation Editor), if the user turns off the **Scale Plot to Raw Data** flag, the plot window will scale to the validated data, potentially leaving any original data beyond the validated value bounds outside of the plot window.



40 Figure 1 Graphical Editing Area

The data that is visible in the tabular data area is highlighted as a green-hatched column in the graphical editing area (Figure 1). A vertical black line (Figure 1) in the graphical editing area, or running through the green-hatched column in the graphical edit area, indicates the specific record in the tabular data area you have selected.

10.3.1 Graphical Editing Tools

The **Graphical Editing Tools** on the Data Validation Editor allow viewing and editing of data in the graphical editing area. The available tools are:

 **Pointer/Selector Tool**

The **Pointer/Selector Tool** allows the user to navigate in the graphical editing area and move quickly through a large dataset. Select one data point then hold the shift key and select another point in the graphical editing area to select a range of data points.

 **Zoom Tool**

The **Zoom Tool** allows the user to magnify specific sections in the graphical editing area. To use the **Zoom Tool**, click and drag a rectangle around the area to be magnified. If the user needs to zoom out, right-clicking repeatedly will progressively reverse the zoom magnification.

 **Single Point Edit Tool**

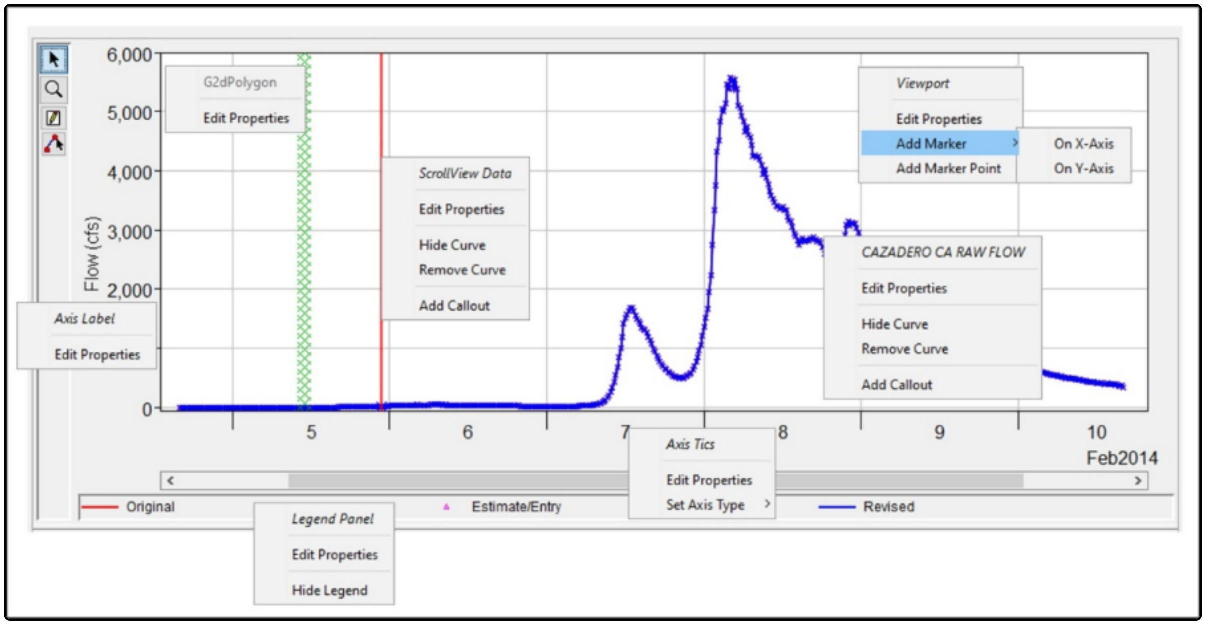
The **Single Point Edit Tool** allows the user to select an individual or a range of data points  , execute estimates  , and to set quality flags in the graphical editing area.

 **Multi-Point Edit Tool**

With the **Multi-Point Edit Tool**, the user can edit data by drawing new plot lines in the graphical editing area.

10.3.2 Shortcut Menus

The user can customize the appearance of the graphical editing area by using several properties editors (Figure 2). These properties editors are accessed by right-clicking on the different elements of the plot, and from the shortcut menu, clicking **Edit Properties**. The available shortcut menus and properties editors are listed below:

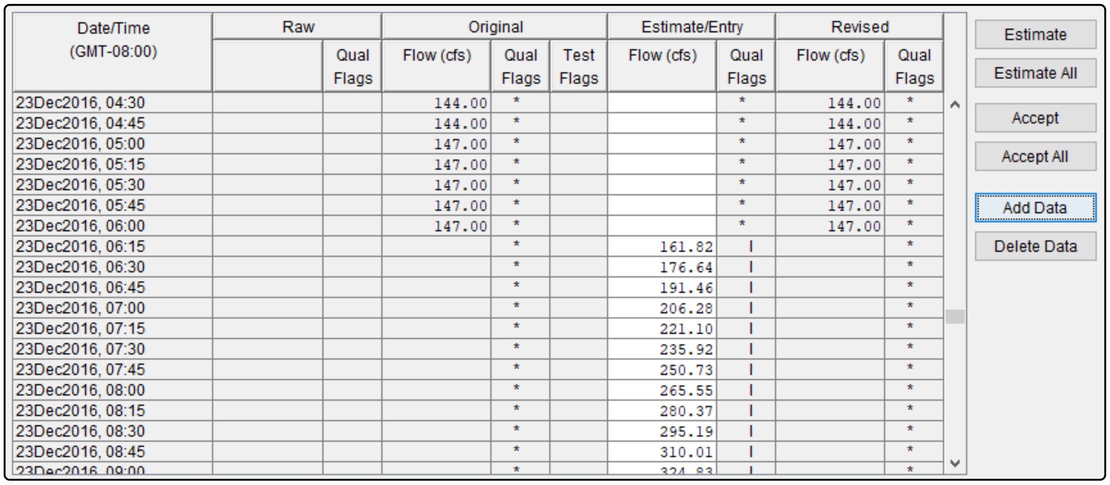


41 Figure 2 Graphical Editing Area - Shortcut Menus

Polygon Properties	This editor (Figure 2) will allow you to customize borders and backgrounds of the green-hatched column in the graphical edit area (*G2dPolygon* shortcut menu) of the Data Validation Editor.
Viewport Properties	This editor (Figure 2) will allow you to customize the border, background, and gridlines of the graphical edit area, and also add markers (*Viewport* shortcut menu) to the Data Validation Editor.
Axis Properties	This editor (Figure 2) will allow you to customize the axis scale, tic marks, and axis type (*Axis Tics* shortcut menu) of the Data Validation Editor. Also, if you click on an axis title, from this editor (Figure 2), you will be able to view the axis label, change the font, alignment of the axis title, add a border, and add color (*Axis Label* shortcut menu) to the Data Validation Editor.
Legend Properties	This editor (Figure 2) will allow you to move the legend to the right of the plot or to the bottom of the plot, or hide the legend (*Legend Panel* shortcut menu) of the Data Validation Editor.
Curve Properties	This editor (Figure 2) will allow you to edit various properties of the displayed curves in the Data Validation Editor. You can display, remove, change the color, add label, show in legend, and many other properties that can be manipulated (*Curve Name* shortcut menu).
Specific Record Properties	This editor (Figure 2) will allow you to customize the vertical red line (Figure 2) that indicates the specific record in the tabular data area of the Data Validation Editor that you have selected (*ScrollView Data* shortcut menu).

10.4 Tabular Data Area

The tabular data area (Figure 1) is a table that allows you to do text-entry data editing. The following is a description of the columns available from the table:



42 Figure 1 Tabular Data Area

- The **Date/Time** column (Figure 1) specifies the time zone for the data and lists the specific date and time for each data point.
- The **Raw** column (Figure 1) will display the data that was originally entered into a DSS file(s), along with any quality flags.
- The **Original** column (Figure 1) contains the validated data that you are reviewing, again along with any quality flags.
- The **Estimate/Entry** column (Figure 1) allows you to modify data values. When cells in this column are white, you may edit the data. When cells in this column are gray, the rows are protected as being acceptable, having passed validation testing. To allow editing, select **Edit Acceptable Data** or select **Edit Acceptable Data** from the **Options** menu.
- The **Revised** column (Figure 1) displays the values you have accepted. The corrected values appear in the **Revised** column, along with quality flags.

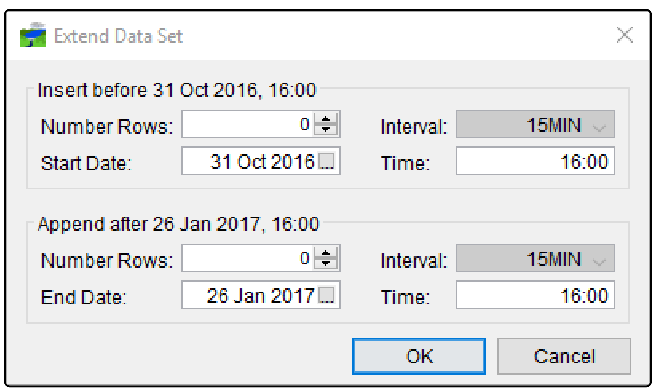
Table 1 Quality Flags for Data Validation

Quality Flag	
*	no quality (data has not been validated)
Q	Questionable
P	Protected

K	keyboard entry
E	estimate (graphical)
I	interpolated estimate
NI	negative incremental (rate of change)
R	Rejected
M	set to missing
A	explicit form of blank for manual entry of quality
+	inserted value
	blank; indicates value passed all tests

10.4.1 Inserting/Deleting Values

To add or delete data values in the tabular data area, from the **Data Validation Editor**, first from the **Options** menu, select **Edit All Data**. When selected the **Add Data** and **Delete Data** buttons become visible (Figure 1). To insert a data value, click **Add Data** (Figure 1). The **Extend Data Set** dialog (Figure 2) will open.



43 Figure 2 Extend Data Set Dialog

Through the **Extend Data Set** dialog (Figure 2) you can insert data into the time window before the time window dates or append data to the time window.

Insert Before In the **Insert Before *DD MMM YYYY, HH:MM* and *DD MMM YYYY, HH:MM*** section (Figure 2) of the **Extend Data Set** dialog, select the number of rows you would like to insert using the **Number Rows** selector. The **Start Date** and **Time** fields will adjust their values according to the number of rows you select to enter. Click **OK**, the **Extend Data**

Set dialog closes (Figure 2) and your new rows will be generated and added to the beginning of the table. You may then enter values in the table for the added time periods.

Append After In the **Append After *DD MMM YYYY, HH:MM*** section (Figure 2) of the **Extend Data Set** dialog, select the number of rows you would like to append after the time window end date using the **Number Rows** selector. The **End Date** and **Time** fields will adjust their values according to the number of rows you select to enter. Click **OK**, the **Extend Data Set** dialog closes (Figure 2) and your new rows will be generated and added to the end of the table.

You may then enter values in the table for the added time periods.

If you wish to delete a data value, select a row from the table, click **Delete Data** (Figure 1), the row will no longer appear in the table. When deleting you will not be warned, so be sure you know that you want to delete the data value before clicking **Delete Data**.

For selected rows, **Accept** (Figure 1), will accept the changes for those selected rows and copy the information to the **Revised** column (Figure 1) of the tabular data area. Your changes are stored in memory and are not yet saved to DSS file(s). Additionally, you can use the **Accept All** (Figure 1) to save and accept all applicable data within the table.

10.4.2 Other Editing Commands

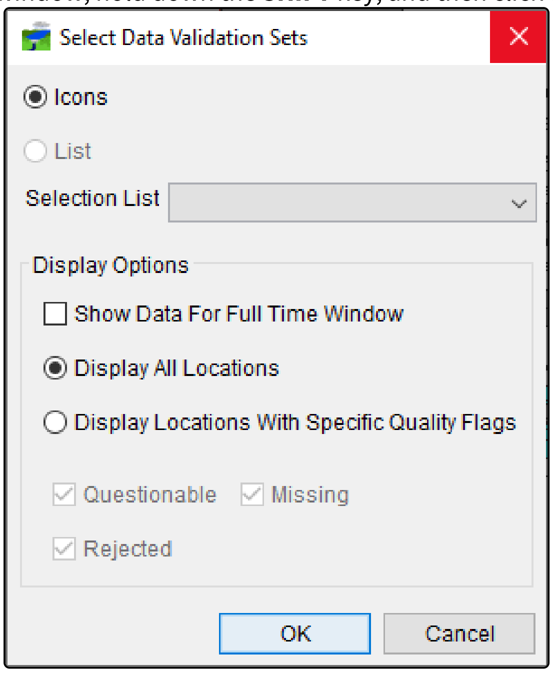
Estimate (Figure 1) will automatically compute a data value for selected rows based on acceptable surrounding data in the **Original** column (Figure 1). You can also multi-select rows (tabular editing) or data values (graphical editing) for estimation, provided that the selected rows are bracketed by acceptable values. Additionally, you can use the **Estimate All** (Figure 1) to apply the estimate to all applicable data within the table.

10.5 Validation Mode - Icons

There are two data validation modes, the **Icons** mode, and the **List** mode. For HEC-RTS, you will be using the **Icons mode** only. The **List** mode is based on validation lists, which are not available in HEC-RTS.

The **Icons** mode is where the selection of time series gages for manual validation occurs from the map window or from the color bars in the **Data Status Summary** dialog. A validation mode is selected from the **Select Data Validation Data Sets** dialog (Figure 1). In addition, the user can control whether a long list of time series datasets should be filtered to only include those time series datasets with specific data quality indicators. These filters are available in the **Display Options** group of the **Select Data Validation Data Sets** dialog (Figure 1). This dialog opens any time the validation of data occurs in the watershed.

1. By default, all the datasets associated with all the defined time series icons in the map window will be selected.

2. If you just want a selected subset of datasets, you simply select the time series icons you want to validate. From the map window, hold down the **SHIFT** key, and then click on the time series icons you want to manually 

validate.
3. Right-click on a time series icon that you have selected, from the shortcut menu click **Validate**.
4. The **Select Data Validation Sets** dialog (Figure 1) opens. **Icons** has been selected, click **OK**. The **Select Data Validation Sets** dialog will close, and the **Data Validation Editor** will open.
5. The **Data Validation Editor** will display data from the time series icons that you selected.

10.6 Editing Data

From the **Data Validation Editor** the user can edit data graphically (graphical editing area); enter data in the table (tabular data area); have HEC-RTS estimate data values based on surrounding data; and, you can accept questioned or rejected data. When you have finished editing data, you can save (i.e., commit) results back to DSS file(s). Whether editing the data graphically or via the tabular display, HEC-RTS keeps the graphical and tabular areas synchronized as the user navigates and make changes.

10.6.1 Graphical Editing

Edit data graphically by using the graphical editing tools. However, the user cannot execute estimates for acceptable quality data with the graphical editing tools. Editing data graphically can also be accomplished by using the **Single Point Edit Tool** or the **Multi-Point Edit Tool**.

To change an existing point with the **Single Point Edit Tool**  :

1. From the **Data Validation Editor**, select the **Single Point Edit Tool**. The **Data Validation Editor** needs to be in the edit mode.
2. From the bottom of the **Data Validation Editor**, select **Edit All Data**, for the **Options** menu, click **Edit All Data**.
3. Point to a location in the graphical editing area and double-click, the point becomes a pink triangle.

4. To move the point, select the **Pointer/Selector Tool**  , click and drag the pink triangle. You will only be able to move the point up and down. As you move the point, the numerical value of the data point will change (see the tabular data area).
5. When you get the place where you want the point, right-click on the pink triangle. From the shortcut menu, click **Accept**. The **Quality Flag** will automatically be set to estimate, and the data will be protected.
6. A red line will be drawn where the old data value for the point was, and the blue line will connect to the new position of the point.
7. If you know that the new data value is correct, click  to save the new data value to the selected DSS file.

With the **Multi-Point Edit Tool** , the user can produce estimates for data along multi-point lines:

1. From the **Data Validation Editor**, select the **Multi-Point Edit Tool**. The **Data Validation Editor** needs to be in the edit mode.
2. From the bottom of the **Data Validation Editor**, select **Edit All Data**, for the **Options** menu, click **Edit All Data**.
3. Click the mouse where you want multiple new values added along a line. You must move progressively right, moving ahead in time.
4. At the end of the line, click and then right-click. You will now have a pink triangle wherever you clicked the mouse.
5. If you are satisfied with the values, click **Accept**. The **Quality Flag** will automatically be set to estimate, and the data will be protected.
6. A red line will be drawn where the old data values were, and the blue line will connect to the new position of the points.
7. If you know that the new data values are correct, click  to save the new data values to the selected DSS file.

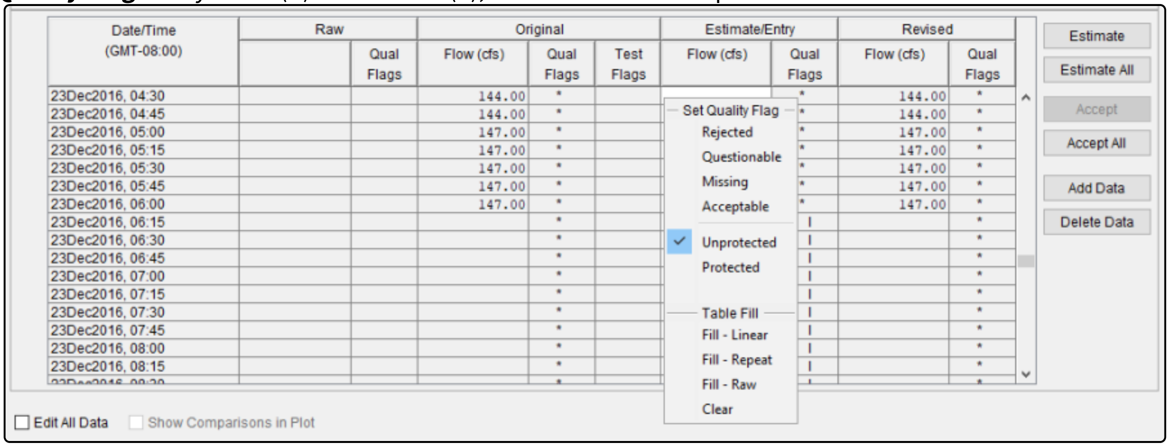
10.6.2 Tabular Editing

When the **Data Validation Editor** opens, you are directly on the first problem data point, if any. In the tabular data area, the **Estimate/Entry** column allows the modification of numerical data values. When cells in this column display with white backgrounds, data can be edited.

To edit data in the table:

1. Be sure to put the **Data Validation Editor** in edit mode, if you need to edit data that is flagged acceptable, from the **Options** menu (**Data Validation Editor**), click **Edit All Data**.
2. To edit data, enter values in the **Estimate/Entry** column (data is given a quality flag of keyboard, K), or select a cell and click **Estimate** (data is given a quality flag of estimate, E), this will create a data value that is based on the surrounding data values.

3. If satisfied with the values, click **Accept**. The new data value will display in the **Revised** column (Figure 1) with a **Quality Flag** of keyboard (K) or estimate (E), and the data will be protected.



4. A red line will be drawn where the old data values were, and a blue line will connect to the new position of the points.
5. If the new data values are correct, click to save the new data values to DSS file(s).

From the tabular data area there is a shortcut menu (Figure 1) that allows the user to set quality flags on data, whether the data is protected or not, and, provides two ways to fill in data values on the table.
Using **Fill - Linear**:

1. This will produce a linear interpolation of values between two values in the **Estimate/Entry** column (Figure 1).
2. Enter values in the first and last rows of the range of interest to create two base values (or use **Estimate** to create the new values).
3. Select the range of rows, starting with the first base value and ending with the second base value. Select a range of rows either by clicking on a row and dragging, or by clicking one row and then selecting another by holding down the **SHIFT** key.
4. Right-click, from the shortcut menu click **Fill – Linear** (Figure 1). The cells in the **Estimate/Entry** column of the selected rows will now contain data values produce from a linear interpolation between the two base values.
5. If satisfied with the values, click **Accept**. The new data values will appear in the **Revised** column (Figure 1) with a **Quality Flag** of keyboard (K) or estimate (E), and the data will be protected.
6. A red line will be drawn where the old data values were, and a blue line will connect to the new position of the points.
7. If the new data values are correct, click to save the new data values to DSS file(s).

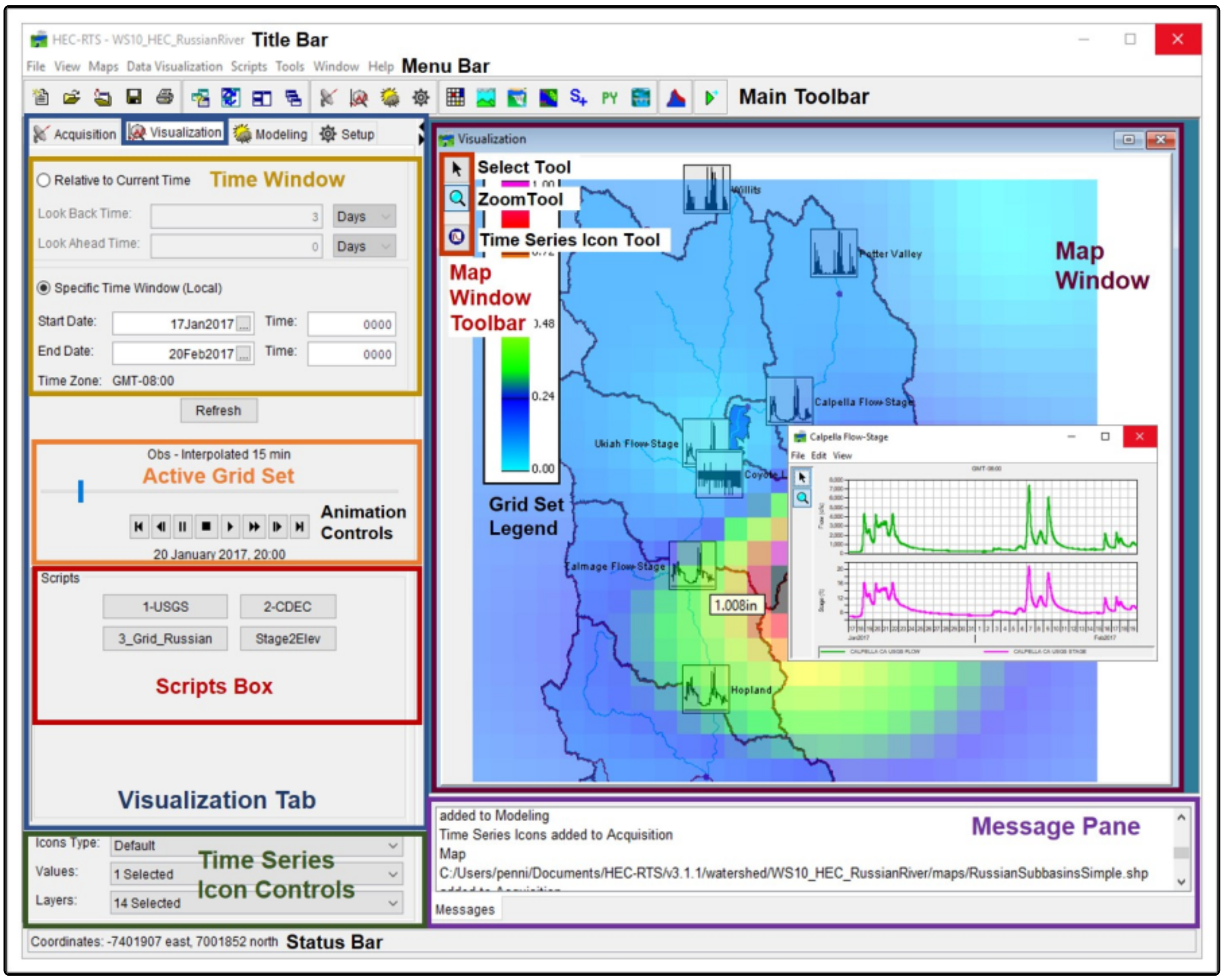
Using **Fill - Repeat**:

1. This will duplicate the selected value in all the selected cells in the **Estimate/Entry** column (Figure 1).
2. Enter a new value in the first row (or use **Estimate** to create the new value).
3. Select the range of rows, starting with the base value. Select a range of rows either by clicking on a row and dragging, or by clicking one row and then selecting another by holding down the **SHIFT** key.
4. Right-click, from the shortcut menu (Figure 1), click **Fill – Repeat**. The cells in the **Estimate/Entry** column of the selected rows will now contain the same value as the data that was entered for the base value.
5. If satisfied with the values, click **Accept** and the new data values will appear in the **Revised** column with a **Quality Flag** of keyboard (K) or estimate (E), and the data will be protected.
6. A red line will be drawn where the old data values were, and a blue line will connect to the new position of the points.

7. If the new data values are correct, click  to save the new data values to DSS file(s).

11 Visualization Module

The **Visualization** module (Figure 1) of HEC-RTS displays observed data and results of computations. This module allows you to monitor and evaluate the current state of the watershed. Spatial data, such as contours or grid cells, and time series data, such as hydrographs, are displayed in the **Map Window** (Figure 1). As in the Acquisition module, time series icons (thumbnail plots) in the **Map Window** can be set up to change color, allowing you to visualize changes in the watershed's hydrology. For further details refer to Time Series Icons and Time Series Icons Layers.



44 Figure 1 Visualization Module

11.1 Visualization Module Overview

The Visualization module allows you to view current conditions in a watershed and the associated hydrometeorological data in a georeferenced context. From the HEC-RTS main window, click the **Visualization** tab (Visualization Module). The map window and components of the Visualization module are illustrated in Visualization Module. A detailed discussion of common screen components is presented in HEC-RTS Interface.

11.1.1 Menu Bar

Following is an overview of the menu bar for the Visualization Module. The **File**, **View, Maps, Scripts, Tools**, **Windows**, and **Help** menus are discussed in HEC-RTS Interface. The commands available from the menus will allow you to view current conditions in a watershed and the associated hydrometeorological data in a geo-referenced context.

Data Visualization	This menu provides access to the gridded data, which represents the precipitation for the watershed. The only available command is **GriddedData**, which includes a sub-menu that allows you to add/edit/delete grid sets (a sequence or time series of grids); cumulate the gridded precipitation for the established time window; provide a list of grid sets, with the active grid set having a check mark; and, do not display grid sets. Available commands are: **New Grid Set**, **Edit Grid Set**, **Delete Grid Sets**, **Cumulative Display**, **Animation Delay**, and **None**. Also, if any grid sets have been created, those available grids sets will be displayed above the **None** command.

11.1.2 Commands

The **Visualization** tab (Visualization Module) contains commands specific for the Visualization module. These commands include updating time series icons (**Refresh**) in the map window and grid animation controls for visualizing gridded precipitation for the watershed.

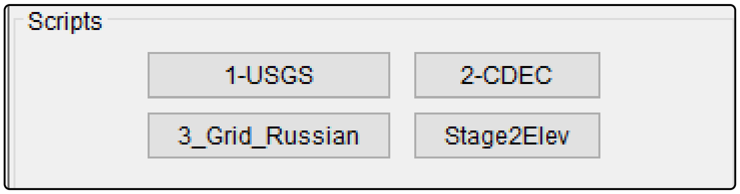
11.1.3 Map Window

The map window (Visualization Module) in the Visualization module contains map layers and layers of time series icons that allow you to visualize the current conditions of a watershed in a geo-referenced context. The primary function of the Visualization module is to provide you with a meaningful visual representation of your observed data. Threshold color bars or thumbnail plots are used to reflect the current conditions of a watershed, and, are usually used/configured for those time series locations that you watch daily or are of critical importance within your watershed. On the other hand, you might wish time series icons to display as space-conserving graphical images or dots if you are monitoring data acquisition for an entire office. Customizing the time series icon layers is done in the setup module, which is described in Time Series Icons Layers.

In most cases, for the Visualization module, you should configure time series icons to display as threshold color bars. Threshold color bars display data in multi-colored strips, such as green, amber, and red (colors can be specified by the user), that allow you to quickly home-in on those locations that are exceeding critical threshold limits.

11.1.4 Scripts Box

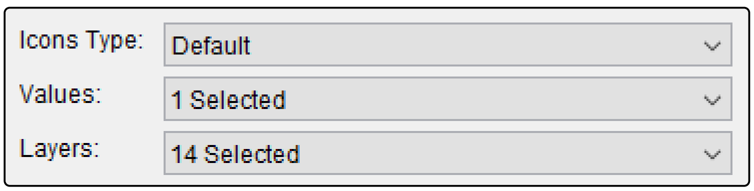
In HEC-RTS, scripts provide a way to automate HEC-RTS for the purpose of executing a prescribed set of actions with a simple trigger. Scripting simplifies user operation, ensures repeatability and consistency of results, reduces time required to generate results, and permits schedules of an operation in the future. Once scripts have been defined (Scripting), the user can run the scripts from the **Scripts Box** (Figure 1). The user defines which scripts will be run from the **Scripts Box**; Scripting provides further details on the management of scripts in HEC-RTS.



45 Figure 1 Scripts Box

11.1.5 Time Series Icon Controls

The **Time Series Icon Controls** (Figure 2) allows you to select attributes for the time series icon layers that are displayed. The time series icon controls are located on the **Visualization** tab near the bottom of its pane (Visualization Module). There are three attributes available: **Icons Type**, **Values**, and **Layers** (Figure 2).



46 Figure 2 Series Icon Controls

Icons Type	forces all icons within the map window to display as the selected **Icon Type** in the list. Selecting the **Default** option in the list will display the icons as they were originally defined during setup (Time Series Icons).
Values	allows you to display the time series icon's data as labels, which display next to or on the location of their associated time series icons. For example, you can choose to display the data's total, minimum, and maximums values as labels. Multiple values can be selected at once.
Layers	allows you to select the time series icon sub-layers that are visible in the map window. Multiple layers can be selected at once.

By selecting the attributes, you can change how time series icons are displayed within the Visualization module. For more detailed information on how to configure the properties that control the time series icons displayed, refer to Time Series Icons Layers.

11.2 Preparing to Review Current Conditions

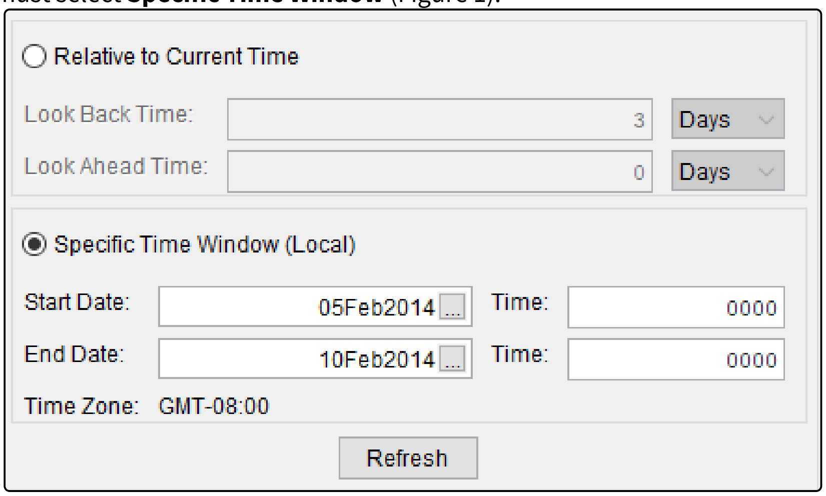
To begin reviewing current conditions in a watershed, you will need to set the time window for viewing and updating the color bars and plot icons in your map display.

11.2.1 Set Time Window

You can either set a specific (i.e., fixed) time window (for reviewing a historical event) or you can set a time window relative to the current time (the default setting, used for reviewing data).

To set or change the time window:

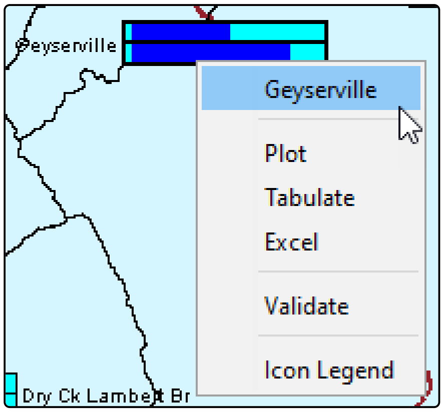
1. From the **Visualization** tab (Figure 1), for the default time window, HEC-RTS assumes that you want to review current data in the watershed, so **Relative to Current Time** is selected. If you want to view historical data, you must select **Specific Time Window** (Figure 1).



2. If you would like to set the **Time Window** (Figure 1) relative to the current date and time, select the **Relative to Current Time** and enter the **Look Back Time** and select either **Days** or **Hours** as the time unit. Normally you will set the time window to extend backward one to thirty days from the current time. Optionally, you can enter a **Look Ahead Time** and select either days or hours; this allows you to look ahead at data.
3. If you would like to set a historical **Time Window** (Figure 1), select **Specific Time Window** and enter the **Start Date** and **Time** of the time window and the **End Date** and **Time** of the time window for the historic data.
4. Click **Refresh** (Figure 1) to complete the time window change.

11.2.2 Time Series Icon Shortcut Menu

From the **Map Window** (Visualization Module), the time series icons display, by right-clicking on a time series icon, a shortcut menu (Figure 2) will display that allows you to view the data associated with a selected time series icon either in a plot, table, or using Microsoft Excel®. Also, from the shortcut menu you can access the Data Validation Editor (**Validate**) to edit the data associated with the selected time series icon (Data Validation Editor). Another option is that you can view the legend of the selected time series icon. The shortcut menu is the same for all of the available time series icon types (i.e., for the color bars, thumbnail plots, graphics, or dots).



47 Figure 2 Time Series Icon Shortcut Menu

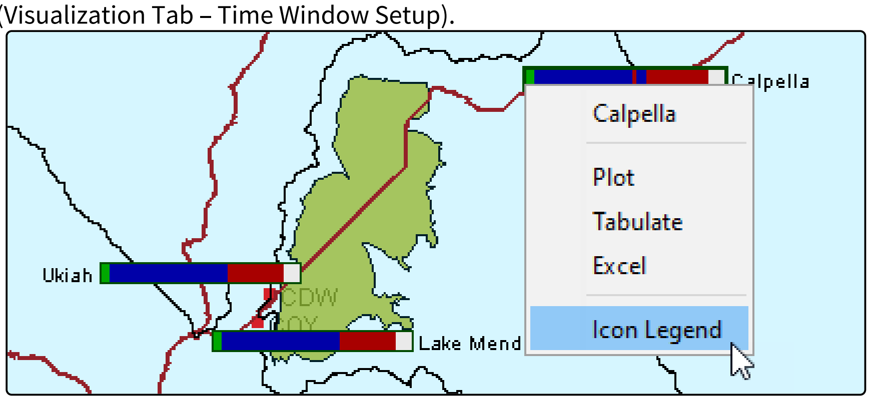
11.3 Visualizing Data

The Visualization module provides tools for evaluating the current state of the watershed. The module's main display has several visualization tools that help to identify whether critical conditions exist. Time series icons, plots and graphs, and animated background images can all be used for reviewing observed data for a watershed.

Time series icons in the Visualization module provide a flexible set of tools to facilitate evaluating the current state of the watershed. To view current conditions at a glance, you can use thumbnail plots and threshold color bars that change color to reflect the status of various parameters at a particular location.

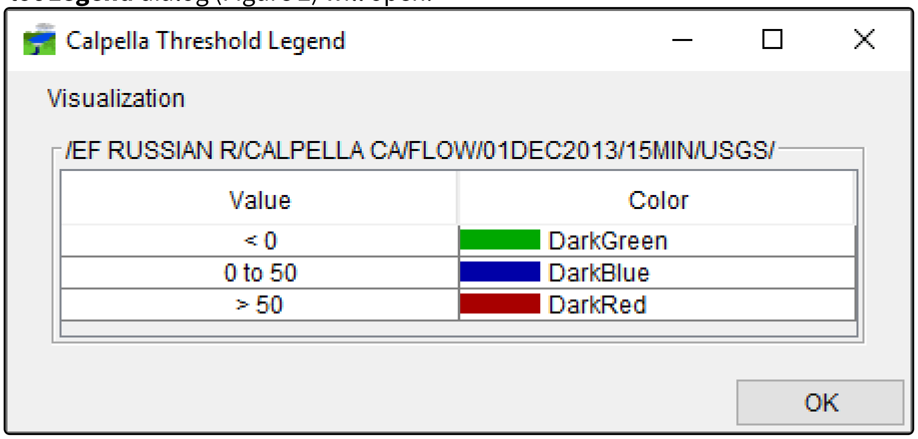
11.3.1 Threshold Color Bars

A **Threshold Color Bar** (Figure 1) is a thick, segmented line representing a linear histogram that compares current data against threshold values set for a location. Because threshold color bars are location dependent, you must set the values for each time series icon individually (Time Series Icon Editor). The name of the location appears adjacent to the threshold color bar, and the length of the **Threshold Color Bar** represents a timeline equal to the duration of the time window. Segments along the color bar reflect time intervals, with colors corresponding to the limits you have defined. The threshold color bars are updated dynamically at the time interval that you entered in for the time window (Visualization Tab – Time Window Setup).



As part of your daily routine, you should review threshold color bars to determine if the current data is good and which data needs to be validated. To view the legend for threshold color bars:

1. From the **Map Window**, right-click a thumbnail plot, from the shortcut menu, click **Icon Legend**. The **Thumbnail Plot Legend** dialog (Figure 2) will open.

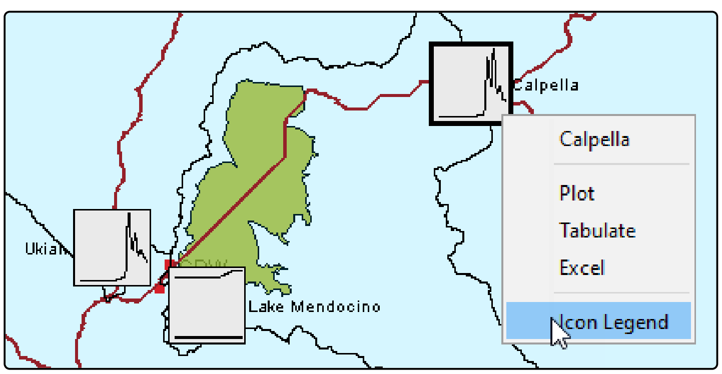


2. The legend shows the threshold values for the current data and the colors that represent each threshold value.

As conditions change at a location, the threshold color bar automatically changes colors. For example, at a precipitation location, DarkGreen might represent precipitation between .01 to .11 inches, corresponding to normal conditions. DarkBlue might represent values greater than .51 inches, but less than 1.0 inch, which indicate a flood warning level. DarkRed would represent values above 1.0 inch, signaling flows above flood stage.

11.3.2 Thumbnail Plots

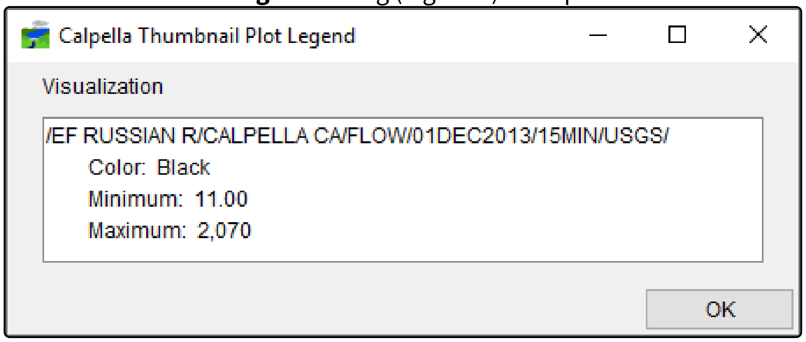
In some cases, you may wish to use thumbnail plots to represent gages and time series locations. **Thumbnail Plots** (Figure 3) are miniature versions of the full-size plots available in HEC-RTS, showing data for the set time window. The name of the location appears adjacent to the thumbnail plot. As the time window updates, HEC-RTS refreshes the data and updates the thumbnail plot.



48 Figure 3: Thumbnail Plots - Shortcut Menu

To view the legend for thumbnail plots:

1. From the **Map Window**, right-click on a thumbnail plot, from the shortcut menu (Figure 3), click **Icon Legend**. The **Thumbnail Plot Legend** dialog (Figure 4) will open.



2. The legend displays the pathname, the color of the curve, and the minimum and maximum values of the curve. The **Thumbnail Plot Legend** dialog (Figure 4) provides you with information, to edit the color of the curve shown in the thumbnail plot (Time Series Icons).

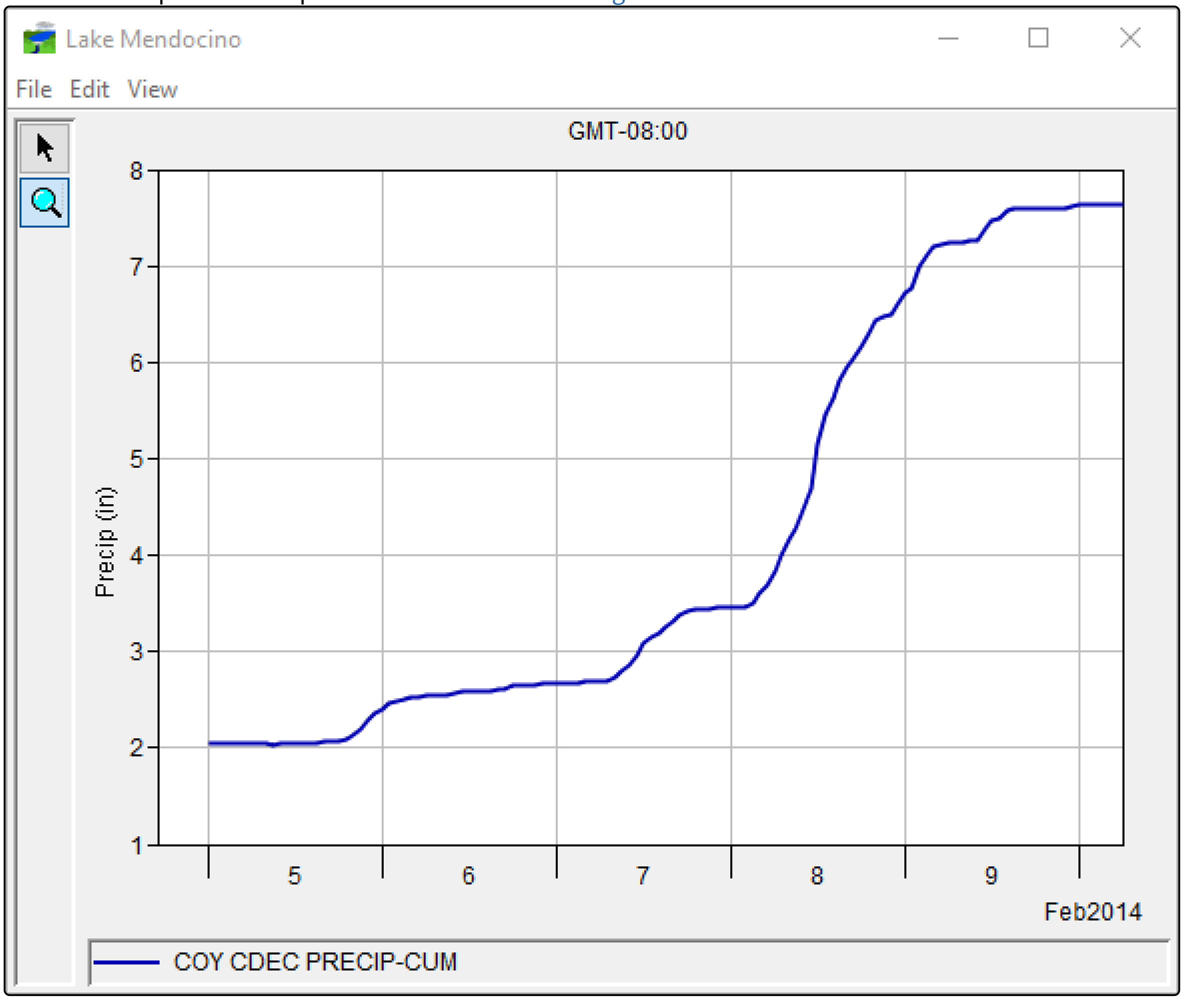
11.3.3 Using Plots

If the threshold color bar or thumbnail plot for a location indicates that data is questionable or bad, viewing the data in a plot can help you to determine the nature of the problem.

To access plots of incoming data:

1. From the **Map Window**, right-click a time series icon, from the shortcut menu (Figure 3), click **Plot**.
2. This opens a two-dimensional plot (Figure 5) that contains the data for that location for the current time window. Figure 5 shows a sample plot for a precipitation gage. Use the **Zoom Tool** to zoom in on a specific portion of the plot. See Map Window for a more detailed description of the **Zoom Tool**. In addition, a more

detailed description on the plots can be found in Viewing HEC-RTS Data and Results.



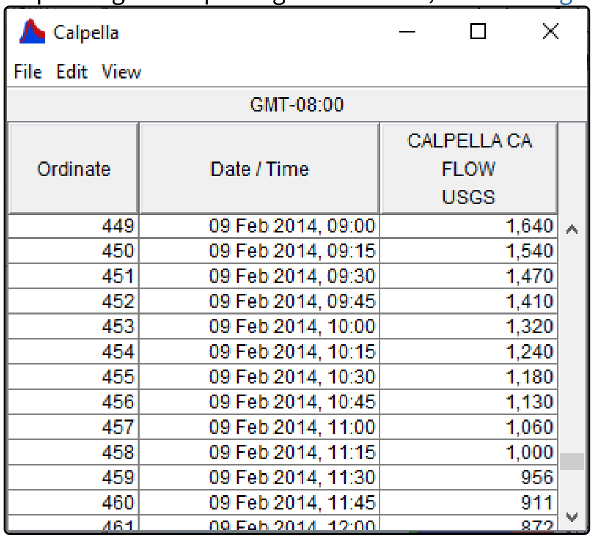
11.3.4 Using Tables

If the threshold color bar or thumbnail plot for a location indicates that data is atypical or unexpected, viewing the data in a table can help you to determine the nature of the problem.

To access tables of incoming data:

1. From the **Map Window**, right-click a time series icon, from the shortcut menu (Figure 3), click **Tabulate**.
2. This opens a table that contains data for that location for the current time window. Figure 6 shows a sample table of the data for a flow gage.
3. If you have already opened a plot, you can also tabulate the data displayed in the plot from the **Plot** dialog (Figure 5). From the **File** menu, click **Tabulate**. For more details on tabular data in HEC-RTS, see Visualizing Data.

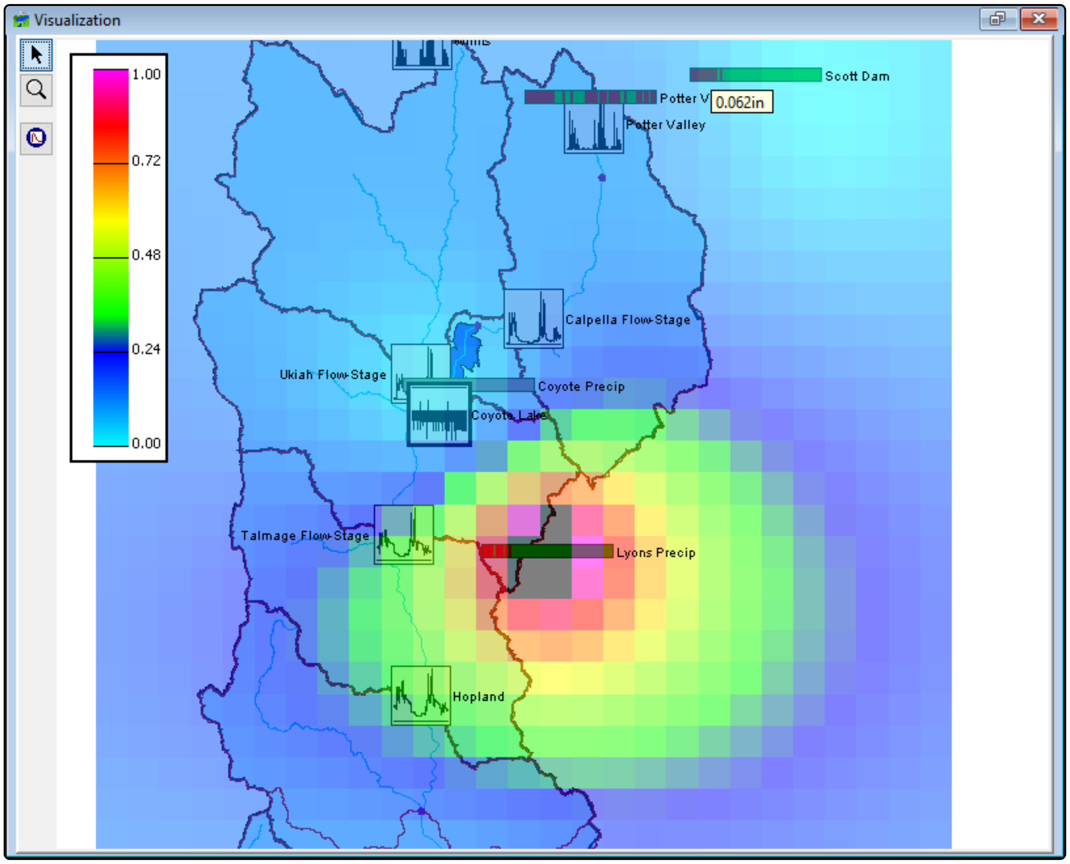
For printing and exporting tabular data, see Printing and Copying HEC-RTS Data.



Ordinate	Date / Time	CALPELLA CA FLOW USGS
449	09 Feb 2014, 09:00	1,640
450	09 Feb 2014, 09:15	1,540
451	09 Feb 2014, 09:30	1,470
452	09 Feb 2014, 09:45	1,410
453	09 Feb 2014, 10:00	1,320
454	09 Feb 2014, 10:15	1,240
455	09 Feb 2014, 10:30	1,180
456	09 Feb 2014, 10:45	1,130
457	09 Feb 2014, 11:00	1,060
458	09 Feb 2014, 11:15	1,000
459	09 Feb 2014, 11:30	956
460	09 Feb 2014, 11:45	911
461	09 Feb 2014, 12:00	872

11.4 Precipitation Grids

Precipitation data can be reviewed at time series icons to examine individual gage reports, identify problems, and confirm the validity of the data. The Visualization module also allows precipitation data to be viewed as grids (Figure 1), which gives a view of the precipitation over the whole watershed for an interval of time.

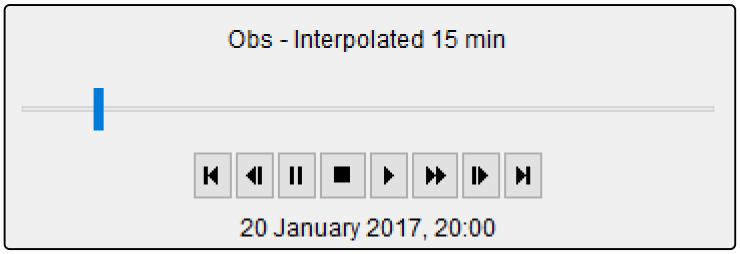


49 Figure 1 Precipitation Grids

Precipitation grids are created by interpolation between gages or by loading radar precipitation reports from the National Weather Service or other external sources. Individual precipitation grids are grouped into sequences called "grid sets" using the HEC-GridUtil software. To view grid sets in HEC-RTS you will use the grid animation controls in the Visualization module.

11.5 Grid Animation Controls

The grid animation controls (Figure 1) allow you to view precipitation grids for the time window you have chosen. The controls are located from the Visualization module. Above the slider bar (Figure 1), is displayed the name of the active grid set (*Obs – Interpolated 15 min*). Beneath the slide bar, the date and time of the data currently being displayed is shown. A **slider bar** and **video-player-style buttons** control the animated playback. The **slider** allows you navigate the active precipitation grid set for a specific time during the time window. With these playback controls, you can play an animation of the active precipitation grid set from the beginning to the end of the current time window.



The video-player-style buttons (Figure 1) function as follows:

 **Skip to Beginning:** Sets the slider at the beginning of the time window and displays the status of data at that time.

 **Step Back:** Steps the image back one-time interval.

 **Pause:** Pauses playback.

 **Stop:** Stops playback.

 **Play:** Plays the animation, one-time interval at a time, starting at the location of the slider.

 **Step Forward:** Steps the image forward one-time interval.

 **Fast Forward:** Plays through the animation in fast forward mode.

 **Skip to End:** Moves the slider and the image to the status at the end of the time window.

Tip

For playback to occur, you must define the coordinate system for both the watershed layers and the grid set and there must be data present for the time window specified. If no grid is available for the time step, the label centered over the grid animation controls will end with **No Data**.

Use HEC-DSSVue to check for available grids in *$HEC-RTS_HOME/dated/db/grid/precip.YYYY.MM.dss*. You may need to run **gageInterp** or your grid-loading program to populate the DSS files for this time interval.

11.6 Precipitation Grid Sets

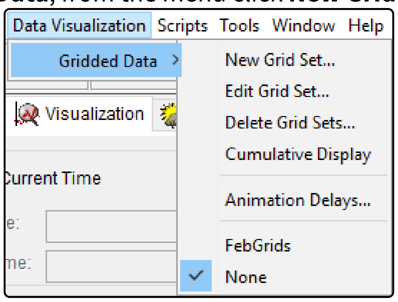
A **grid set** can be thought of as a sequence or time series of grids. Since a single DSS record of grid data contains data for only a one-time interval, you must create grid sets before you can view the precipitation grids changing through time over the watershed. Grid sets are created by **HEC-GridUtil**.

11.6.1 Add a Grid Set

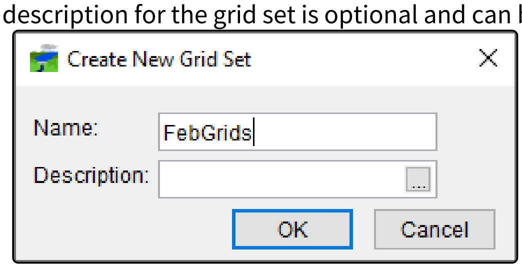
Once a grid set has been created for a watershed, you can add it to the Visualization Module for viewing in the **Map Window**.

To add a grid set to the watershed:

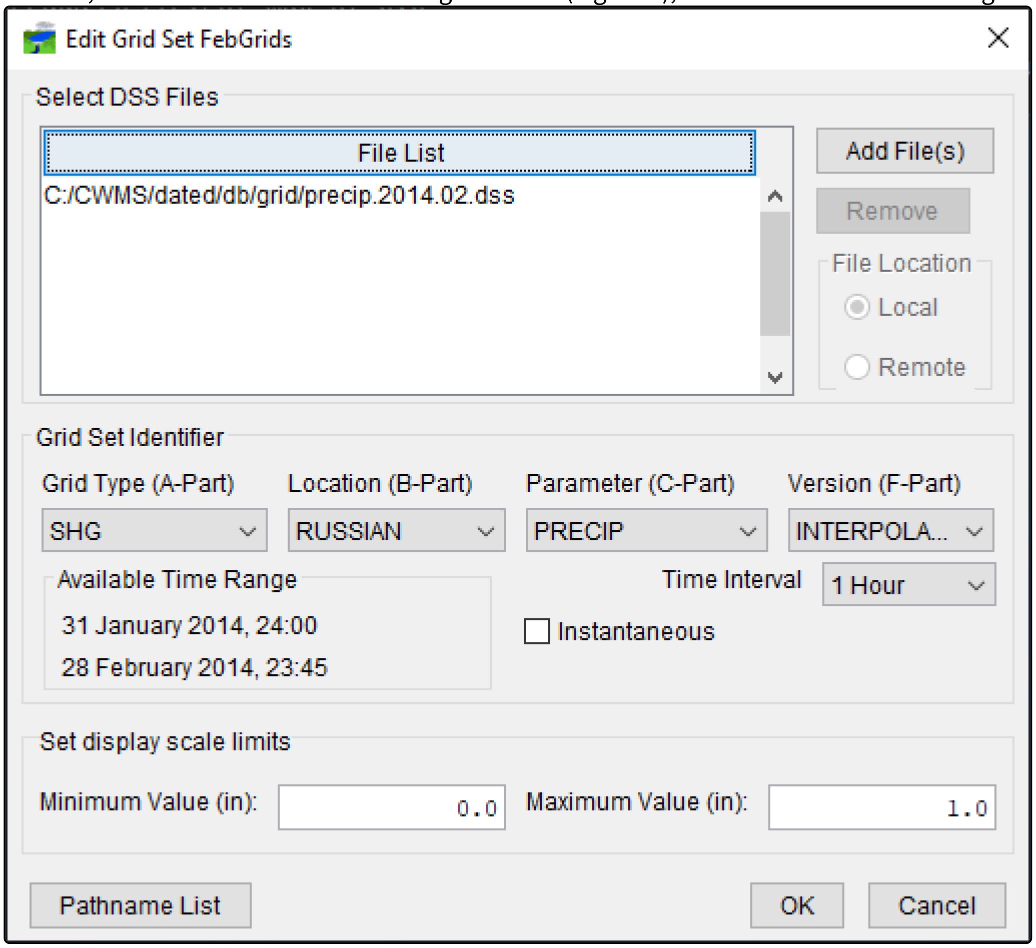
1. From the HEC-RTS main window (Visualization Module), from the **Data Visualization** menu, point to **Gridded Data**, from the menu click **New Grid Set** (Figure 1).



2. The **Create New Grid Set** dialog will open (Figure 2). In the **Name box** enter a name for the grid set. A description for the grid set is optional and can be entered in the **Description** box (Figure 2).



3. Click **OK**, the **Create New Grid Set** dialog will close (Figure 2), and the **Edit Grid Set** dialog will open (Figure 3).



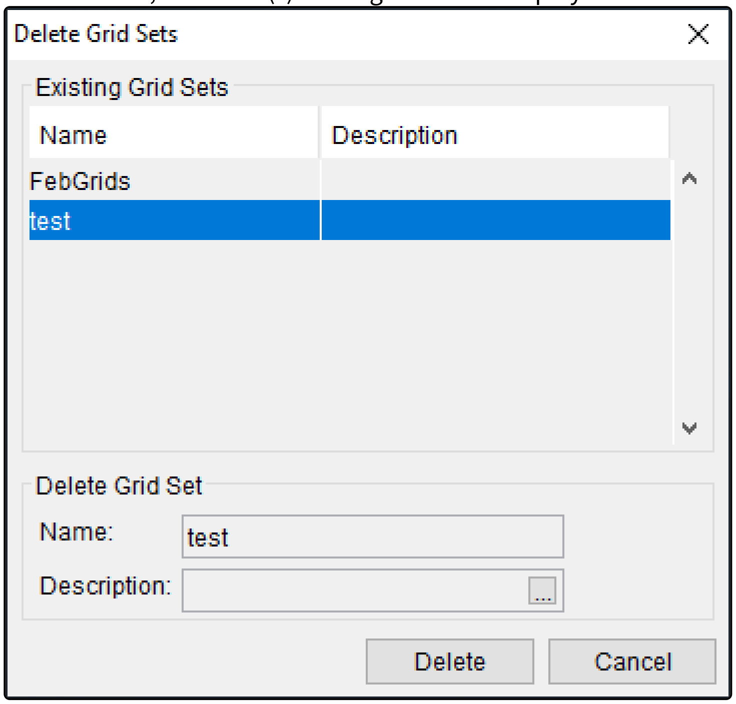
4. Click **Add File(s)**, an **Open** browser will open. Navigate to the location where available precipitation grids are. Click on a grid set (*precip.yyyy.mm.dss*), click **Open**, the **Open** browser will close, and the pathname and the filename will display in **File List** (Figure 3).
5. From the **Grid Set Identifier** box (Figure 3), you will need to select the individual identifiers from the available lists (i.e., **Grid Type (A-Part)**; **Location (B-Part)**, etc.). You also need to select the time interval from the **Time Interval** list (Figure 3). Click **OK**, the **Edit Grid Set** dialog will close. From the **GriddedData** menu (Figure 1). the name of the grid set that was just created will display.

11.6.2 Delete a Grid Set

To delete a grid set from the watershed:

1. From the HEC-RTS main window (Visualization Module), from the **Data Visualization** menu, point to **Gridded Data**, from the menu click **Delete Grid Sets** (Figure 1).

2. The **Delete Grid Sets** dialog will open (Figure 4). From the **Existing Grid Sets** list, select the grid sets that you wish to delete; the name(s) of the grid set will display in the **Name** box (Figure 4).



3. Click **Delete**, a **Warning** window will display, asking you if you really want to delete the selected grid set, click **Yes**. The **Warning** window and the **Delete Grid Sets** dialog (Figure 4) will both close. The grid set will no longer display in the **Gridded Data** menu (Figure 1) and will be removed from the watershed.

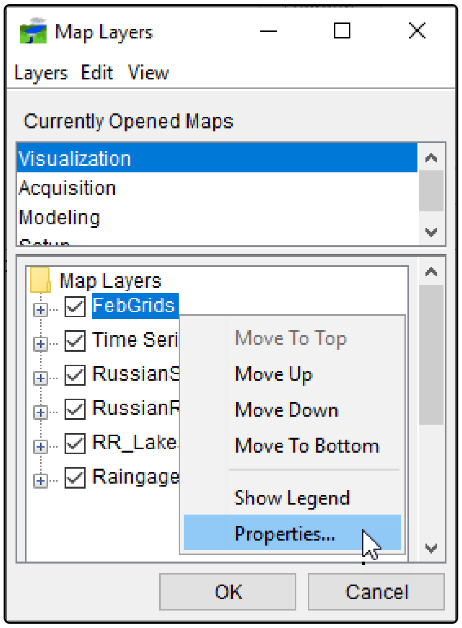
11.6.3 Selecting a Grid Set

To select a grid set:

1. From the HEC-RTS main window (Visualization Module), from the **Data Visualization** menu, point to **Gridded Data**. From the menu (Figure 1), at the bottom of the menu, the names of the grid sets that have been added to the watershed are displayed, along with **None**.
2. To select a grid set for viewing in the Visualization module, click on the grid set name (e.g., *FebGrids*). The name of the selected grid set will display in the **Active Grid Set** section of the HEC-RTS main window (Visualization Module). Also, the **Animation Controls** will become active, and the **Grid Set Legend** will display in the **Map Window** (Visualization Module).
3. To unselect a grid set, from the **Gridded Data** menu (Figure 1), click **None**.

11.6.4 Properties of a Grid Set

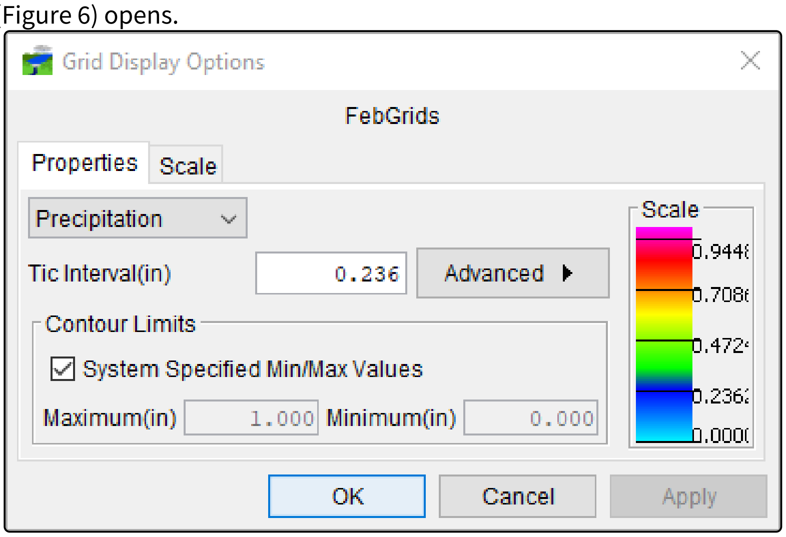
You can control the visibility and color scheme of the grid set through the **Map Layers** dialog (Understanding Layers). In the Visualization module, the **Map Layers** dialog (Figure 5) displays the selected grid set layer, from which you can edit various properties of the grid set. These properties include color scheme, tic intervals, and the minimum/maximum values for the contour.



50 Figure 5 Map Layers Dialog - Grid Set Shortcut Menu

Changing a grid set's properties:

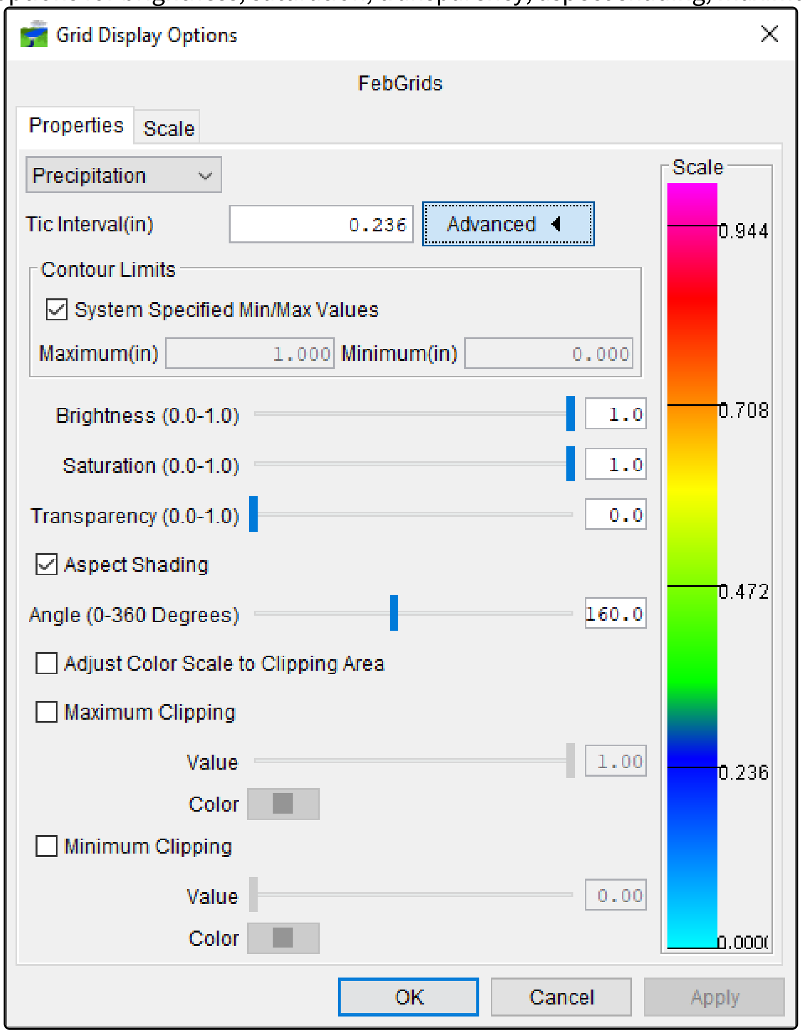
1. From the **Maps** menu, click **Map Layers** and the **Map Layers** dialog (Figure 5) opens. From the tree, right-click on the grid set layer name (*FebGrids*), from the shortcut menu, click **Properties**. The **Grid Display Options** dialog (Figure 6) opens.



2. The name of the active grid set is displayed at the top of the **Grid Display Options** dialog.
3. From the **Properties** tab (Figure 6), the list at the top left contains six pre-set color contour schemes, with the default being **Precipitation**.
4. From the **Tic Interval** box (Figure 6), you can set the tic interval for the selected grid set.
5. In the **Contour Limits** box (Figure 6), you can set the minimum and maximum contour values for the selected grid set. By default, HEC-RTS automatically determines the minimum and maximum contour values. If you wish

to enter these values manually, clear **System Specified Min/Max Values**. Then enter values in the **Maximum** and **Minimum** boxes.

6. For other settings to adjust, click **Advanced**. The **Grid Display Options** dialog expands (Figure 7) to include options for brightness, saturation, transparency, aspect shading, maximum clipping, and minimum clipping.



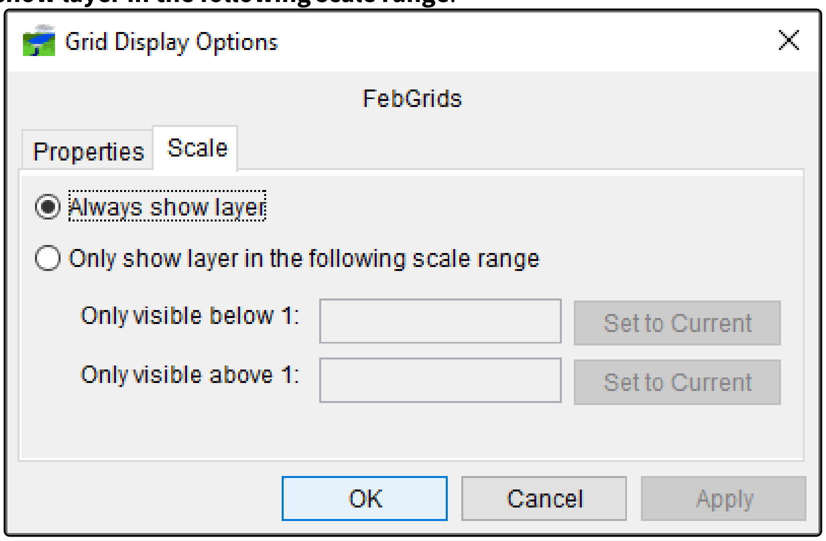
7. You can control the **Brightness** (the amount of white, measured from 0.0 to 1.0), **Saturation** (the amount of black, measured from 0.0 to 1.0), and **Transparency** (the level of opacity or alpha, measured from 0.0 to 1.0) of your gridded data layer either by moving the sliders or by typing values into the white text fields. The color scale to the right will update according to your selections.
8. If you are using elevation maps, you may wish to choose **Aspect Shading** as your color contour scheme. Use the **Aspect Shading** option to make the elevation map appear in relief by placing an imaginary light source above the map and shading the elevation contours. When the **Aspect Shading** option is selected, you can use the **Angle** slider to adjust the angle of the light source.
9. By default, the **Maximum Clipping** and **Minimum Clipping** options are deselected. If you activate these options, the **Value** sliders and **Color** buttons become available. 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 boxes. When you click on

the **Color** buttons, the **Color Chooser** appears, for more information on the **Color Chooser** see Using the Color Chooser.

11.6.5 Visualization Scale of a Grid Set

To set the visualization scale for a grid set:

1. From the **Maps** menu, click **Map Layers**, the **Map Layers** dialog (Figure 5) opens. From the tree, right-click on the grid set layer name (*FebGrids*), from the shortcut menu, click **Properties**. The **Grid Display Options** dialog (Figure 6) opens.
2. Click the **Scale** tab (Figure 8). Both zoom-in and zoom-out scale factor can be set from the **Grid Display Options** dialog. By default, the **Gridded Data Layer** is set for **Always show layer**. To see visualization scales, click **Only show layer in the following scale range**.

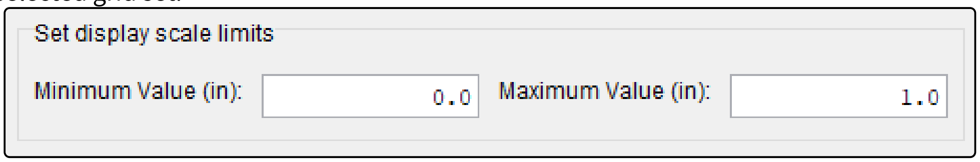


3. To set the scale so that the gridded data layer becomes visible as you zoom-in, you can set the scale to the current scale by clicking **Set to Current**. If you wish to enter a value, from the **Only visible below 1** field, enter the scale factor.
4. To set the scale so that the gridded data layer becomes visible as you zoom-out, you can set the scale to the current scale by clicking **Set to Current**. If you wish to enter a value, from the **Only visible above 1** field enter the scale factor.
5. Click **OK**, the **Grid Display Options** dialog (Figure 8) closes, and the visualization scale is set for the gridded data layer.

Another way to set the visualization scale for a grid set:

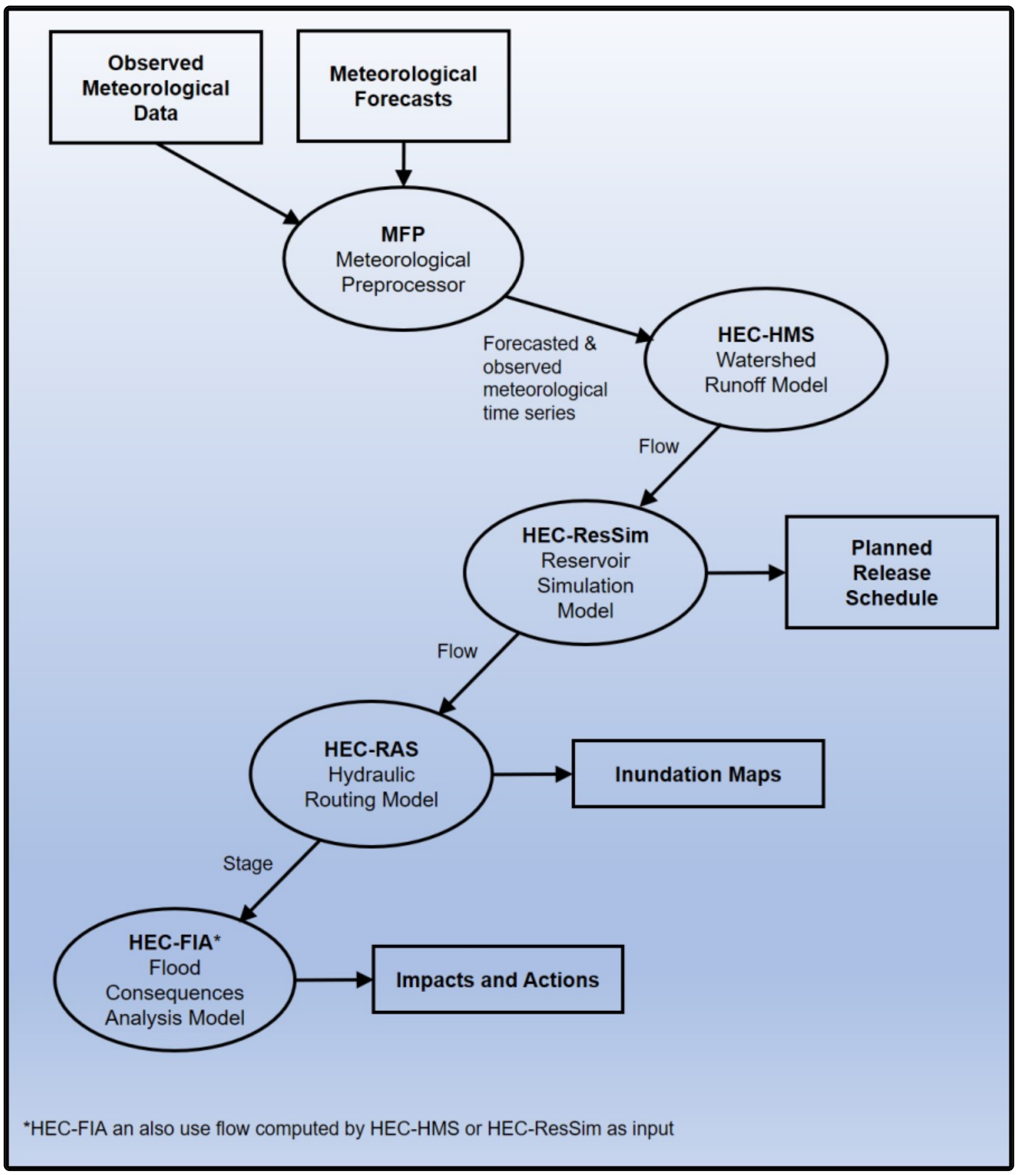
1. From the HEC-RTS main window (Visualization Module), from the **Data Visualization** menu, point to **Gridded Data**, from the menu click **Edit Grid Set** (Figure 1).
2. An **Open** dialog will open (like Figure 4). From the **Existing** list, select a grid set, click **Open**. The **Open** dialog will close, the **Edit Grid Set** dialog will open (Figure 3).

3. From the **Set display scale limits** box (Figure 9), you can set the minimum and maximum scale values for the selected grid set.



12 Modeling Module

The **Modeling** module for HEC-RTS is the grouping of commands for performing forecasts for a watershed. These commands include: creating and managing forecast alternatives, editing selected model parameters, executing alternatives, and evaluating results. Figure 1 displays the flow of data and information through the software applications involved in a forecast. HEC-RTS has five standard software applications available: the meteorological preprocessors MFP, the watershed runoff model HEC-HMS, the reservoir simulation model HEC-ResSim, the hydraulic routing model HEC-RAS, and the flood consequences analysis model HEC-FIA. These software applications and their inputs and outputs are described in HEC-RTS Overview.



51 Figure 1 HEC-RTS Workflow

The meteorological preprocessor MFP combines observed precipitation data and future precipitation information into a single precipitation dataset that can be used by the watershed runoff model (HEC-HMS). Using the observed and future precipitation from a meteorological preprocessor (MFP), HEC-HMS computes runoff. HEC-ResSim uses uncontrolled flows (typically computed by HEC-HMS) as input and simulates reservoir releases. In turn, controlled flows (computed by HEC-ResSim) are used as boundary conditions to the hydraulic routing model HEC-RAS. (HEC-RAS can also use flows computed by HEC-HMS or supplemental models as input). HEC-RAS computes stage, which is input to HEC-FIA. HEC-FIA analyzes flood consequences. (HEC-FIA can also use flow computed by HEC-HMS, or HEC-ResSim as input.) Results include impact and action reports from HEC-FIA, inundations maps that can be viewed with RAS Mapper (tool that is part of the HEC-RAS software), and simulated release schedules from HEC-ResSim.

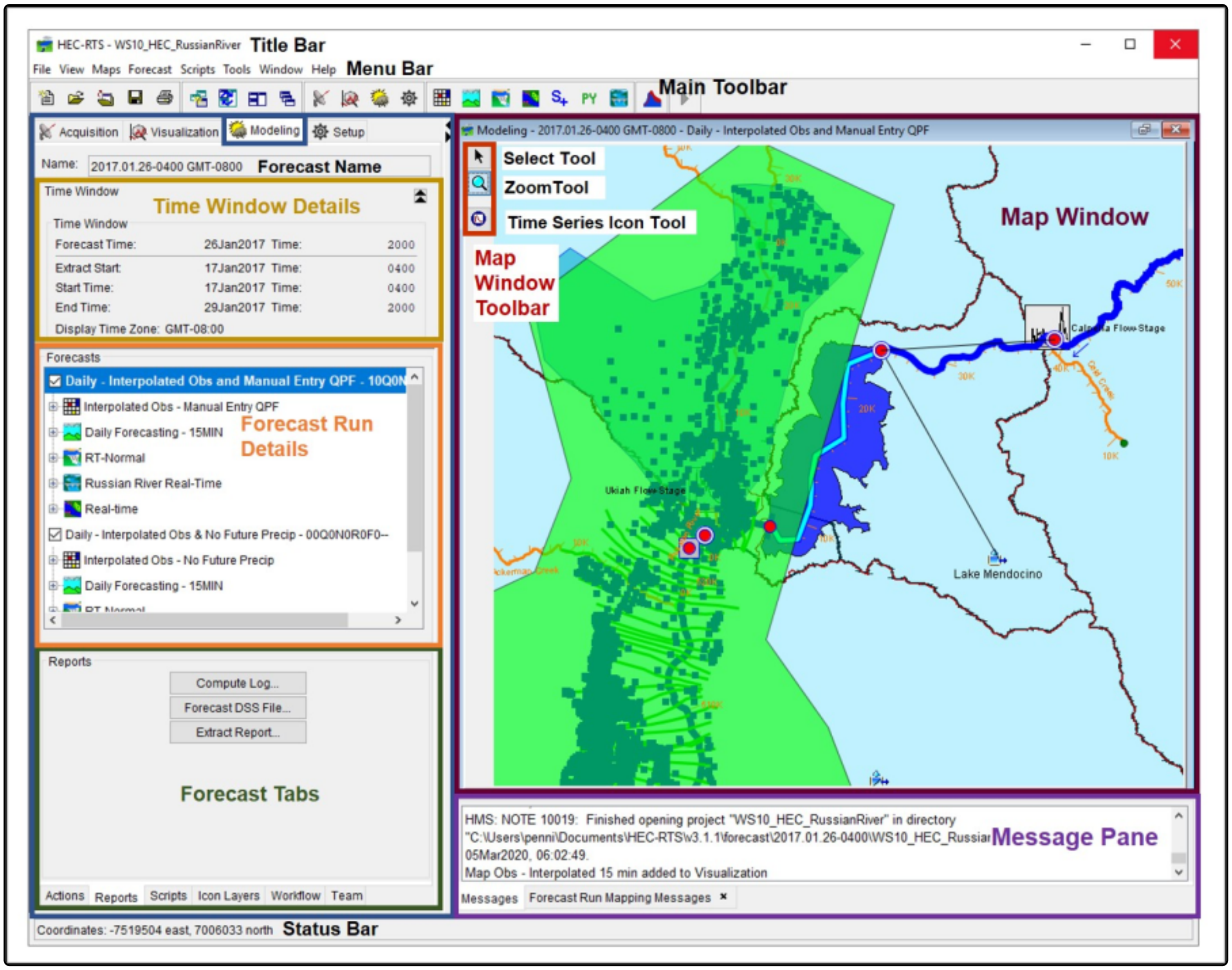
12.1 HEC-RTS Forecast Workflow

The following is a typical workflow for forecasting:

1. Check that critical data are available - **Acquisition** module.
2. Create a new HEC-RTS forecast. An HEC-RTS forecast is a simulation of watershed processes and consequences of flooding based on input data and information and hydrologic, reservoir operation, hydraulic, and impact analysis applications configured in HEC-RTS. During a forecast, a **forecast run** is run at a specific forecast time. (Recall that, a forecast run is a single forecast scenario defined by a specific set of data, information, and HEC-RTS alternatives.) HEC-RTS copies all the necessary files to a dated forecast directory (HEC-RTS Application Settings). You should check the data for errors and ensure that the data were extracted properly.
3. Compute the forecast and check results. Forecast results include flow and stage in the channel from watershed runoff, reservoir release schedules, floodplain inundation maps, floodplain consequence reports, and reports listing actions for emergency responders to take. These results inform water management decision making. You can view results tables and plots in the Modeling module. Compute messages provide information about each step of the computation process. Model-specific reports provide details about the individual models. RAS Mapper allows you to view HEC-RAS inundation maps. If your models alternatives require adjustment, you can access model editors your model forecast runs and override controls. These features are discussed in Model Alternatives and Forecast Runs.
4. Manage forecast information. When model results have been reviewed and validated, HEC-RTS provides tools to assist in managing information. Forecast results and data can be posted to DSS file(s), save modeling data, and protect your forecast from further editing.

12.2 Overview

The Modeling module is the grouping of commands for performing forecasts for a watershed. From the HEC-RTS main window, click the **Modeling** tab (Figure 1). A detailed discussion of common screen components is presented in HEC-RTS Interface. The **Modeling** tab (Figure 1) displays the time window; current date and time of the active forecast; details, about the current forecast run; provide easy access to commands and processes for the user; display the selected forecast run in a map window; and, provide information about the execution of a forecast.



52 Figure 1 HEC-RTS Main Window - Modeling Module

12.3 Modeling Module Menu Bar

The **File**, **View**, **Maps**, **Scripts**, **Tools**, **Windows**, and **Help** menus are discussed in HEC-RTS Interface. The commands available from the **Forecast** menu will allow you to create/edit/delete forecasts; run forecasts daily so you can review conditions during an event; and, post results to DSS files. Refer to HEC-RTS Forecasts for further details.

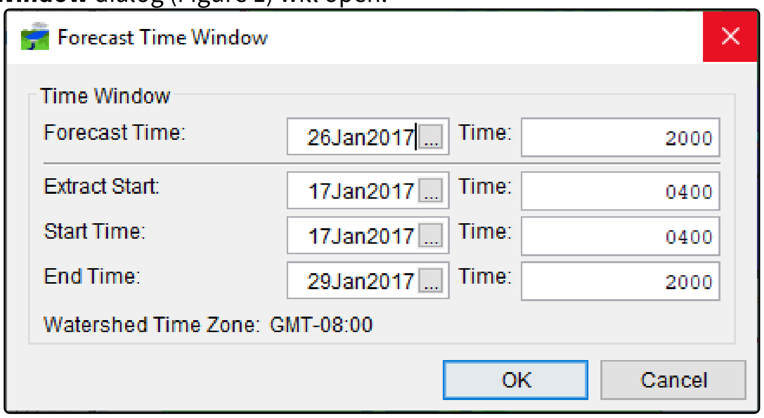
Forecast From this menu you can create, edit, open, save, delete, and archive forecasts; review information about the active forecast; post active forecast results to DSS files; and compute all model alternatives (forecast run) or compute individual model alternatives. Available commands are: **New**, **Open**, **Recent**, **Continue**, **Compute Manager**, **Run Selector**, **Edit Time Window**, **Information**, **Extract**, **Post**, **Logs**, **Close**, **Save**, **Archive**, **Delete**, **Restore**, **Default Time Window**, **Model Linking**, **Forecast Scheduler**, and **Scheduled Forecast Manager**.

The **Logs** sub-menu allows you to view the log files from extract, post, and compute processes. Available commands are: **Extract Report**, **Post Report**, and **Compute**.

12.4 Time Window Details

The **Time Window Detail** area (HEC-RTS Main Window - Modeling Module) of the Modeling module, provides you with details about the time window for the selected forecast. Information includes forecast time; extract time; start and end time of the time window; and, the display time zone. You can also edit the time window for the selected forecast:

1. From the **Modeling** tab (Figure 1), from the **Forecast** menu, click **Edit Time Window**. The **Forecast Time Window** dialog (Figure 1) will open.



2. You can now edit the time window for the selected forecast. You can edit the **Forecast Time** date and time; **Extract Start** date and time; **Start Time** date and time; and, **End Time** date and time.
3. Once you have entered your new time window, click **OK**. The **Forecast Time Window** dialog (Figure 1) will close. The **Time Window Detail** area (HEC-RTS Main Window - Modeling Module) will update with the new forecast time window information.

12.5 Forecast Run Details

The **Forecast Run Details** area (HEC-RTS Main Window - Modeling Module) of the Modeling module provides you with details about the forecast runs that are available for the selected forecast. Information includes: the name of the forecast runs; and, for each forecast run, the model alternatives associated with a forecast run.

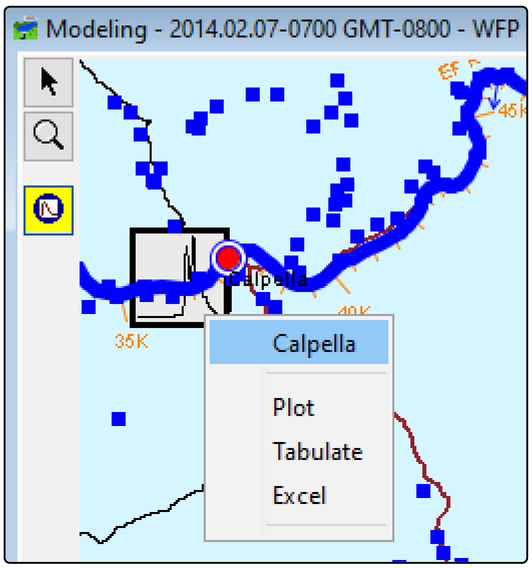
For the forecast runs you can perform various processes (e.g., **Compute**, **Post**, etc.) from either a shortcut menu or from the Forecast Tabs. Further details on these processes is available in HEC-RTS Forecasts. Also, for the individual model alternatives associated with a forecast run, you can perform various process (e.g., **Compute**, view **Log File**, etc.). or by the elements (e.g., **Edit**, **Plot**, etc.) associated with each model alternative. Further details on these processes is available in HEC-RTS Forecasts.

12.6 Modeling Module Map Window

In the Modeling module, the **Map Window** (HEC-RTS Main Window - Modeling Module) contains the model schematic elements for the active forecast in a geo-referenced context. In addition, time series icons are available that provide access to additional model data or other time series data. Time series icons are setup in the **Setup** module.

12.6.1 Time Series Icon Shortcut Menu

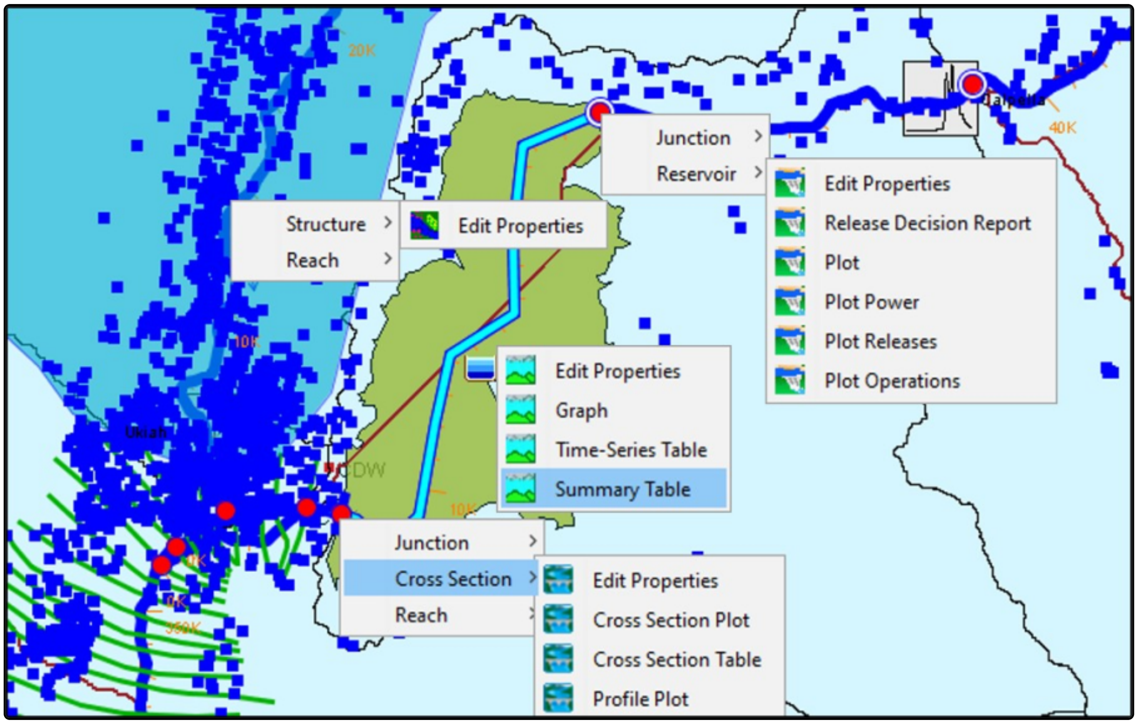
In the Modeling module**,** from the **Map Window** (HEC-RTS Main Window - Modeling Module), click (**Time Series Icon Tool**). Right-click on a time series icon in the map window, a shortcut menu will display (Figure 1). From this shortcut menu (Figure 1), you can view the data and results associated with a selected time series icon in a plot, tabulate table, or open the Microsoft Excel® application, so you can view data or results in a spreadsheet format.



53 Figure 1 Modeling Module - Map Window - Time Series Icon Menu

12.6.2 Edit and Review Results by Models

In the Modeling module**,** from the **Map Window** (Figure 11.2), click (**Select Tool**). From the schematic, right-click on individual model elements, from the shortcut menu (Figure 2), you can edit and review results for the individual models of the selected forecast.



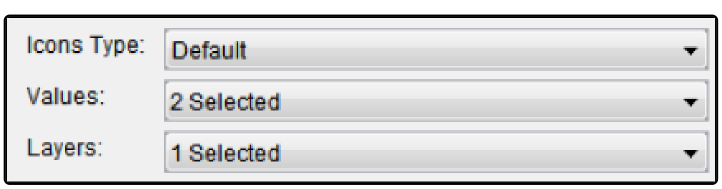
54 Figure 2 Modeling Module - Map Window - Individual Model Elements

12.7 Modeling Module Forecast Tabs

The **Forecast Tabs** (HEC-RTS Main Window - Modeling Module) in the Modeling module provide easy access to commands and processes for the user. There are six tabs – **Actions**, **Reports**, **Scripts**, **Icon Layers**, **Workflow**, and **Team**. The **Icon Layers** tab, in the Modeling module, allows you to select attributes for the **Time Series Icons Layer**, Forecast Tabs provides further details. The other five tabs contain information and/or commands based on whether you have selected a model alternative or an element of that model alternative, Forecast Tabs will provide an example using the **Actions** tab.

12.7.1 Icon Layers Tab

From the **Forecast Tabs** (HEC-RTS Main Window - Modeling Module), click the **Icon Layers** tab, the **Time Series Icon Controls** display (Figure 1). The **Time Series Icon Controls** allows you to select attributes for the time series icon layers that are displayed. There are three attributes available: **Icons Type**, **Values**, and **Layers** (Figure 1).



55 Figure 1 Forecast Tabs - Icon Layers Tab - Time Series Icon Controls

Icons Type forces all icons within the **Map Window** (HEC-RTS Main Window - Modeling Module) to display as the selected **Icon Type** in the list. Selecting the **Default** option in the list will display the icons as they were set originally during setup (Time Series Icons).

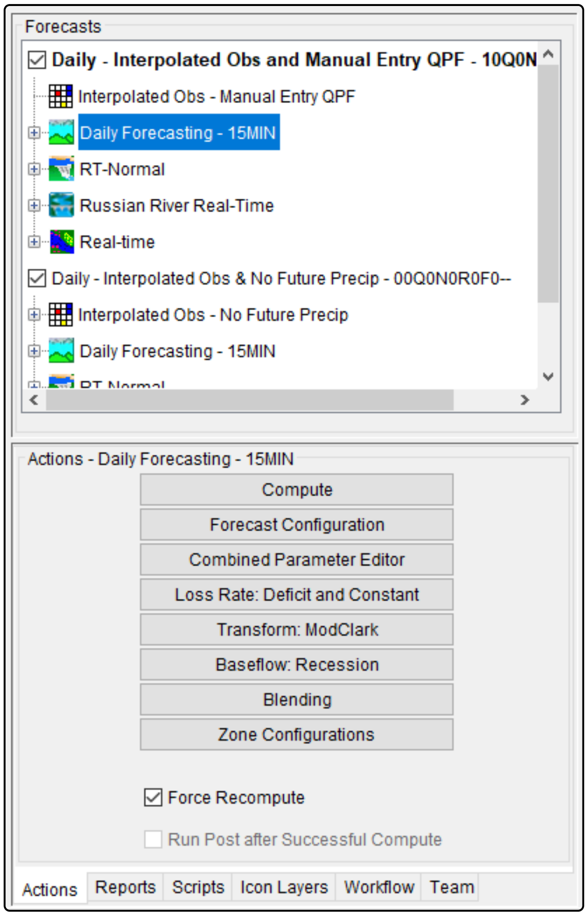
Values allows you to display the time series icon's data as labels, which display next to or on the location of their associated time series icons. For example, you can choose to display the data's total, minimum, and maximums values as labels. Multiple values can be selected at once.

Layers allows you to select the sub-layers that are visible in the map window. Multiple layers can be selected at once.

By selecting the attributes, you can change how time series icons are displayed within the Modeling module. For more detailed information on how to configure the properties that control time series icons, see Time Series Icons.

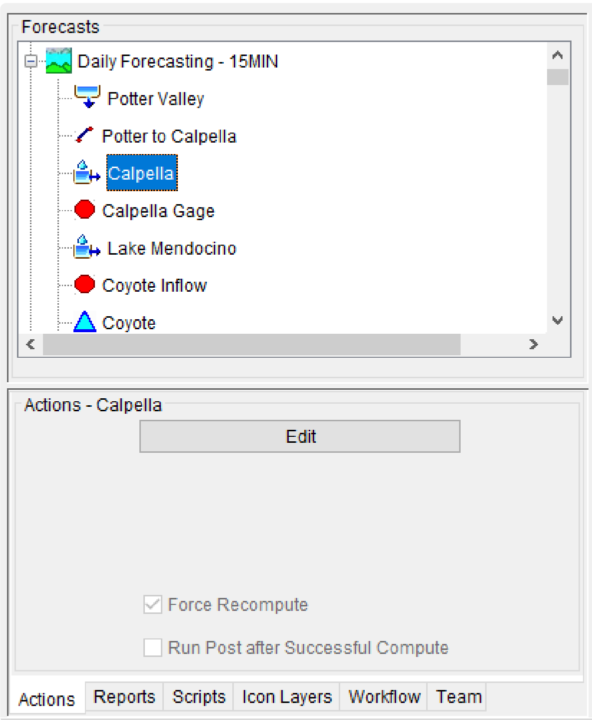
12.7.2 Actions Tab

Like several other of the **Forecast Tabs**, the **Actions** tab (Figure 2) is influenced by the user's choices in the **Forecast Run Details** area (HEC-RTS Main Window - Modeling Module). From the **Forecast Run Details** area, if you select a model alternative (*Daily Forecasting – 15MIN*), the **Actions** tab (Figure 2) displays the ability to compute the individual model alternative, and gives you access to several editors that are specific to the selected model alternative (e.g., HEC-HMS). This mean the user can make changes to the selected model alternative without having to go to the **Setup** module to make edits.



56 Figure 2 Actions Tab - Selected Forecast Run - Daily Forecasting - 15MIN

If the user expands (clicks +) the model alternative (Figure 3), a list of elements specific to that model alternative will display (e.g., source, reservoir, subbasin, etc.). For example, in Figure 3, you have selected *Calpella* (a subbasin), from the **Actions** tab, you can access an editor specific to that element type and you can make changes specific to that element. This means the user can make changes to the specific model alternative element without having to search the **Map Window** for the element to make edits.



57 Figure 3: Actions Tab - Selected Forecast Run Elements - Daily Forecast - 15MIN

12.8 Modeling Module Message Pane

The **Message Pane** (HEC-RTS Main Window - Modeling Module) contains messages that the HEC-RTS software is providing for informational purposes. The information in the **Message Pane** is a record of HEC-RTS activity, recording the opening of a watershed, opening map windows, accessing DSS files, and many other HEC-RTS activities.

12.9 Preparing to Forecast

Before you forecast, you will need to set up the Program Order, create Model Alternatives, assign Model Alternative Keys, and create at least one Forecast Run. You will also set the default time window for your forecast and ensure that data is available for your forecast. Remember, it is essential that you setup the program order, before forecasting.

MFP Model Alternative contains information on how to create MFP model alternatives. For the other software applications, refer to Model Alternatives and Forecast Runs and the individual user documentation (HEC-HMS, HEC-ResSim, HEC-RAS, and HEC-FIA).

13 HEC-RTS Forecasts

In HEC-RTS, a **forecast** is where a forecast run is computed for a specific time window, with boundary condition data available. Forecasts are created and edited in the Modeling module. Before you begin your forecast, you should check your critical data and make sure it is available for the time window of your forecast (Preparing to Review Current Conditions). In addition, you need to setup your Program Order, create Model Alternatives, and, assign Model Alternative Keys.

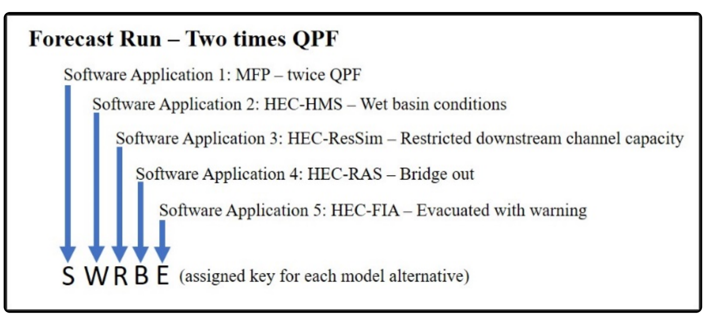
13.1 Forecast Runs

A **forecast run** is a set of model alternatives that run sequentially during a forecast. As with model alternatives, forecast runs exist independent of time. Therefore, you can use a forecast run for any forecast time window. You create a forecast run by selecting a model alternative for each software application that could be part of an HEC-RTS forecast. There are five standard software applications in HEC-RTS, a typical forecast run will include a combination of five model alternatives. When you execute a forecast, HEC-RTS executes the software applications in sequential order.

The table below provides suggested forecast runs. The codes are the names HEC-RTS automatically assigns to each forecast run based on the one-character model alternative keys you have assigned to the model alternatives. This coding is *program order dependent*.

Example Forecast Runs	
Name	**Code**
No future precip	NNNNN
QPF	QNNNN
Two times QPF	SWRBE

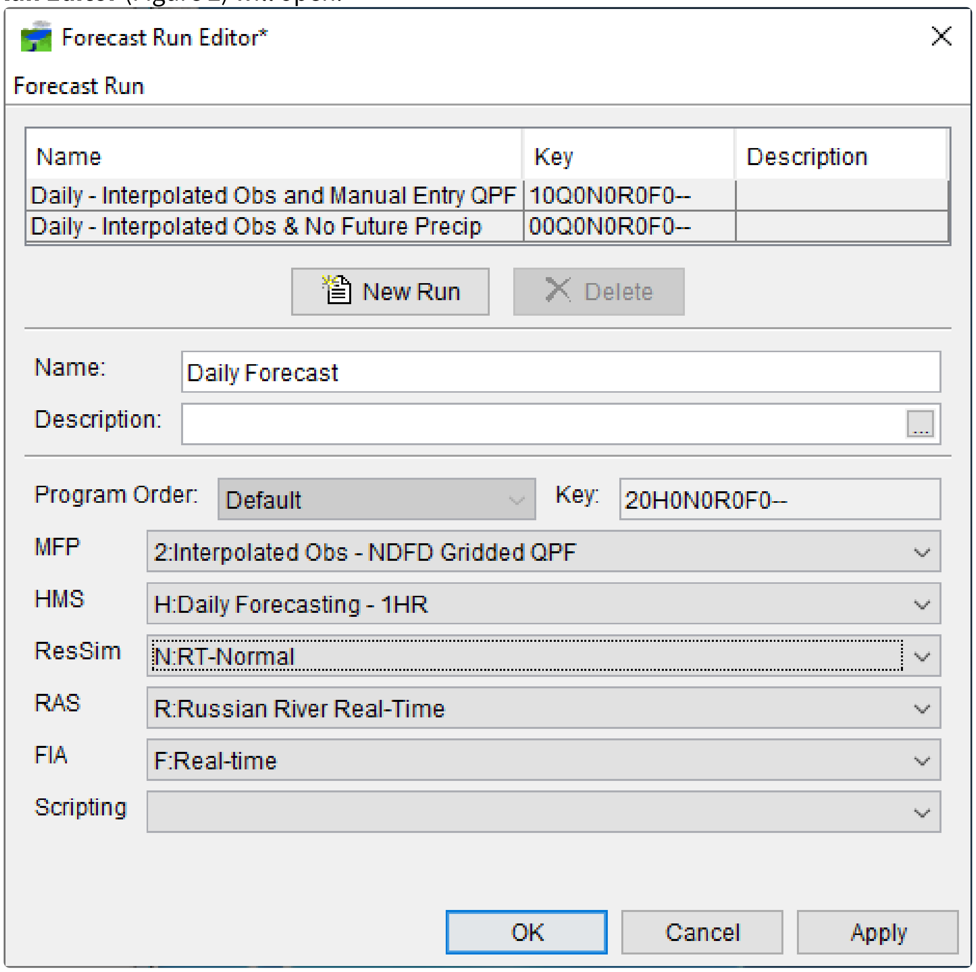
In the following example (Figure 1), the forecast run - *Two times QPF* has a key code of *SWRBE*. Each character is the key code that was assigned to each model alternative for each of the five standard software applications in HEC-RTS.



Before creating a forecast run, you must have model alternatives. Model Alternatives and Forecast Runs provides details on how to create model alternatives. Creating forecast runs occurs in the **Setup** module. Further details on managing forecast runs (e.g., opening, deleting) is provided in Model Alternatives and Forecast Runs.

Creating a forecast run:

1. From the HEC-RTS main window, click the **Setup** tab, from the **Models** menu, click **Forecast Runs**. The **Forecast Run Editor** (Figure 2) will open.

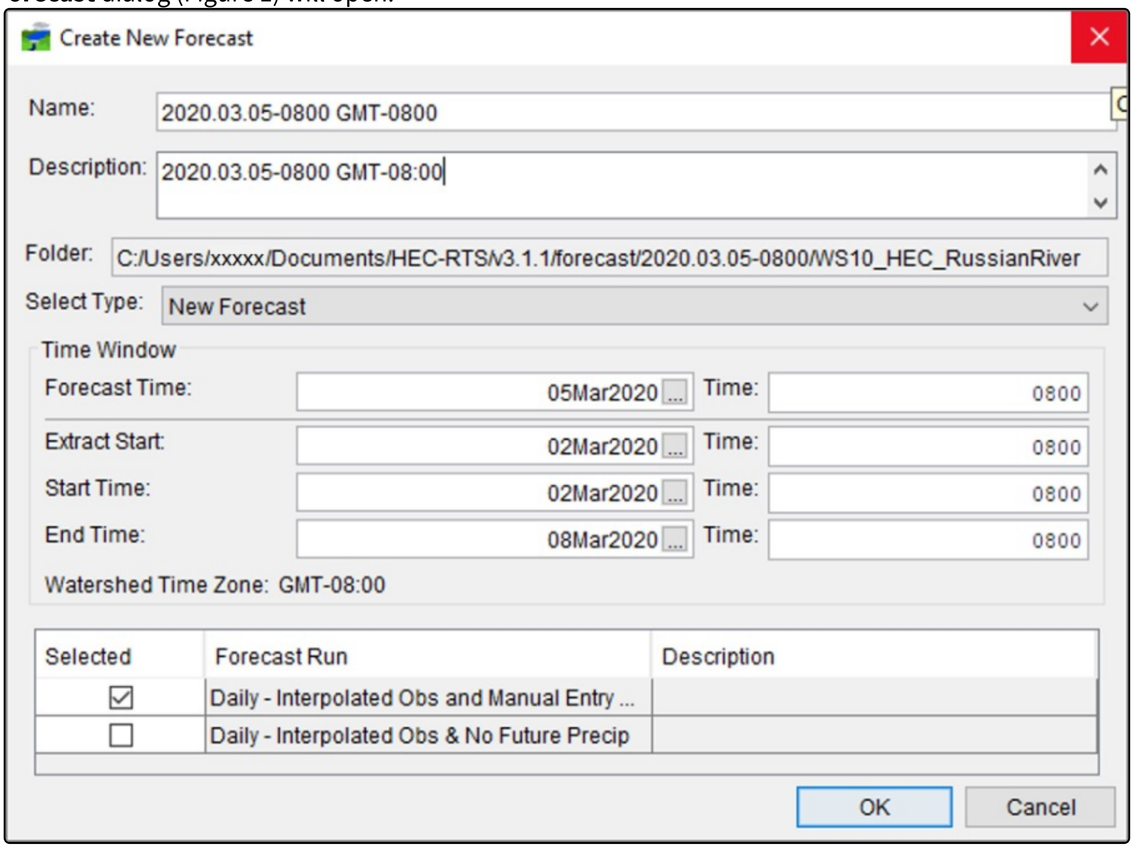


2. Click **New Run**, enter a new forecast run name in the **Name** box (required), a description in the **Description** box (optional).
3. From the **Program Order** list (Figure 2), select a program order. Program Order provides detailed information on program order.
4. From the software applications list (Figure 2), select the individual model alternates required for the forecast run. As you are selecting those model alternatives, in the **Key** box (Figure 2), HEC-RTS is automatically creating the key for the forecast run, based on the individual model alternative keys (Setup Module).
5. From the **File** menu, click **Save**. The new forecast alternative appears in the table on the **Forecast Run Editor** (Figure 2). Click **OK**, the **Forecast Run Editor** closes.
6. It is recommended at this point to save your watershed. From the HEC-RTS main window, from the **File** menu, click **Save Watershed**.

13.2 Forecasts

A **forecast** is a time window, that is associated with one or more forecast runs. Following are the steps for creating a forecast:

1. From the HEC-RTS main window, click the **Modeling** tab, from the **Forecast** menu, click **New**. The **Create New Forecast** dialog (Figure 1) will open.

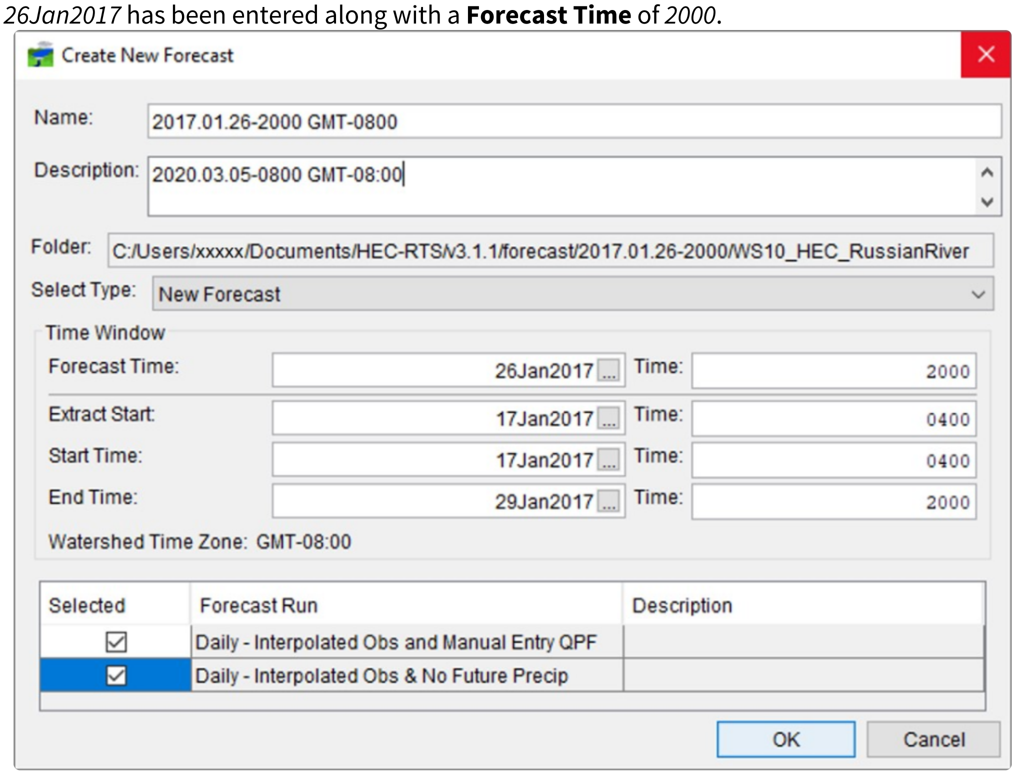


2. By default, from the **Select Type** list (Figure 1) you are creating a new forecast (**New Forecast**). From the list you have the following options:

New Forecast	This is the default option in which you specify the **Forecast Time**, **Extract Start**, **Start Time**, **End Time** and **Forecast Run**(s) used in the forecast.
From Existing Forecast	This option copies the **Forecast Time**, **Extract Start**, **Start Time**, **End Time** and **Forecast Run**(s) from a forecast already created. Here you select a forecast already created and saved in the watershed.

From Archived Forecast	This option copies the **Forecast Time**, **Extract Start**, **Start Time**, **End Time** and **Forecast Run**(s) from a forecast that has been archived. Here you select the zip file containing the archived forecast. For more information on archiving your forecast see Managing Forecast Data.
From Imported Forecast	This option copies the **Forecast Time**, **Extract Start**, **Start Time**, **End Time** and **Forecast Run**(s) from a forecast file stored to disk. Here you specify the **.forecast file* to use.

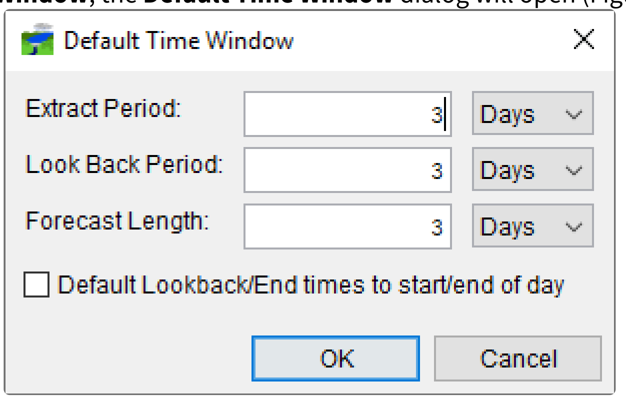
3. By default, HEC-RTS creates a forecast name, based on the time window (Figure 1). In addition, by default, HEC-RTS creates a time window based on the current date and time; for examples in this user's manual, the time window is an historical event.
4. You will enter a forecast date and time that you determined in the Acquisition module, this would be the latest time a complete set of reliable data is available (Figure 2). For example, in Figure 2, a **Forecast Date** of *26Jan2017* has been entered along with a **Forecast Time** of *2000*.



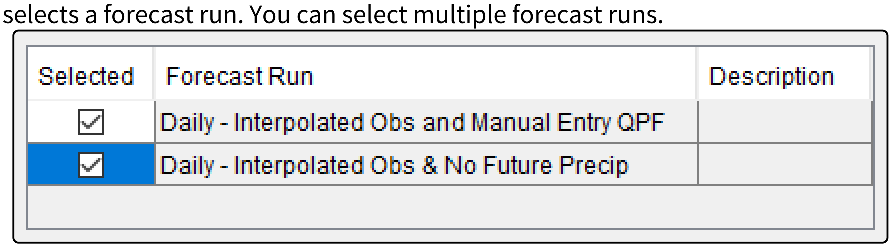
5. Once the **Forecast Date/Time** has been entered, HEC-RTS creates a forecast name. As illustrated in Figure 2, in the **Name** box, by default, the forecast name of *2017.01.26-2000* has been generated. Also, by default, HEC-RTS creates a folder based on the forecast date and time. As illustrated in Figure 2, the **Folder** box contains the generated pathname and file for the forecast name – *C:/Users/xxxxx/Documents/HEC-RTS/forecast/2017.01.26-2000/WS10_HEC_RussianRiver*. *WS10_HEC_RussianRiver* is the name of the watershed. The **Folder** box (Figure 2) is non-editable.
6. The selected **Start Time** and **Extract Start** dates (Figure 2) should be far enough in the past to meet the following:

a. Allow enough time steps to pass between the **Start Time** date and the **Forecast Time** date to allow routing reaches in the reservoir model to show correct flows at the time of forecast.
b. Generally, the **Start Time** and the **Extract Start** dates should be before the start of the event for calibration of loss rates for HEC-HMS. If this is not practical, then a time after the initial losses have been satisfied should be chosen. In most cases these dates and time are equal. However, from time to time, the forecaster may wish to extract data prior to the simulation window or may wish to update only the last few hours of data (for example when re-computing a forecast later in the day after more observed data are available).

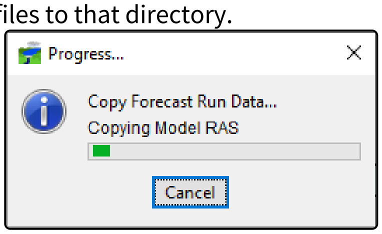
7. The **Extract Start**, **Start Time** and **End Time** fields will default to dates based on information given in the **Default Time Window** dialog (Figure 3). To view this information from the **Forecast** menu, click **Default Time Window**, the **Default Time Window** dialog will open (Figure 3).



8. From the **Create New Forecast** dialog (Figure 2) you can override the default time window. You will also need to enter times for the corresponding dates.
9. From the **Forecast Runs** table (Figure 4) on the **Create New Forecast** dialog (Figure 2) select the already defined forecast runs you wish to simulate for the forecast. Clicking the check box in the **Selected** column of the table selects a forecast run. You can select multiple forecast runs.

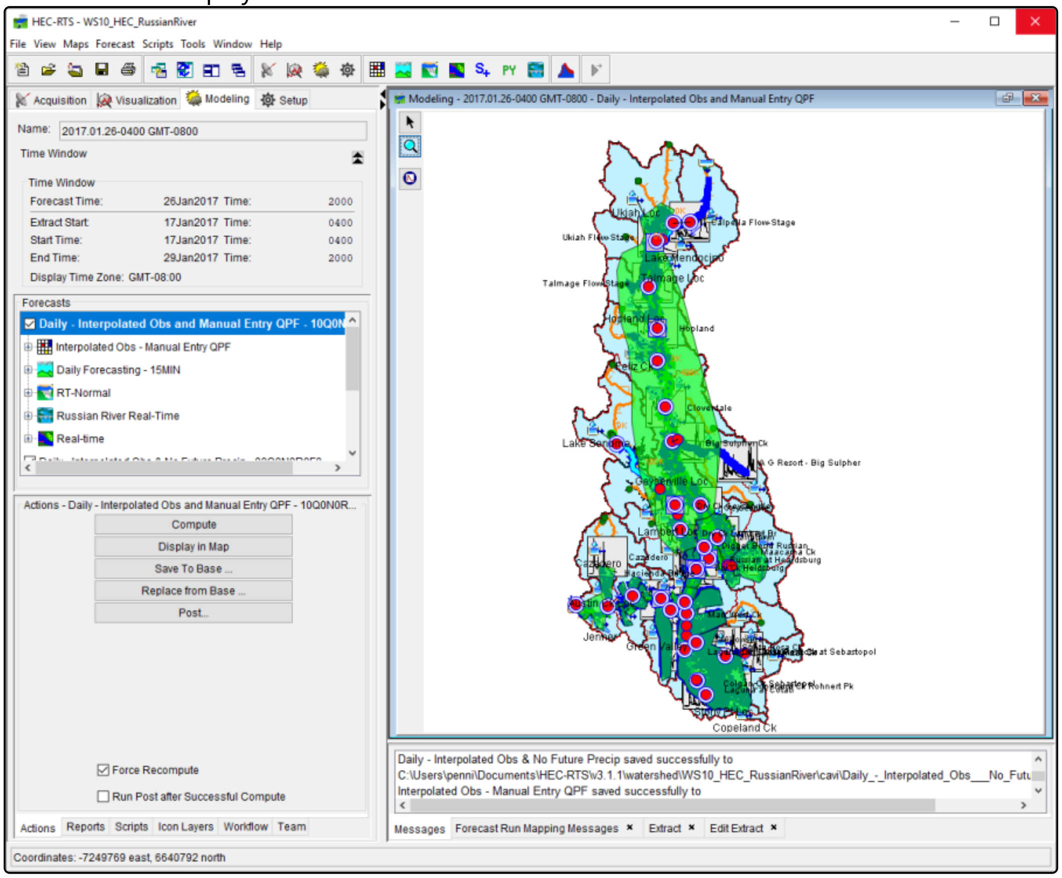


10. Click **OK**, and the **Create New Forecast** dialog closes (Figure 2). A **Progress** window will open (Figure 5), which provides information on the creation of the forecast, its associated directory, and the copying of the appropriate files to that directory.



11. In addition, HEC-RTS retrieves (extracts) data from defined DSS file(s) based on the entered times. These DSS file(s) are identified in an **Extract Group**, the **Extract Group** should be defined before creating a forecast. Extract groups are defined in the Setup module. The data is copied to a DSS file (*forecast.dss*) for use by the model alternatives. Check the **Extract Report** (Review Forecast Data) for any issues that might have occurred.

12. The **Modeling** tab (Figure 6) will display the created forecast (with a forecast run selected) in the **Map Window** (Figure 6). In addition, the name of the forecast; time window information about the forecast; and forecast run(s) information are displayed.



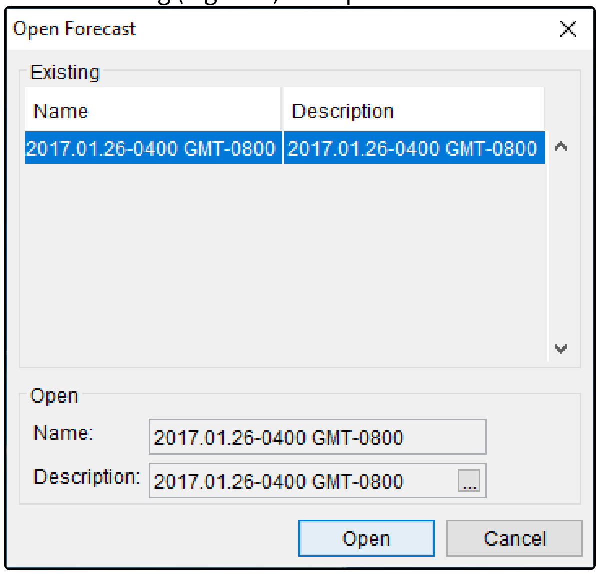
13.3 Managing Forecasts

HEC-RTS provides many options for managing forecasts - opening an existing forecast; deleting a forecast; closing an active forecast; editing the active forecast time window; rerun extracts; add or remove forecast runs from an active forecast; and, saving an active forecast. The following sections provide details on managing forecasts.

13.3.1 Open a Forecast

To open a forecast:

1. From the HEC-RTS main window, click the **Modeling** tab, from the **Forecast** menu, click **Open**. The **Open Forecast** dialog (Figure 1) will open.

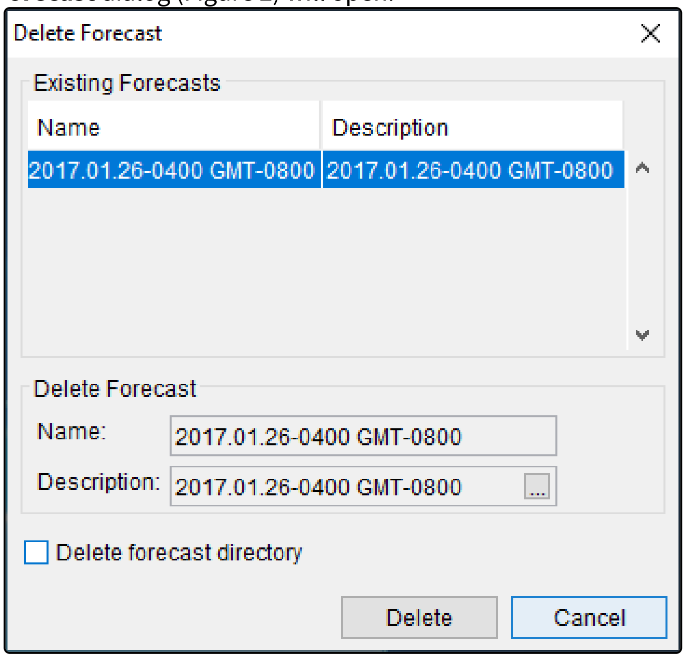


2. From the **Existing** list of available forecasts for the watershed, select the forecast you wish to open.
3. Click **Open**, the **Open Forecast** dialog closes (Figure 1). HEC-RTS will load all the information associated with the selected forecast. In the **Modeling** tab (Modeling Module - Forecast Information Displayed), the top checkbox of the tree will display the name of the selected forecast and the forecast runs associated with the selected forecast. The **Map Window** will display the model representations of the active forecast run.

13.3.2 Delete a Forecast

Delete a forecast:

1. From the HEC-RTS main window, click the **Modeling** tab, from the **Forecast** menu, click **Delete**. The **Delete Forecast** dialog (Figure 2) will open.



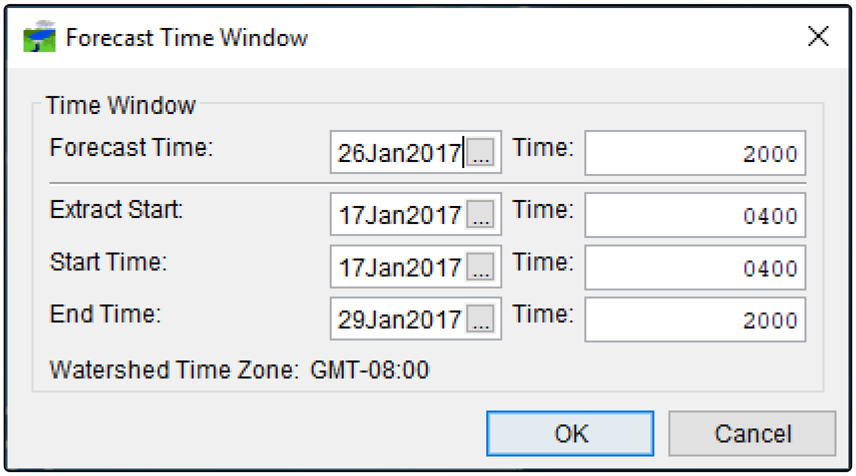
2. In the **Existing Forecasts** box (Figure 2) is a list of the forecasts that have been created for the watershed, select a forecast from the list.
3. Click **Delete**, a **Warning** window will open asking you if you really want to delete the forecast. Click **Yes**, the **Warning** window closes, and the forecast is removed from the watershed. This process does not physically delete any files; it only deletes the reference of the forecast from the watershed.
4. If you wish to delete the associated directory of the selected forecast, from the **Delete Forecast** dialog (Figure 2), click **Delete forecast directory**. This option will delete the whole forecast directory.

13.3.3 Close a Forecast

To close a forecast, from the HEC-RTS main window, click the **Modeling** tab, from the **Forecast** menu, click **Close**. If you have made changes to the forecast, you will be given options on how to manage those change, further details are provided in Managing Forecast Data of this chapter. HEC-RTS will unload all information associated with the active forecast and the Modeling module (Modeling Module - Forecast Information Displayed) will be blank.

13.3.4 Edit the Time Window

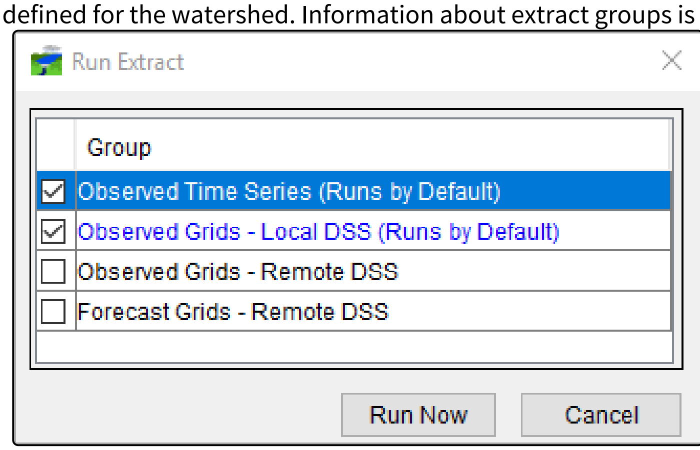
If you want to change any of the dates or times associated with the active forecast, from the HEC-RTS main window, click the **Modeling** tab, from the **Forecast** menu, click **Edit Time Window**. The **Forecast Time Window** dialog (Figure 3) will open. In the **Forecast Time Window** dialog, enter the new information in the corresponding date and time fields. Click **OK**, the **Forecast Time Window** dialog (Figure 3) will close.



58 Figure 3 Forecast Time Window Dialog

13.3.5 Run Extracts

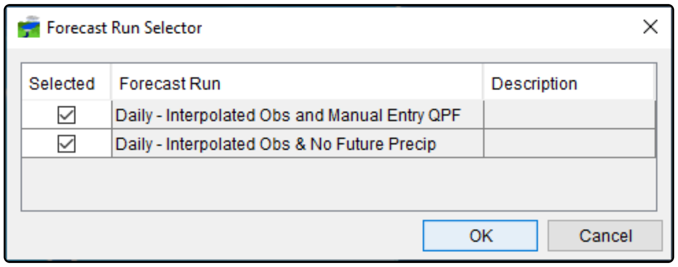
If you want to retrieve data from the defined DSS file(s) contained in the extract groups again, from the **Forecast** menu, click **Extract**, the **Run Extract** dialog will open (Figure 4). In the table are displayed the extract groups that have been defined for the watershed. Information about extract groups is detailed in Data Extract and Post Editors.



Select an extract group(s) the you wish to rerun, and un-select the extract group(s) that you do not want to run. Click **Run Now** (Figure 4), the **Run Extract** dialog will close. Progress of the extract is displayed in the **Message Pane** (Modeling Module - Forecast Information Displayed), from the **Extract** tab.

13.3.6 Add Forecast Runs

If you wish to add already defined forecast runs to the selected forecast, from the **Forecast** menu, click **Run Selector**, the **Forecast Run Selector** dialog (Figure 5) opens. In the **Forecast Run Selector** dialog, select the defined forecast runs you wish to add to the forecast by clicking check boxes in the **Selected** column. Click **OK**, the **Forecast Run Selector** dialog (Figure 5) closes, a **Progress** window will open displaying the progress of adding the forecast run to the active forecast.



13.3.7 Remove Forecast Runs

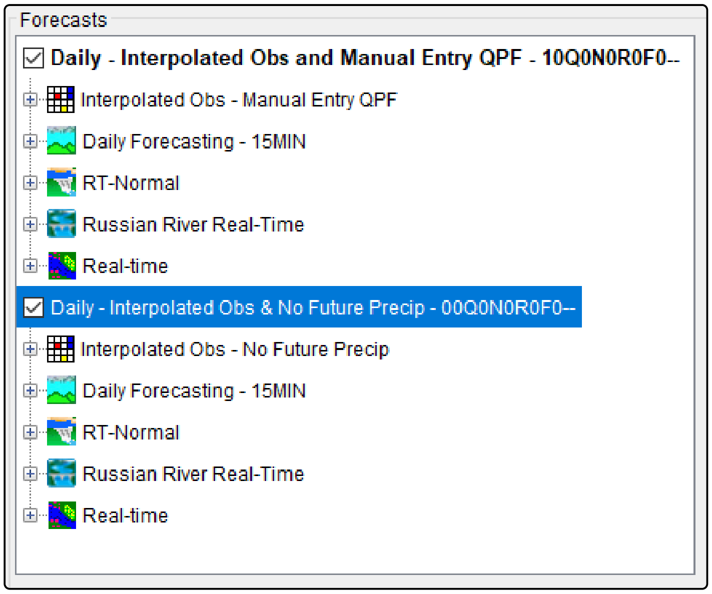
To remove a forecast run from a forecast, from the **Forecast Run Selector** dialog (Figure 5) clear the check box in the **Selected** column (Figure 5) next to the name of the forecast run you wish to remove from the active forecast. Click **OK**, the **Forecast Run Selector** dialog closes, the forecast run no longer displays on the HEC-RTS main window (Modeling Module - Forecast Information Displayed).

13.3.8 Save the Active Forecast

To save the active forecast, from the **Forecast** menu, click **Save**, and HEC-RTS will save information for the active forecast.

13.4 HEC-RTS Forecast Run Details

From the **Forecast Run Details** section of the Modeling module (Figure 1), you can view what forecast runs are part of the forecast that has been selected. In addition, you can view the individual model alternatives for a forecast run. By default, the first forecast run displays in the **Map Window** and the name of the forecast run is bolded (Figure 1) in the **Forecast Run Details** section. When a forecast run is bolded, this means that the forecast run is being displayed in the active **Map Window** of the Modeling module (Modeling Module - Forecast Information Displayed).

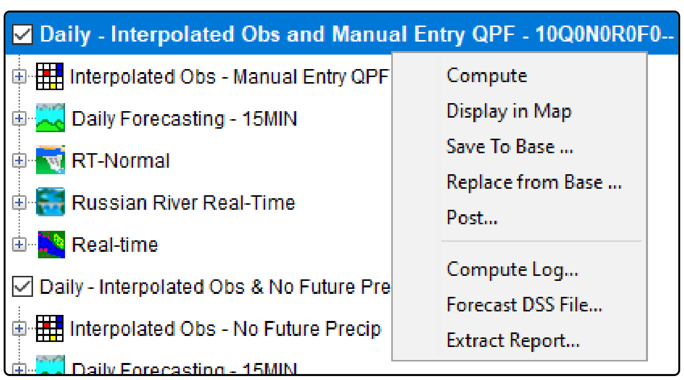


59 Figure 1 Modeling Module - Forecast Run Details Section

When you have clicked the checkbox next to a forecast run (Figure 1), you are indicating that you want to view output from the model schematic that is displayed in the **Map Window** for multiple forecast runs. In this version of HEC-RTS, the only software application that displays output results for multiple forecast runs is HEC-ResSim. In addition, the time series icons will display output for multiple forecast runs. MFP will display separate summary reports for each forecast run.

13.4.1 Forecast Runs – Shortcut Menu

From the **Forecast Runs Detail** section (Figure 2) of the Modeling module, when you right click on a forecast run name, a shortcut menu will display (Figure 2). From this shortcut menu, you can perform many processes. Further details about these processes are detailed in Model Alternatives and Forecast Runs.



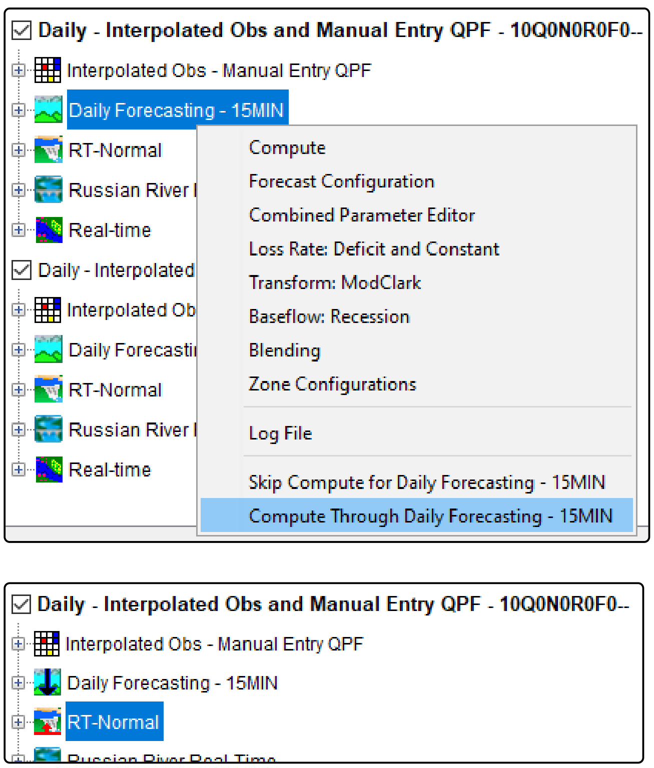
13.4.2 Model Alternatives – Shortcut Menu

From the **Forecast Runs Detail** section (Figure 3) of the Modeling module, under a forecast run name, is the list of model alternatives that are associated with the forecast run. When you right-click on a model alternative name, a shortcut menu will display (Figure 3). From this shortcut menu, you can perform many processes. Further details about these processes are detailed in Model Alternatives and Forecast Runs.

From the shortcut menu you can compute individual model alternatives by clicking **Compute** (Figure 3). You can also identify whether you want a model alternative to be skipped during a forecast run or have the forecast run compute through a selected model alternative.

To skip a model alternative computing in a forecast run, right-click on a model alternative, from the shortcut menu (Figure 3), click **Skip Compute**. The icon of the model alternative will now have a down blue arrow drawn through it (Figure 4), and the **Skip Compute** item in the shortcut menu will have a have checkmark.

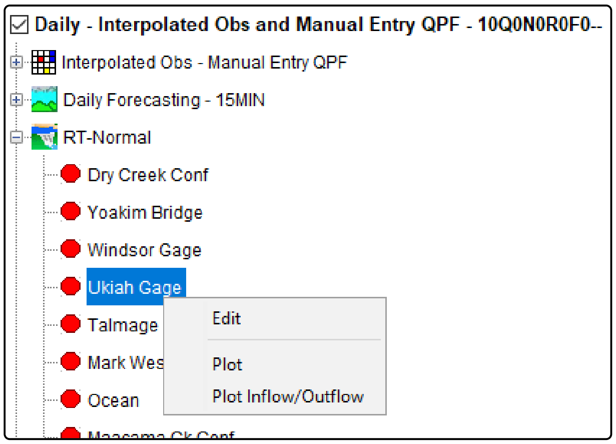
To have a forecast run compute through a selected model alternative, from the shortcut menu (Figure 3), click **Compute Through**. The icon of the model alternative will now have a red line and a red up arrow drawn through it (Figure 4), and the **Compute Through** item in the shortcut menu will have a have checkmark.



60 Figure 4 Model Alternative - Shortcut Menu - Skip Compute and Compute Through

13.4.3 Model Alternative Map Schematic Element – Shortcut Menu

From the **Forecast Runs Detail** section (Figure 5) of the Modeling module, you can expand a model alternative (an MFP alternative does not have map schematic elements), which lists the map schematic elements associated with that model alternative. When you right-click on an element, a shortcut menu will display (Figure 5). Depending on the model alternative and the map schematic element, the shortcut menu will display different options. In addition, on the **Model Schematic** that is displayed in the **Map Window**, the selected element will be highlighted.



61 Figure 5 Model Alternative Map Schematic Element - Shortcut Menu

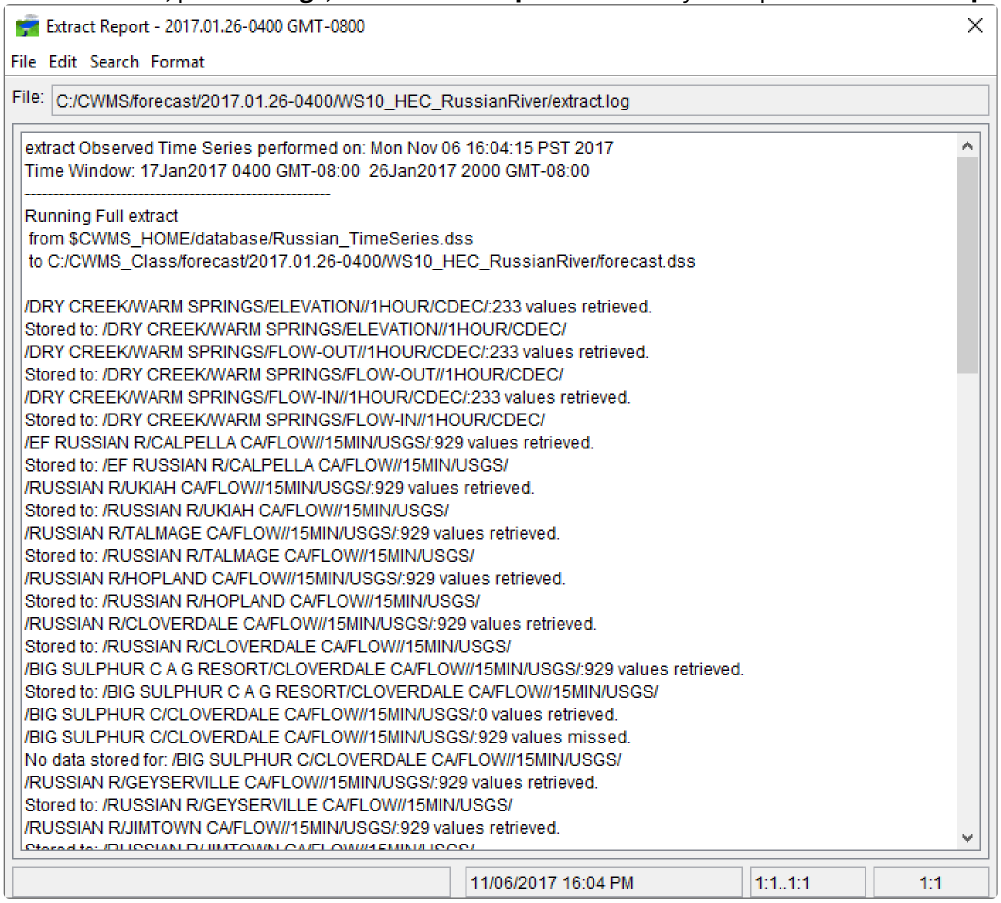
13.5 Review Forecast Data

Now that you have created a forecast, you are ready to compute the forecast runs associated with that forecast. Before you do a compute, you need to check the data that was retrieved from the defined DSS file(s) contained in the extract groups that will be used during the compute.

To review forecast data:

1. From the HEC-RTS main window, click the **Modeling** tab, from the **Forecast** tabs (Modeling Module - Forecast Information Displayed), click **Reports**, click **Extract Report**. Another way to review forecast data, from the

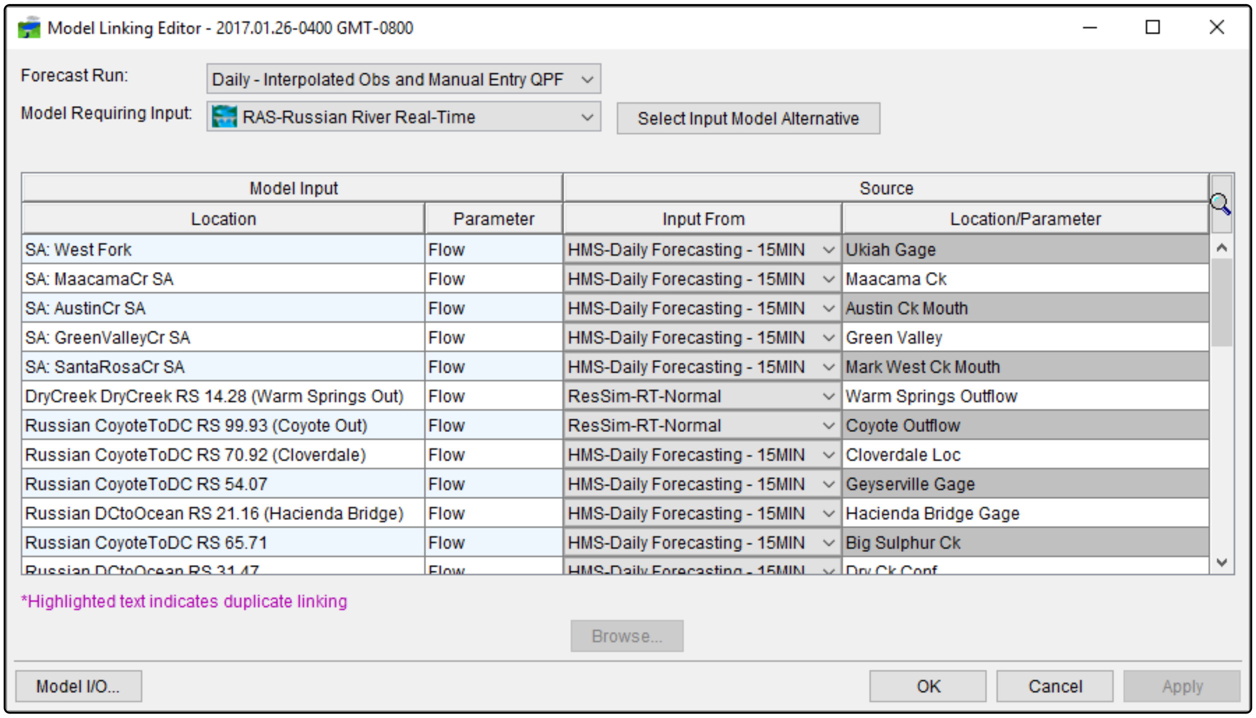
Forecast menu, point to **Logs**, click **Extract Report**. Either way will open the **Extract Report** dialog (Figure 1).



2. The **Extract Report** displays the log file that was generated when the extract process was executed. The name of the active forecast appears in the title bar of the **Extract Reports** dialog (Figure 1). The location of the log file is displayed in the **File** box (Figure 1).
3. Displayed are the results from the retrieval of precipitation data and the data needed by the models associated with the selected forecast. The model data is displayed in the first part, with the precipitation data being display last.
4. If errors are shown, correct the data using HEC-DSSVue. Remember you are only correcting or editing the data for the active forecast.
5. When you have finished reviewing the extract results, from the **File** menu (Figure 1), click **Close**, the **Extract Report** dialog closes.

13.6 Model Linking

Now that you have created forecast runs, created a forecast, and have reviewed the data for a forecast, you need to link the model alternatives that are part of the forecast runs that are associated with the forecast. From the HEC-RTS main window, click the **Modeling** tab, from the **Forecast** menu, click **Model Linking**. The **Model Linking Editor** will open (Figure 1), for each defined forecast run, you will be linking each model alternative. For example, you will want to use output from your HEC-HMS and HEC-ResSim models alternatives as input to your HEC-RAS model alternative. Model Linking Editor provides further details on model linking.



62 Figure 1 Model Linking Editor

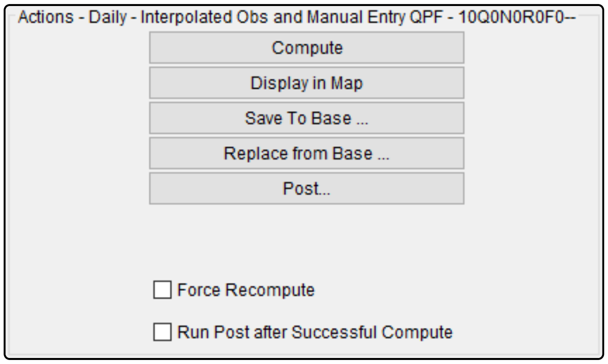
13.7 Executing an HEC-RTS Forecast

Now you are ready to compute an HEC-RTS forecast. The Modeling module (Modeling Module - Forecast Information Displayed) displays the time information associated with an active forecast, and a list of forecast runs associated with that active forecast. For the active forecast, you can compute for a single forecast run or for compute multiple forecast runs. There are several ways to compute an HEC-RTS forecast and the following sections describe in detail how you would execute a single forecast run or execute multiple forecast runs.

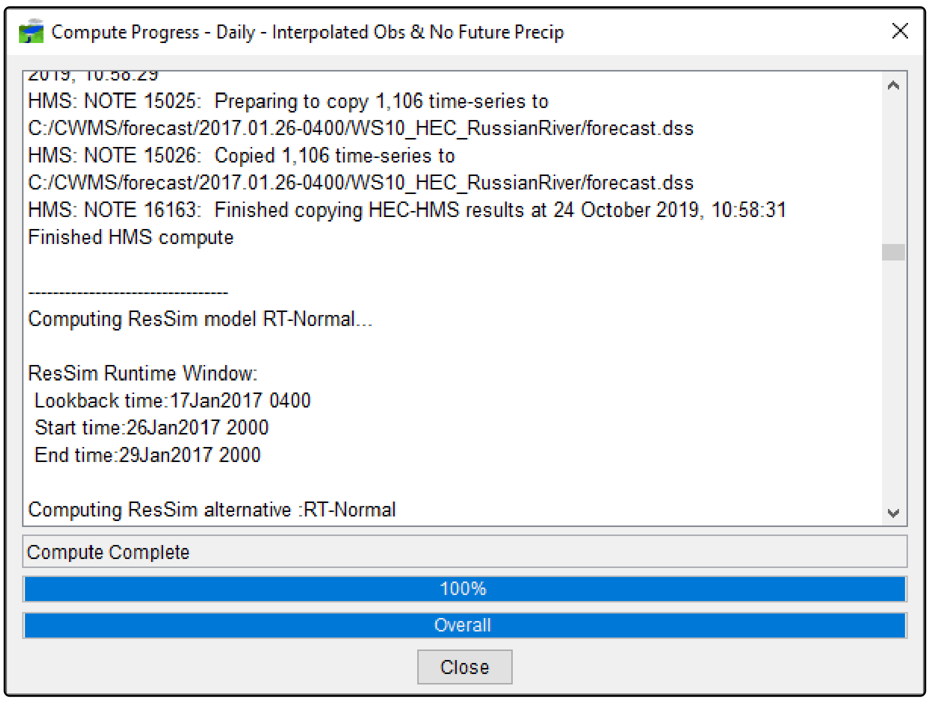
You can also change the behavior of a forecast run compute, by telling the compute to skip a model alternative (Forecast Run Details) or tell the compute to execute through a specific model alternative (Forecast Run Details).

13.7.1 Computing a Single Forecast Run

When a forecast run is highlighted, from the HEC-RTS main window, click the **Modeling** tab, from the **Forecast Tabs**, click **Actions** (Figure 1). Click **Compute** (Figure 1), the **Compute Progress** dialog will open (Figure 2) and the highlighted forecast run will be computed.



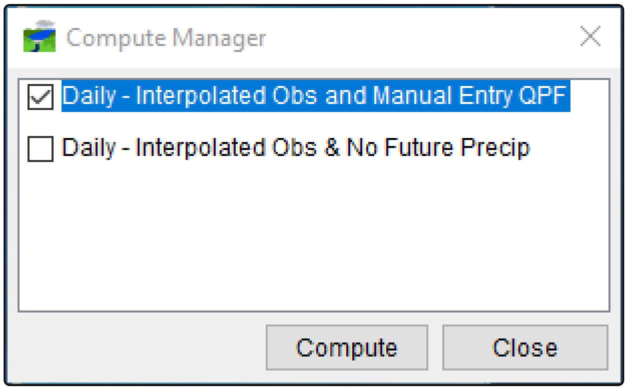
63 Figure 1 Forecast Tab - Actions



64 Figure 2 Compute Progress Dialog

Another option for computing a single forecast run is, from the **Forecast Run Details Area** of the Modeling module, right-click on a forecast run, from the shortcut menu, click **Compute**. The **Compute Progress** dialog will open (Figure 2) and the highlighted forecast run will be computed.

An additional way to compute an individual forecast is, from the HEC-RTS main window, click the **Modeling** tab, from the **Forecast** menu, click **Compute Manager**. The **Compute Manager** dialog (Figure 3) will open, select the forecast run your wish to compute, click **Compute**. The **Compute Progress** dialog will open (Figure 2) and the highlighted forecast run will be computed.



13.7.2 Computing Multiple Forecast Runs

HEC-RTS provides you with a way to execute more than one forecast run at one time, from the main window, click the **Modeling** tab, from the **Forecast** menu, click **Compute Manager**. The **Compute Manager** dialog (Figure 3) will open, select all forecast runs you want to compute, click **Compute**. The **Compute Progress** dialog will open (Figure 2) and the highlighted forecast run will be computed.

13.7.3 Recompute Forecast Runs

When a forecast run is executed, HEC-RTS remembers what HEC-RTS models executed for that forecast run. If changes were made to a model in the forecast sequence, HEC-RTS will re-compute the model when you compute the sequence. If no changes were made to a model and the model was previously computed, HEC-RTS will not recompute the model in the sequence. If for some reason you need to recompute the entire model sequence for a forecast run, from the Modeling module, from the **Actions** tab (Figure 1), click the **Force Recompute** check box. This will "force" a full recompute of the selected forecast run when you click **Compute** (Figure 1). The **Compute Progress** dialog (Figure 2) will open and all the models will be run.

13.8 Viewing Results

After you have computed your active forecast runs, you can now review the results in the Modeling module. Results include the log files, which display computational messages from some of the HEC-RTS software applications, reports from the individual HEC-RTS software applications, and some of the HEC-RTS software applications have results available in a graphical form from the **Map Window**. More detail on viewing results can be found in HEC-RTS Results.

13.9 Calibrating and Editing Models

You may want to run the models individually for calibration before doing a full compute of the forecast run. This is particularly important for HEC-HMS because there are event-dependent parameters (loss rate and baseflow) that you will need to adjust for the forecast. HEC-HMS will produce simulation hydrographs, which you can compare with observed hydrographs up to the time of forecast. These comparisons will determine if you need to adjust your HEC-HMS model.

In the case of HEC-ResSim, you are using the simulated releases as a guide. After reviewing the releases, you might decide that you need to adjust the releases to better simulate actual reservoir operations. For HEC-RAS, you are checking profile information throughout the lookback period; if results are not satisfactory, you can calibrate HEC-RAS by modifying "n-values" at selected cross sections. In the case of HEC-FIA, you might need to adjust the levee failure stages to see the associated damage and impacts during an event.

Each HEC-RTS software application has its own individual editor; these editors are described in detail in Model Alternatives and Forecast Runs. For each HEC-RTS software application, there are only certain parameters that you can edit. If you need to make additional changes beyond those specific parameters, you will have to revise the model alternative using the model's standalone application and re-import the model into your HE-RTS watershed.

In Version 3.1.1, you can edit any of the model data because of the native module interface used. Care should be exercised when editing calibrated models configured in HEC-RTS to prevent models from "breaking".

13.9.1 Computing Individual Model Alternatives

To run a forecast through an individual model:

1. From the **Forecast Runs Detail** section (Model Alternative - Shortcut Menu) of the Modeling module, right-click on a model alternative of a forecast run. From the shortcut menu, click **Compute**. Note that the models are computed in the same order as the Program Order you have set up. For example, if you want to compute a forecast run through the HEC-ResSim model alternative, right-click on the HEC-ResSim model alternative.
2. The **Compute Progress** dialog (Compute Progress Dialog) opens and HEC-RTS begins execution of the HEC-RTS software applications up to the model that you have chosen. For example, if you choose to run through **HEC-ResSim**, HEC-RTS will compute the **MFP**, **HEC-HMS**, and **HEC-ResSim** model alternatives.

After the completion of the computation, you will need to review reports to help in your calibration. HEC-RTS Results describes in detail the available reports for each individual model.

13.9.2 Editing Individual Model Parameters

After you have made your initial run and reviewed results, you might decide you need to edit parameters for an individual model. From the **Forecast Runs Detail** section (Model Alternative - Shortcut Menu) of the Modeling module, depending on the model alternative that you have right-click on, the shortcut menu displays a range of editing options (Model Alternative - Shortcut Menu). The editor associated with that forecast run, opens and you can edit the necessary parameters. Each model has its own individual editor, which are described in detail in Model Alternatives and Forecast Runs.

13.10 Managing Forecast Data

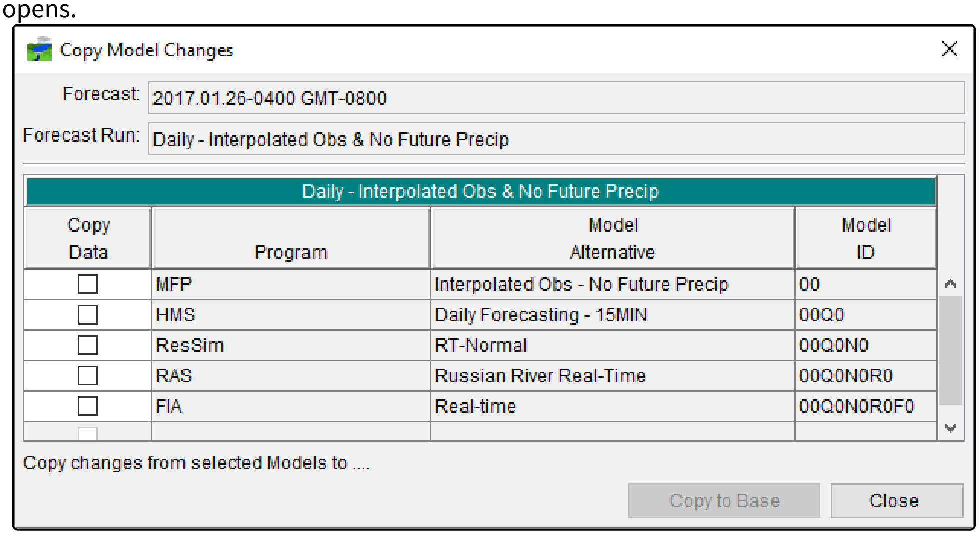
HEC-RTS facilitates archiving and sharing of forecast data through four different mechanisms. One way is to save changes to your selected forecast's model alternatives to the base directory to make them available for other forecasts. Another way is to replace an HEC-RTS model alternative for a specific forecast with the version in the base directory. In addition, you can post your results to the defined DSS file(s) contained in the extract groups, to make the results available to other users; and, the forecast can be frozen to prevent additional computes and further editing of HEC-RTS model alternatives.

13.10.1 Saving Changes to the Base Directory

When you have edited parameters, your changes apply only to an individual forecast run and are saved in the forecast directory (see HEC-RTS Application Settings for an overview of the HEC-RTS directory structure). If you want your changes to be available for subsequent forecasts, you will need to save the changes back to the base directory.

To save data to the base directory:

1. From the **Forecast Runs Detail** section (Forecast Run - Shortcut Menu) of the Modeling module, right click on a forecast run name, from the shortcut menu, click **Save To Base**. The **Copy Model Changes** dialog (Figure 1) opens.



2. The **Copy Model Changes** dialog (Figure 1) displays the name of the **Forecast** and **Forecast Run** that are active. The table displays the **Programs**, **Model Alternatives**, and **Model IDs** that makeup the active forecast run.
3. From the **Copy Data** column (Figure 1) of the table, select the check boxes beside the **Model Alternative** you wish to copy from the forecast directory to the base directory.
4. When you have selected the appropriate **Model Alternatives**, click **Copy to Base** (Figure 1). HEC-RTS will now save all files associated with the selected model alternative back to the base directory. When HEC-RTS is finished, a confirmation message will appear telling you that all selected model alternatives were successfully saved.

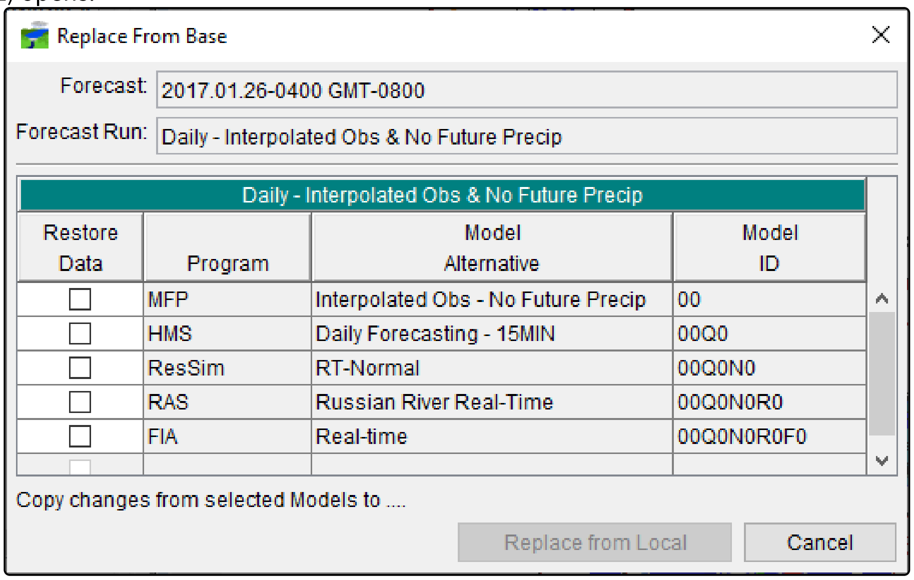
Note that when you have edited model parameters, the **Copy Model Changes** dialog (Figure 1) will also display when you leave the current forecast by exiting HEC-RTS, creating a new watershed, creating a new forecast, opening an existing forecast, or closing the current forecast. If you want your changes to be available for subsequent forecasts, you will need to save the data back to the base directory. If you do not want to save your changes for later forecasts, click **Close**.

13.10.2 Replacing Data from the Base Directory

If, while editing a model alternative, you need to revert to the original model alternative (or propagate modifications made through to the forecast), you can replace the changed model alternative in your forecast directory with the model alternative from the base directory.

To replace a model alternative from the base directory:

1. From the **Forecast Runs Detail** section (Forecast Run - Shortcut Menu) of the Modeling module, right click on a forecast run name, from the shortcut menu click **Replace from Base**, and the **Replace From Base** dialog (Figure 2) opens.

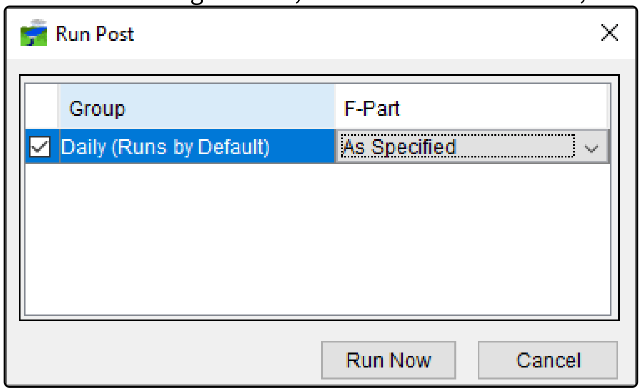


2. The **Replace From Base** dialog (Figure 2) displays the name of the **Forecast** and **Forecast Run** that are active. The table displays the **Programs**, **Model Alternatives**, and **Model IDs** that makeup the active forecast run.
3. From the **Restore Data** (Figure 2) column of the table, select the check boxes beside the **Model Alternative** you wish to restore from the base directory to the forecast directory.
4. When you have selected the appropriate **Model Alternative**, click **Replace from Local** (or **Replace From Server**) (Figure 2). The program will now restore all files associated with the selected **Model Alternative** from the base directory to the forecast directory. When HEC-RTS is finished, a confirmation message will display telling you that all selected HEC-RTS model alternatives were successfully replaced.

13.10.3 Posting Results to the HEC-RTS Watershed

Once you have run the forecast runs associated with your forecast and you are satisfied with the results, you will have one or more forecast runs that represent final forecast results. At this stage, you will want to post your modeling results to the defined DSS file(s) contained in the post groups again to make them available to other users. You will need to create **Post Groups**, which are detailed in Data Extract and Post Editors.

1. From the Modeling module, from the **Forecast** menu, click **Post**, the **Run Post** dialog will open (Figure 3).

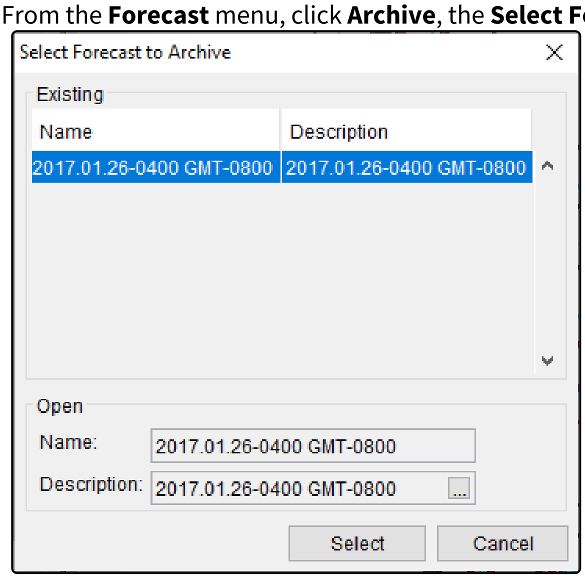


2. All defined **Post Groups** associated with the active forecast appear in the table. In the **Group** column, select the **Post Groups** that will post results to the defined DSS file(s).
3. Click **Run Now**, the **Message Pane** on the **Post Forecast Results** dialog (Model Alternative - Shortcut Menu - Skip Compute and Compute Through) will notify you that a post action has been sent to the DBI.
4. When the data has been successfully posted to the defined DSS file(s), the **Message Pane** of the HEC-RTS main window (Modeling Module - Forecast Information Displayed) will now contain a new tab - **Post**. This will notify you that a post action has been successful.

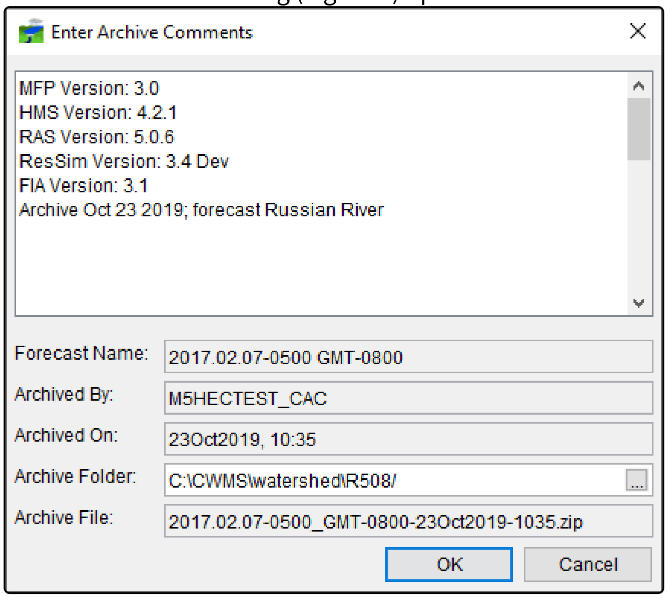
13.10.4 Archive a Forecast

Once you have a configured a forecast you may want to keep an archive your forecast for future retrieval. To archive a forecast:

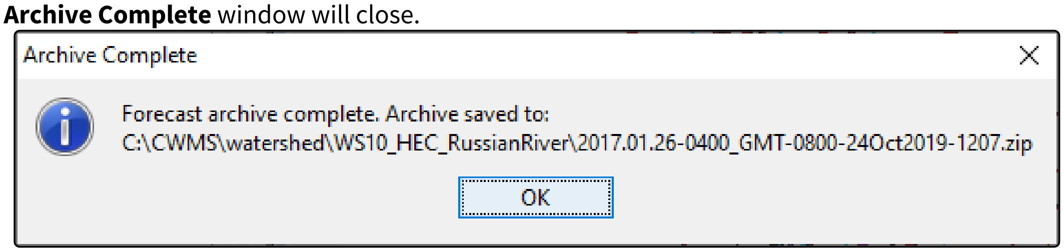
1. If you have the forecast you are going to archive open, close the forecast. From the Modeling module, from the **Forecast** menu, click **Close**.
2. From the **Forecast** menu, click **Archive**, the **Select Forecast** dialog (Figure 4) opens.



3. Select the forecast name you wish to archive from the list in the **Existing** box (Figure 4), click **Select**. The **Enter Archive Comments** dialog (Figure 5) opens.



4. Enter your comments in the field at the top of the dialog box and from the **Archive Folder** box (Figure 5) enter a folder to save the archived forecast.
5. Click **OK**, an **Archiving** progress window will display while the archiving process is working. When finished the **Archiving** progress window will close and an **Archive Complete** window will display (Figure 6). Click **OK**, the **Archive Complete** window will close.



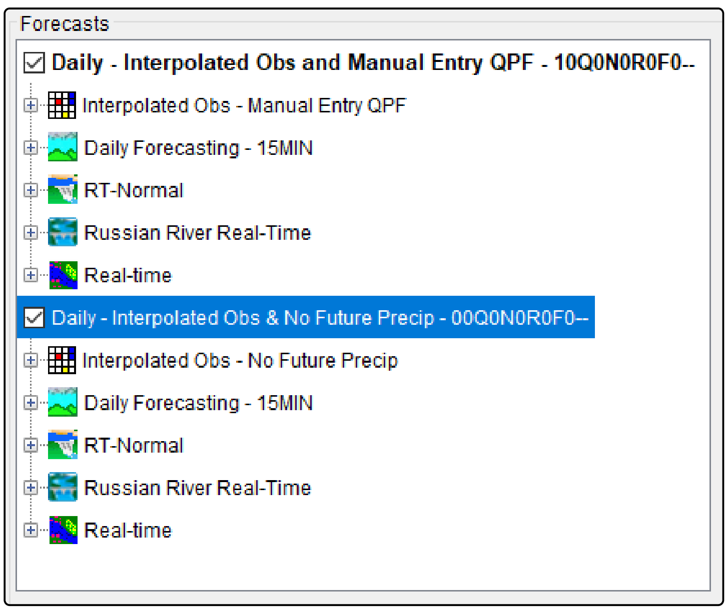
6. If the forecast has multiple forecast runs, the **Save Models to Base** dialog (like Figure 2) will open. If this dialog opens, you must select the model alternatives you wish to include in the archive. Once the model alternatives are selected click **Save to Local** and the **Progress** widow will open as well as the archiving dialog will be displayed for a short while archiving the forecast files and closes.

14 HEC-RTS Results

After a forecast has been executed, results can be reviewed from the Modeling module. Results include log files (forecast; forecast run; individual software application); a report on the extract process (forecast); a report on the post process (forecast); access to the forecast DSS file; and, reports from the individual software applications. From the **Map Window** some of the software applications (HEC-HMS, HEC-ResSim, HEC-RAS) have results available in a graphical form; and from time series icons, results are available in a graphical or tabular form. If the HEC-RAS model included the creation of inundation data, this data can be viewed from the RAS Mapper Tool, available through the HEC-RAS software application.

14.1 HEC-RTS Results Overview

From the Modeling module (Chapter 11), open an existing forecast, once the forecast is loaded, the **Forecast Runs Details** area (Figure 1) will display the forecast runs associated with the selected forecast. When a forecast run is bolded (*Daily – Interpolated Obs and Manual Entry*), this means that the forecast run is being displayed in the active **Map Window** of the Modeling module.

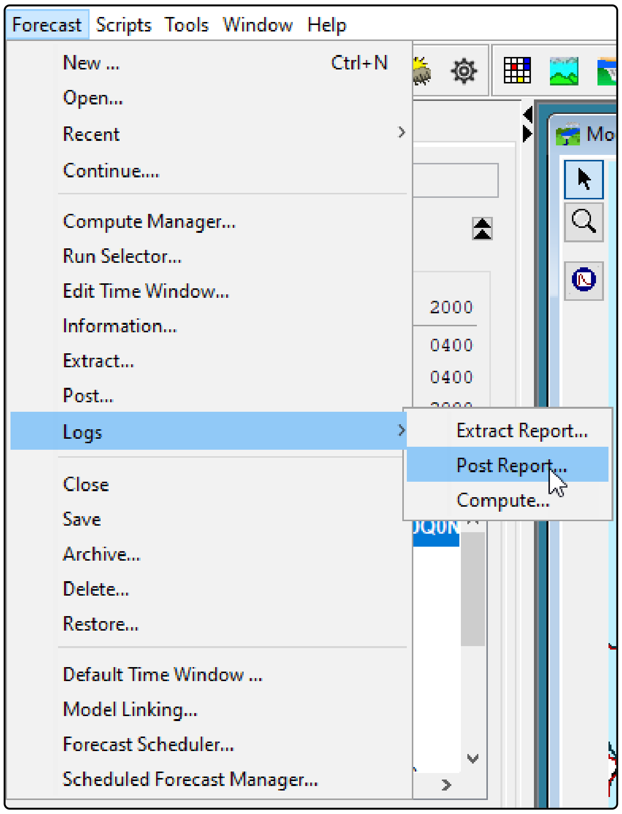


65 Figure 1 Modeling Module - Forecast Run Details Section

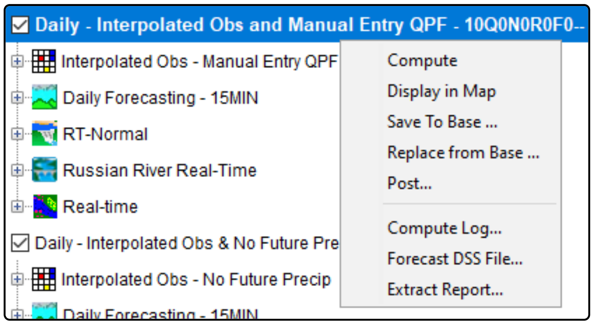
When you have clicked the checkbox next to a forecast run (Figure 1), you are indicating that you want to view output from the model schematic that is displayed in the **Map Window** for the selected forecast run. You can select multiple forecast runs, however, in this version of HEC-RTS, the only software application that displays output results for multiple forecast runs is HEC-ResSim. In addition, the time series icons will display output for multiple forecast runs. MFP will display separate summary reports for each forecast run.

14.2 General Reports

There are three general reports - **Extract Report**, **Post Report**, and **Compute Log**. The **Compute Log** report can provide details on the forecast compute; the compute for each forecast run associated with a forecast; and, the compute for each model alternative associated with a forecast run. There are several ways to access these general reports in the Modeling module. From the HEC-RTS main window, from the **Forecast** menu, point to **Logs** (Figure 1), the three reports are available.

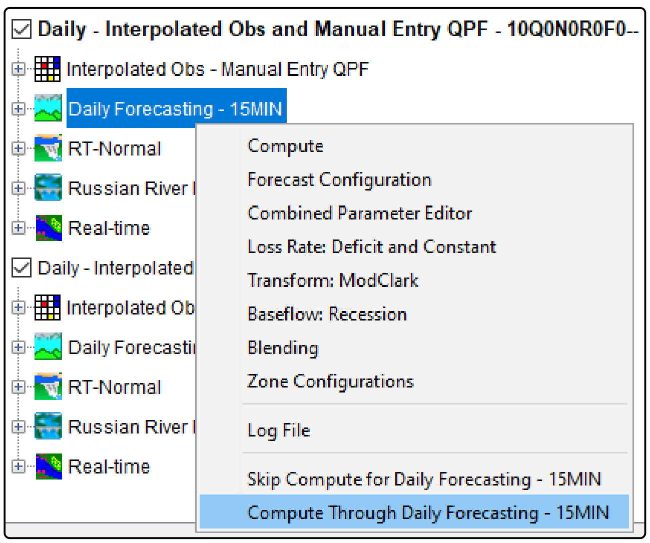


From the **Forecast Runs Detail** section (Figure 2) of the Modeling module, when you right click on a forecast run name (*Daily – Interpolated Obs and Manual Entry QFP*), a shortcut menu will display (Figure 2). From this shortcut menu, you can access two of the general reports - **Extract Report** and **Compute Log**.



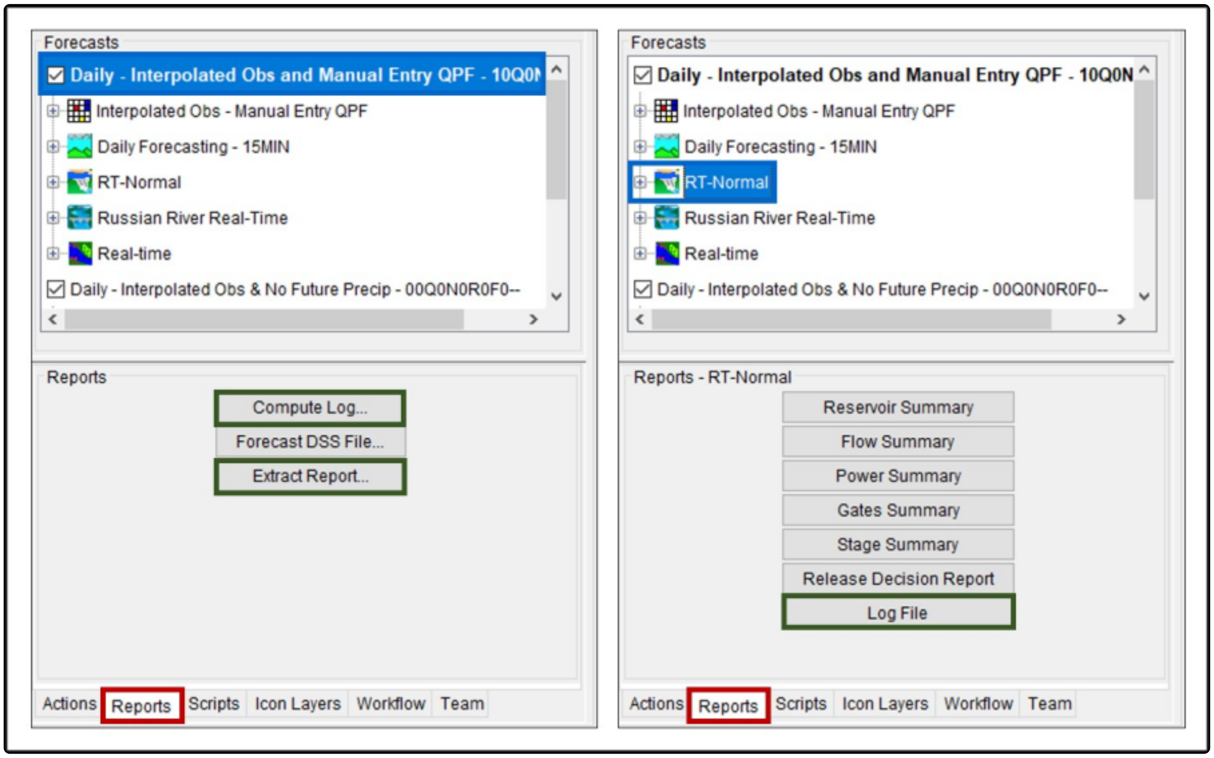
66 FIgure 2 Forecast Runs Detail Section - Forecast Run - Shortcut Menu

From the **Forecast Runs Detail** section (Figure 3) of the Modeling module, under a forecast run name (*Daily – Interpolated Obs and Manual Entry QFP*), is the list of model alternatives that are associated with the forecast run. When you right-click on a model alternative name, a shortcut menu will display (Figure 3). From this shortcut menu, you can access the **Compute Log** (Log File) for the selected model alternative.



67 FIgure 3 Forecast Runs Detail Section - Model Alternative - Shortcut Menu

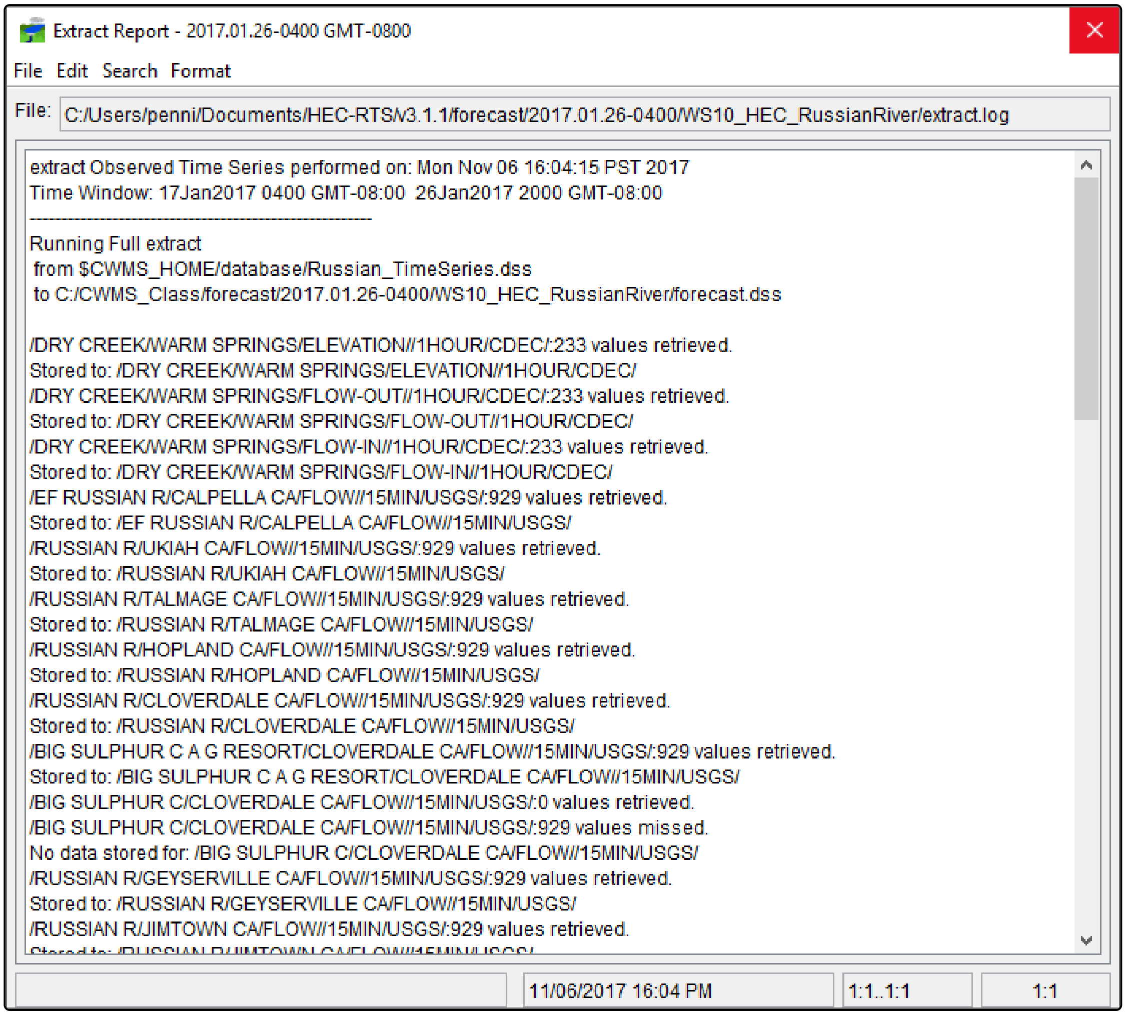
In addition, from the **Forecast Runs Details** section when you select a forecast run or a model alternative, from the **Forecast Tabs** (Figure 4), click **Reports**, the **Compute Log** and the **Extract Report** (only forecast runs) are available.



68 Figure 4 Forecast Runs Detail Section - Forecast Tabs - Reports Tab

14.2.1 Extract Report

The **Extract Report** provides details on what occurred during the extract process for each extract group included in the extract process. To access the **Extract Report** (Figure 5):

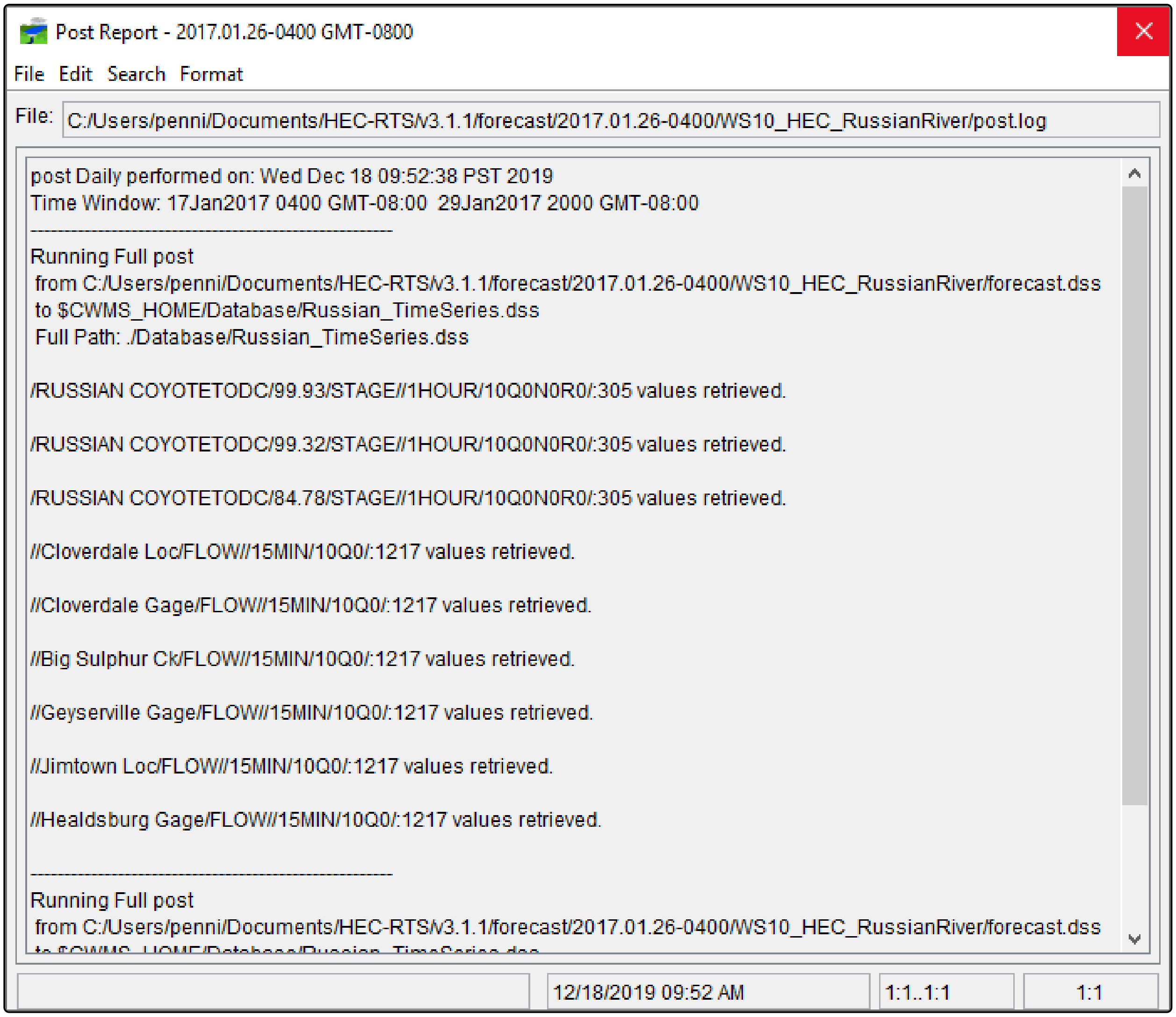


69 Figure 5 Extract Report Dialog

1. From the HEC-RTS main window, access the **Extract Report** from one of the ways detailed above, click **Extract Report**, the **Extract Report** dialog will display (Figure 5). The name of the active forecast appears in the title bar of the **Extract Reports** dialog (Figure 5). The location of the log file is displayed in the **File** box (Figure 5).
2. Displayed are the results from the retrieval of precipitation data and the data needed by the models associated with the selected forecast. The model data is displayed in the first part, with the precipitation data being displayed last.
3. When you have finished reviewing the extract results, from the **Extract Report** dialog, from the **File** menu (Figure 5), click **Close**, the **Extract Report** dialog closes.

14.2.2 Post Report

The **Post Report** provides details on each post group included in the post process. To access the **Post Report** (Figure 6) for a forecast:

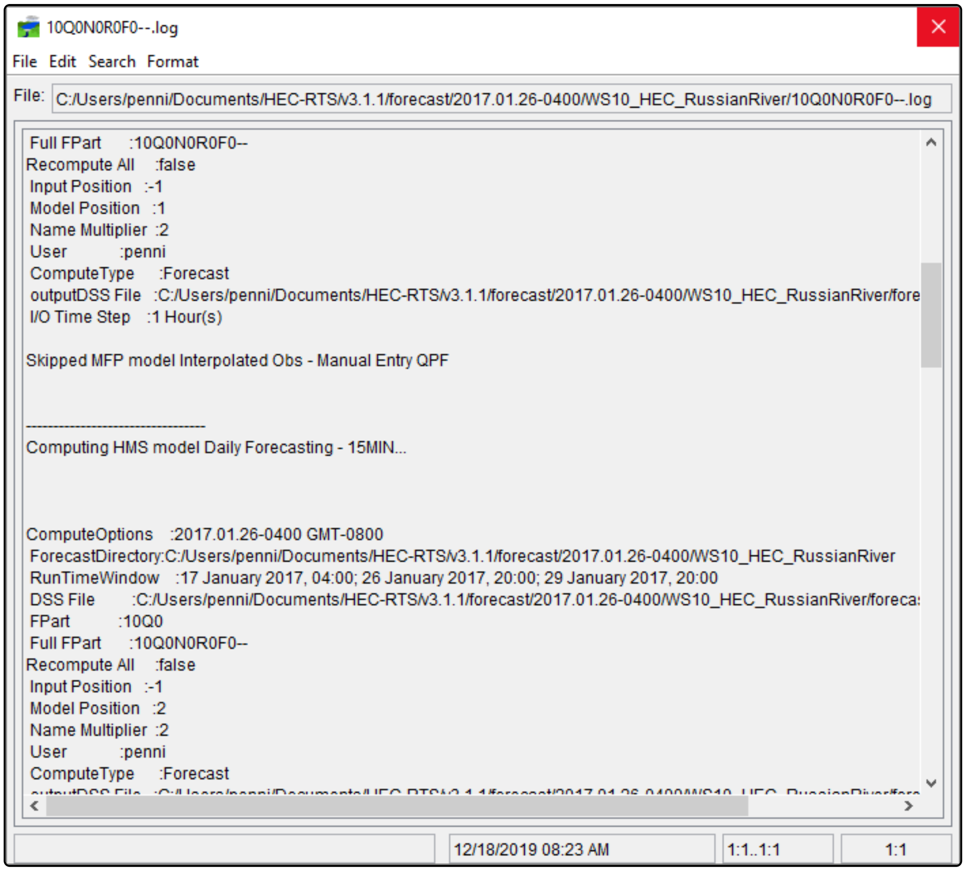


70 Figure 6 Figure 13.7 Post Report Dialog

1. From the HEC-RTS main window, from the **Forecast** menu, point to **Logs**, click **Post Report**, the **Post Report** dialog will display (Figure 6). The name of the active forecast appears in the title bar of the **Post Report** dialog (Figure 6). The location of the log file is displayed in the **File** box (Figure 6).
2. Displayed are the results from the posting of precipitation data and data generated by the models associated with the selected forecast. The model data is displayed in the first part, with the precipitation data being displayed last.
3. When you have finished reviewing the post results, from the **Post Report** dialog, from the **File** menu (Figure 6), click **Close**, the **Post Report** dialog closes.

14.2.3 Compute Log – Forecast Run

When computing a forecast, you select which forecast runs to compute. Once the compute is complete, detailed information regarding the forecast selected forecast run(s) and associated model alternatives that were computed, is available from a **Compute Log** report (Figure 7). To view this report, make sure you have an active forecast and at least one forecast run selected.

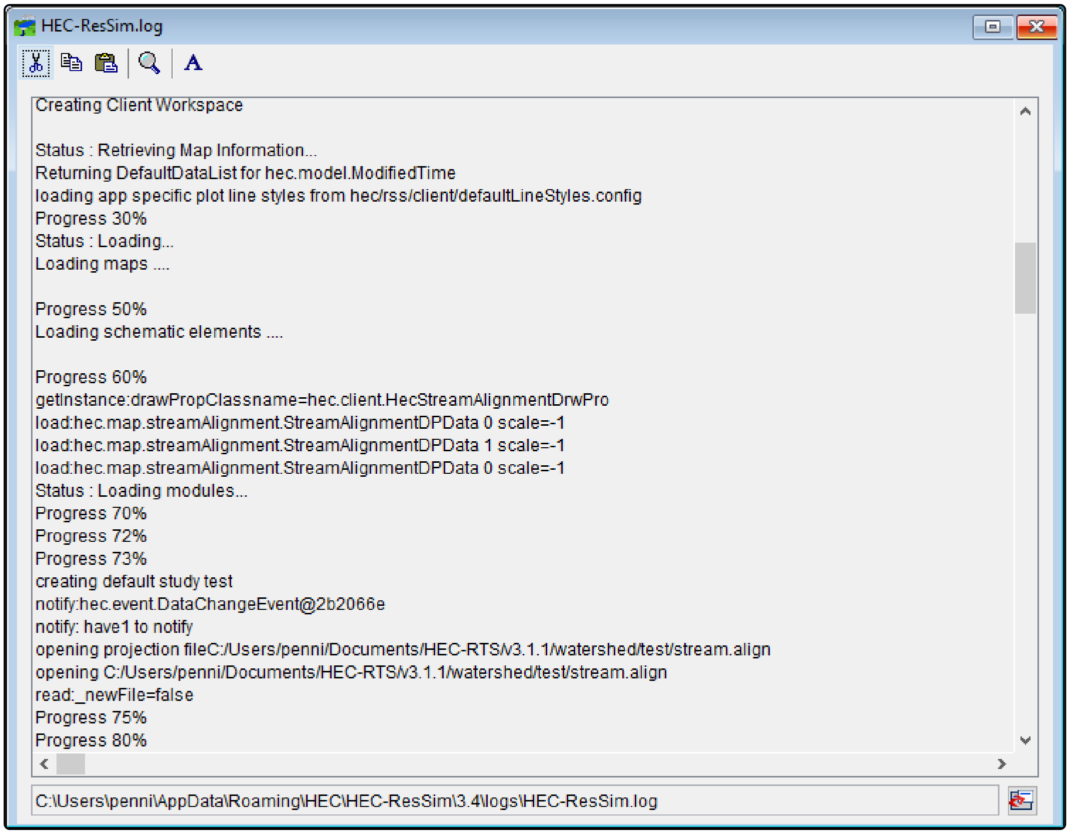


To view the **Compute Log** for a forecast run:

1. From the HEC-RTS main window, click the **Modeling** tab, from the **Forecast Runs Detail** section (Figure 2) right click on a forecast run name (*Daily – Interpolated Obs and Manual Entry QFP*), a shortcut menu will display (Figure 13.3), click **Compute Log**. Another way is once you have selected a forecast run, from the **Forecast** tabs, click **Reports** (Figure 4). Either way, click **Compute Log**, the **Log File** dialog will open (Figure 7).
2. The **Log File** dialog (Figure 7) displays the key code name of the selected forecast run; displays the location and name of the file, which contains the log file; and displays the results of the computation.
3. When you have finished reviewing the forecast compute log, from the **Log File** dialog, from the **File** menu (Figure 7), click **Close**, the **Log File** dialog closes.

14.2.4 Compute Log – Model Alternative

When computing a forecast, you select which forecast runs to compute. Once the compute is complete, if the associated model alternatives software application generates a log file (MFP does not), that detailed information is available from a **Compute Log** report (Figure 8). To view this report, make sure you have an active forecast and at least one forecast run selected.



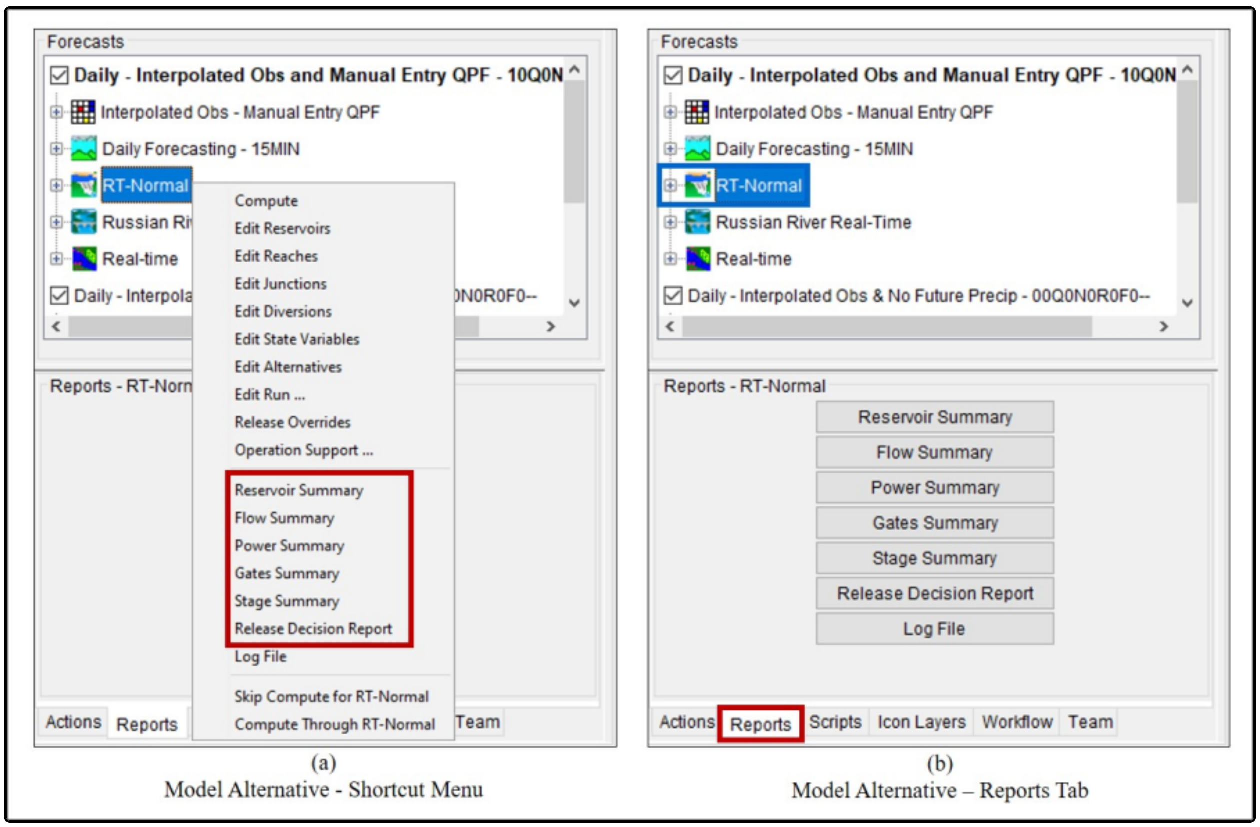
To view the **Compute Log** for a model alternative:

1. From the HEC-RTS main window, click the **Modeling** tab, from the **Forecast Runs Detail** section (Figure 2) select a forecast run (*Daily – Interpolated Obs and Manual Entry QFP*), select a **Model Alternative**, right-click on the model alternative, a shortcut menu will display (Figure 3), click **Log File**. Another way is once you have selected a model alternative, from the **Forecast** tabs, click **Reports** (Figure 4). Either way, click **Log File**, the **Log File** dialog will open (Figure 8).
2. The **Log File** dialog (Figure 8) displays the name of the software application associated with the selected model alternative; displays the location and name of the file, which contains the log file along the bottom of the **Log File** dialog (Figure 8); and displays the results of the computation.
3. When you have finished reviewing the model alternative compute log, from the **Log File** dialog, click  , the **Log File** dialog closes.

14.3 Model Alternative Results

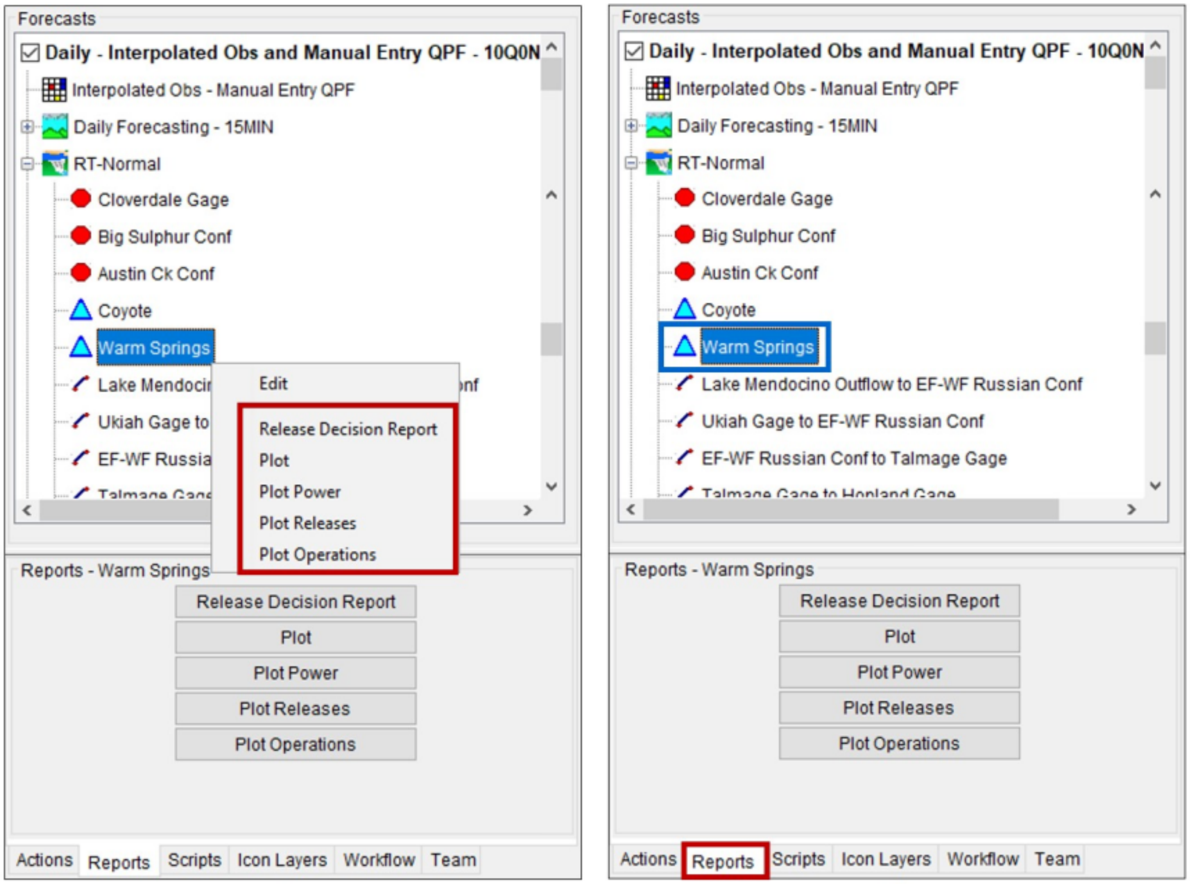
For the selected forecast run(s), the associated model alternatives can provide results in either a tabular form (report) or in a graphical form. Results are only available for model alternatives if the associated software application provides those results to HEC-RTS. From the Modeling module, model alternative results are available from the **Forecast Runs Detail** section, the **Forecast** tabs, or the **Map Window**.

From the **Forecast Runs Detail** section (Figure 1), you can review results by the selected model alternative, or by the individual model elements associated with a model alternative. For a selected forecast run, select a model alternative, right-click on the model alternative, from the shortcut menu (Figure 1), there are several reports available. Another way to review model alternative results from the **Forecast Runs Detail** section, is select a model alternative, from the **Forecast** tabs, click **Reports** (Figure 1). From the **Reports** tab, there are several reports available for the selected model alternative.



71 Figure 1 Forecast Runs Details Section - Model Alternative - Reports

To review results for individual model elements (e.g., reservoirs, reaches, sinks, cross sections, etc.) from the **Forecast Runs Details** section (Figure 2), next to a model alternative, click ⊞ . For an HEC-HMS, HEC-ResSim, and HEC-RAS model alternative, the individual model elements will display (Figure 2). An MFP alternative does not have any individual model elements, and for an HEC-FIA model alternative, individual model elements will display, but results are only provided at the model alternative level. For a model alternative, right-click on an individual model element, from the shortcut menu (Figure 2), there are several reports available. Another way to review individual model element results from the **Forecast Runs Detail** section, for a model alternative, select an individual model element, from the **Forecast** tabs, click **Reports** (Figure 2).

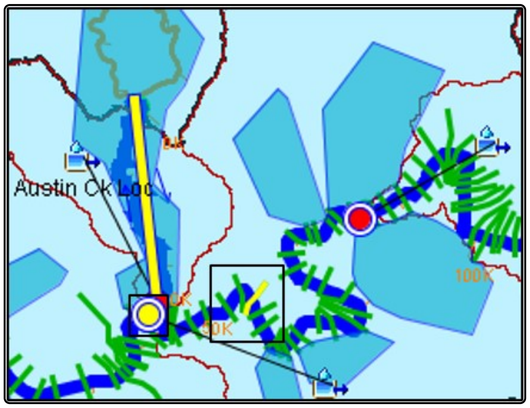


(a) Individual Model Elements - Shortcut Menu

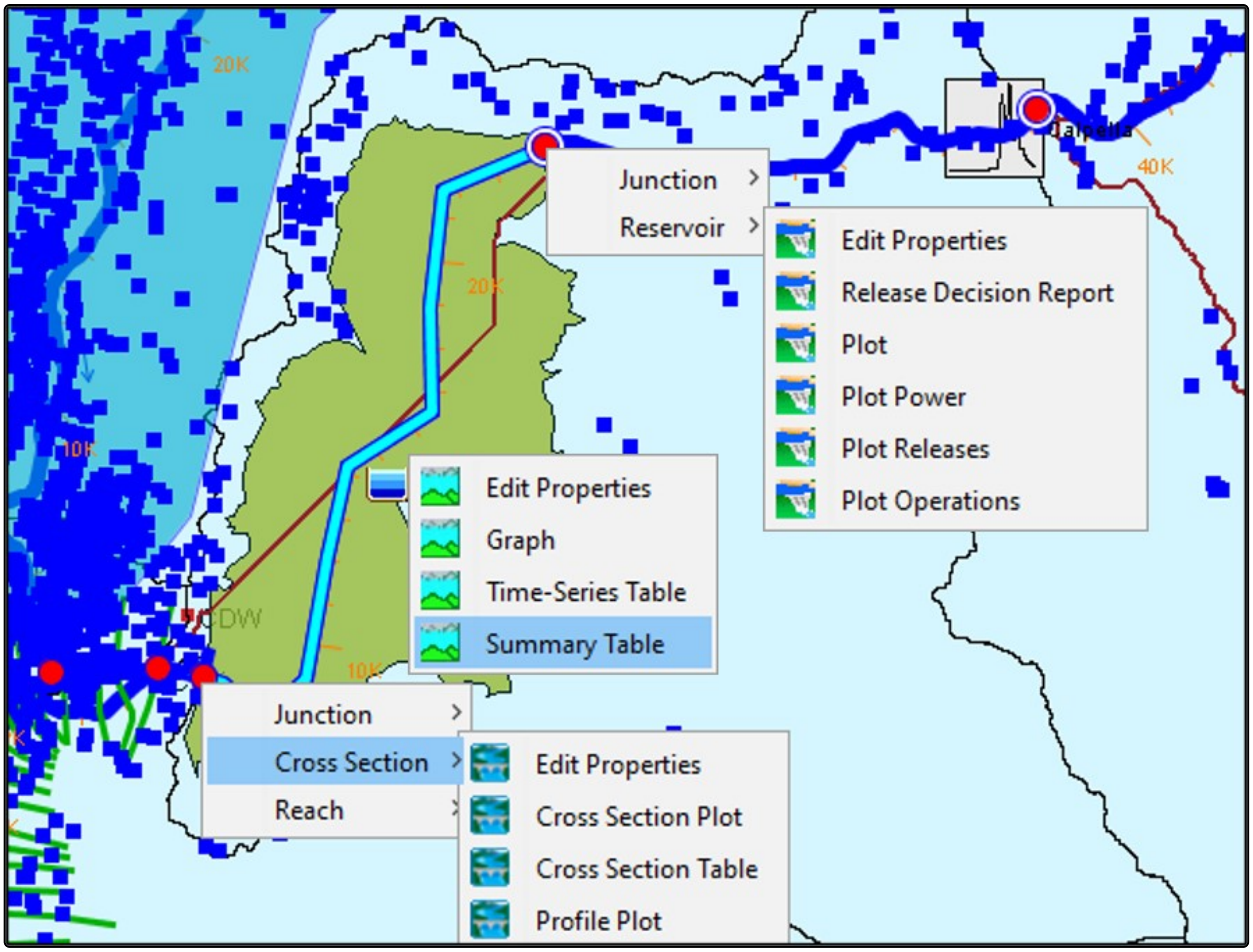
(b) Individual Model Elements – Reports Tab

72 Figure 2 Forecast Runs Details Section - Individual Model Elements - Reports

From the **Reports** tab, there are several reports available for the selected individual model element. In addition, when certain individual model elements are selected, in the **Map Window**, on the schematic (Figure 3), the selected individual model element is highlighted in yellow.



 In the Modeling module, from the **Map Window** (Figure 4), you can also review individual model elements. From the **Map Window**, click (**Select Tool**). From the schematic, right-click on an individual model element, from the shortcut menu (Figure 4), you can review results for the associated model alternatives for a selected forecast run. The available results are either reports or graphical.

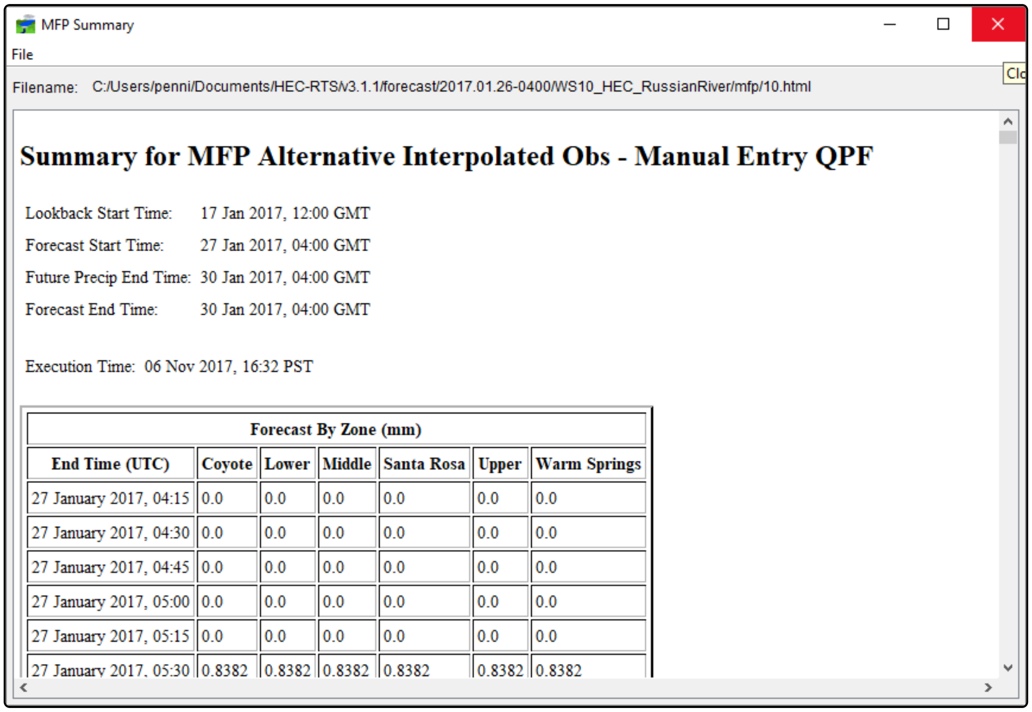


73 Figure 4 Modeling Module - Map Window - Individual Model Element Results

So, as you can see, in HEC-RTS there are various ways to review results. How to access those results will vary, since as this section has detailed, there is just not one way to access results in HEC-RTS.

14.4 MFP Results

The **MFP Summary** dialog (Figure 1) displays information regarding the MFP model alternative associated with a forecast run. The report displays the location and name of the file, which contains the **MFP Summary**. In addition, the report displays the name of the forecast run, the time window used, the date and time of the execution of the MFP model alternative, and the creation of the precipitation grids for observed and future.



74 Figure 1 MFP Summary Report

To view the **MFP Summary** report:

1. To view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results.
2. From the HEC-RTS main window, click the **Modeling** tab, from the **Forecast Runs Details** section, right-click on an MFP model alternative (*Interpolated Obs – Manual Entry QPF*), from the shortcut menu, click **Summary**.
3. In Figure 1, the **MFP Summary** report, displays results for the MFP model alternative *Interpolated Obs – Manual Entry QPF*. The report displays the sequence of precipitation depths entered by zone, and summaries of each grid used in the forecast. Summaries include the maximum, minimum, and average precipitation depth values, and the number of cells in the grid showing precipitation depths greater than zero. Note that the precipitation depths shown in this log file are given in millimeters, regardless of the unit system of the watershed.
4. From the **File** menu, click **Close**, and the **MFP Summary** dialog (Figure 1) closes.

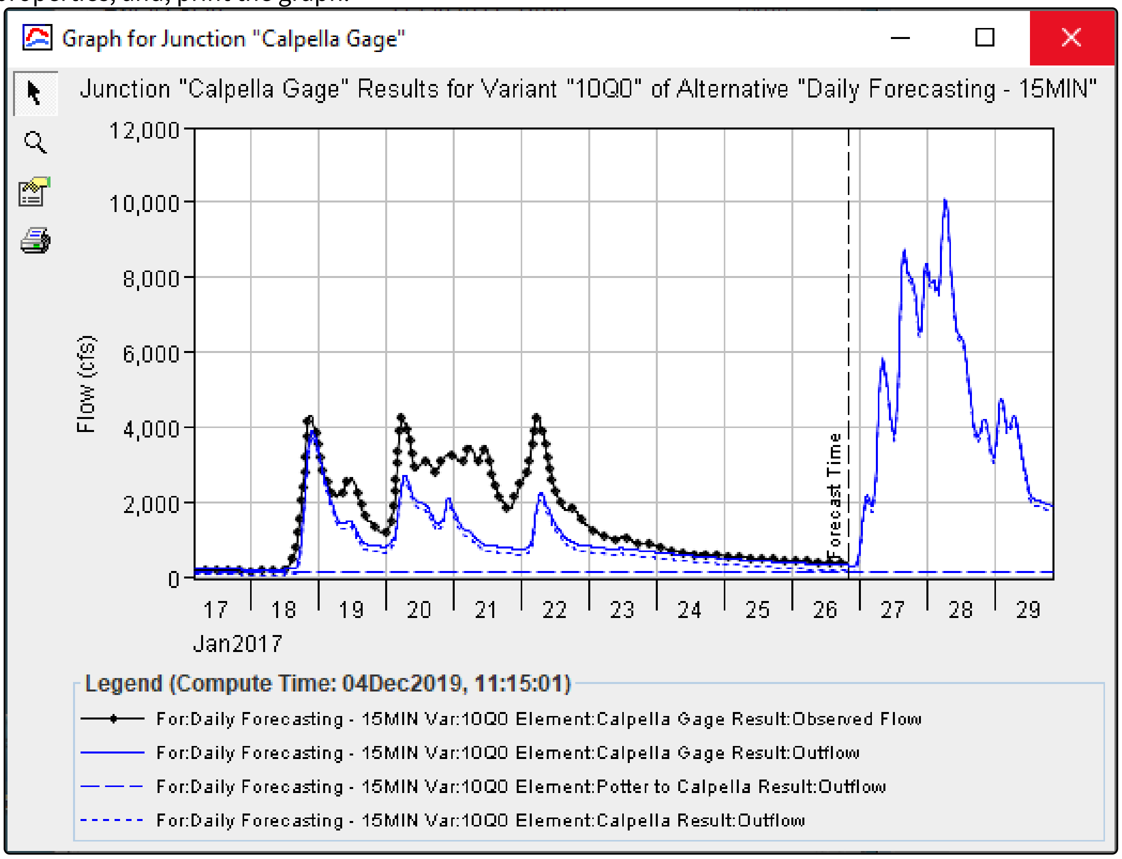
14.5 HEC-HMS Results

HEC-HMS may be used in HEC-RTS to compute watershed runoff, given initial system states and watershed precipitation. HEC-HMS results available from the HEC-RTS interface include a forecast summary report; a table of time-series parameter data, and a plot of the time-series data. These results are available from the individual watershed elements as defined in the HEC-HMS model alternative.

14.5.1 Graphical Results

To view graphical results for individual HEC-HMS model elements (e.g., reaches, sinks):

1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Details** section, next to an HEC-HMS model alternative (*Daily Forecasting – 15MIN*), click . The individual model elements for the selected HEC-HMS model alternative will display.
2. Select an HEC-HMS model element (*Calpella Gage*), from the **Forecast** tabs, click **Reports.** Click **Graph**, a dialog will open (Figure 1), displaying a graph of the selected HEC-HMS model element. The graph provides the name and type of the HEC-HMS model element; you can zoom in/out on graph; make adjustment to the graph's properties; and, print the graph.



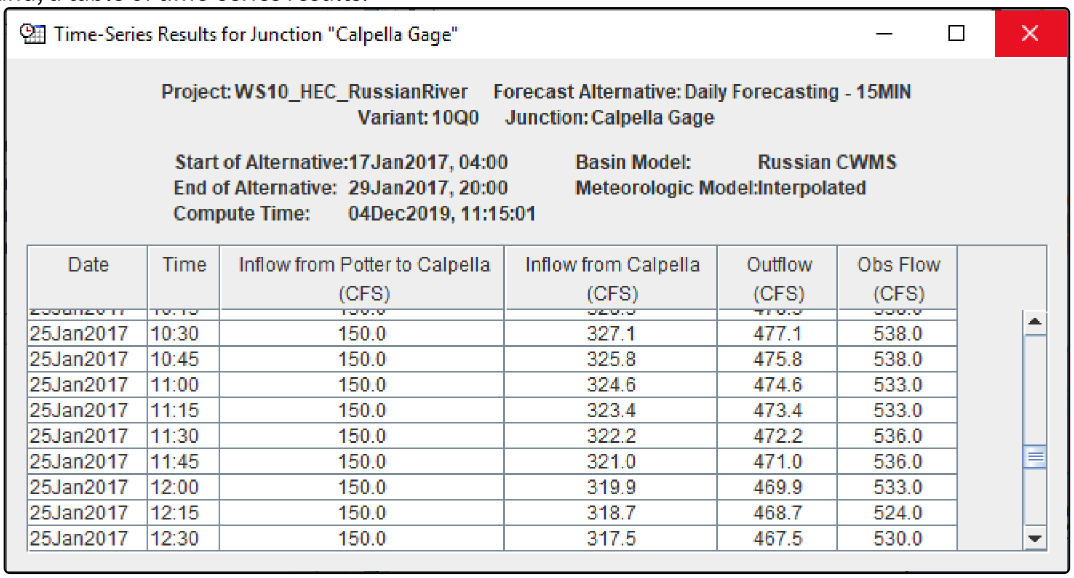
3. From the graphical dialog (Figure 1), click , the dialog will close.

14.5.2 Time-Series Results

To view time-series results for individual HEC-HMS model elements (e.g., reaches, sinks):

1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Details** section, next to an HEC-HMS model alternative (*Daily Forecasting – 15MIN*), click . The individual model elements for the selected HEC-HMS model alternative will display.
2. Select an HEC-HMS model element (*Calpella Gage*), from the **Forecast** tabs, click **Reports**. Click **Time-Series Table**, the **Time-Series Results** dialog will open (Figure 2). For the selected HEC-HMS model element, the **Time-Series Results** dialog, provides information on the watershed; name of the HEC-HMS model alternative; the HEC-HMS model alternative key code (variant); what type of HEC-HMS model element and name; time window

information for the compute; HEC-HMS basin model that is used; HEC-HMS meteorologic model that is used; and, a table of time-series results.



Date	Time	Inflow from Potter to Calpella (CFS)	Inflow from Calpella (CFS)	Outflow (CFS)	Obs Flow (CFS)
25Jan2017	10:30	150.0	327.1	477.1	538.0
25Jan2017	10:45	150.0	325.8	475.8	538.0
25Jan2017	11:00	150.0	324.6	474.6	533.0
25Jan2017	11:15	150.0	323.4	473.4	533.0
25Jan2017	11:30	150.0	322.2	472.2	536.0
25Jan2017	11:45	150.0	321.0	471.0	536.0
25Jan2017	12:00	150.0	319.9	469.9	533.0
25Jan2017	12:15	150.0	318.7	468.7	524.0
25Jan2017	12:30	150.0	317.5	467.5	530.0

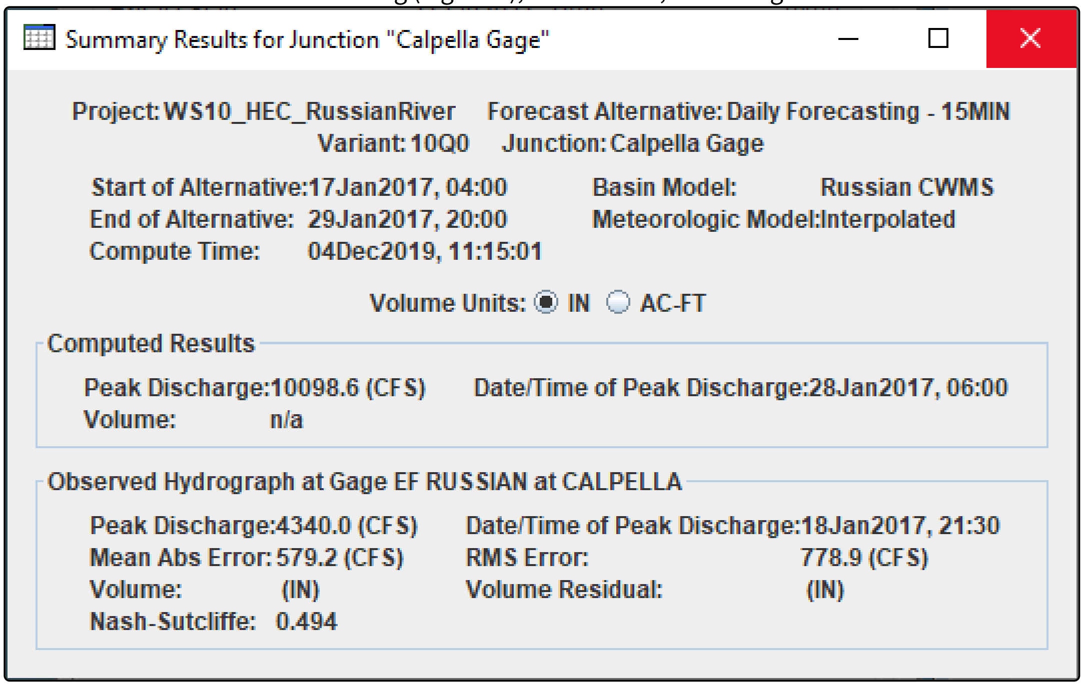
3. From the **Time-Series Results** dialog (Figure 2), click ✕ , the dialog will close.

14.5.3 Summary Results

To view a summary of results for individual HEC-HMS model elements (e.g., reaches, sinks):

1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Details** section, next to an HEC-HMS model alternative (*Daily Forecasting – 15MIN*), click ⊞ . The individual model elements for the selected HEC-HMS model alternative will display.
2. Select an HEC-HMS model element (*Calpella Gage*), right-click on the model element, from the shortcut menu, click **Summary Table**. The **Summary Results** dialog will open (Figure 3). For the selected HEC-HMS model element, the **Summary Results** dialog, provides information on the watershed; name of the HEC-HMS model alternative; the HEC-HMS model alternative key code (variant); what type of HEC-HMS model element and name; time window information for the compute; HEC-HMS basin model that is used; HEC-HMS meteorologic model that is used; volume units; computed results; and, in some cases information about the observed hydrograph at a gage.

3. From the **Time-Series Results** dialog (Figure 3), click  , the dialog will close.



14.6 HEC-ResSim Results

HEC-ResSim may be used in HEC-RTS to simulate operation of a watershed, given inflow and local flow forecasts from HEC-HMS. There are several reports available from HEC-ResSim for the HEC-ResSim model alternative, and for HEC-ResSim model elements. From an HEC-ResSim model alternative you have a series of summary results reports (i.e., reservoir, flow, power, etc.). At HEC-ResSim model elements (i.e., reach, junctions, reservoirs), you can review plots of gage information; plots of inflow/outflow information; plots that provide information on power, releases, and operations at reservoir model elements; and a release decision report.

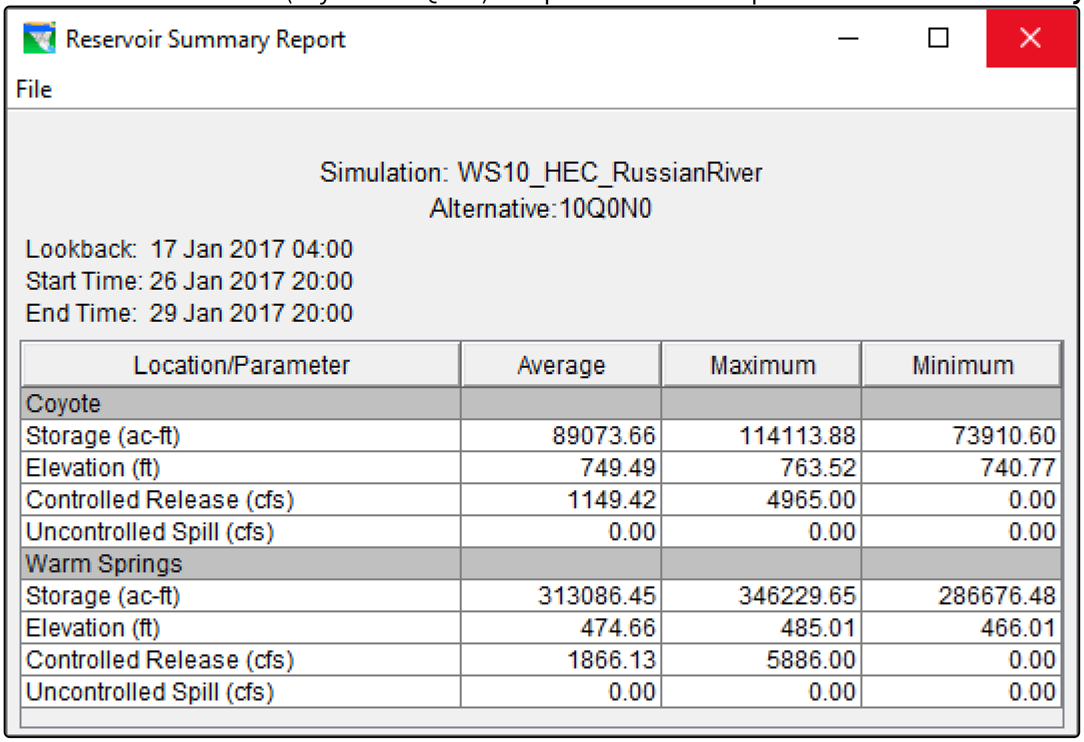
14.6.1 Reservoir Summary Report

The **Reservoir Summary Report** summarizes the reservoirs for the active forecast run. Results include the storage, elevation, controlled release, and uncontrolled spill parameters for each reservoir, and are reported as average, maximum, minimum values for each parameter.

To view the **Reservoir Summary Report**:

1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-ResSim model alternative (*RT-Normal*), from the **Forecast** tabs, click **Reports**.
2. Click **Reservoir Summary**, the **Reservoir Summary Report** (Figure 1) dialog will open. This report provides information about the reservoirs in a watershed. Information includes storage, elevation, controlled release, and uncontrolled spill for each reservoir in a watershed. For each parameter, the report displays the average, maximum, and minimum values. In addition, the report provides information on the time window for the HEC-

ResSim model alternative (key code *10Q0N0*) compute. You can also print the **Reservoir Summary Report**.



Location/Parameter	Average	Maximum	Minimum
Coyote			
Storage (ac-ft)	89073.66	114113.88	73910.60
Elevation (ft)	749.49	763.52	740.77
Controlled Release (cfs)	1149.42	4965.00	0.00
Uncontrolled Spill (cfs)	0.00	0.00	0.00
Warm Springs			
Storage (ac-ft)	313086.45	346229.65	286676.48
Elevation (ft)	474.66	485.01	466.01
Controlled Release (cfs)	1866.13	5886.00	0.00
Uncontrolled Spill (cfs)	0.00	0.00	0.00

3. From the **File** menu, click **Close**, and the **Reservoir Summary Report** (Figure 1) dialog will close.

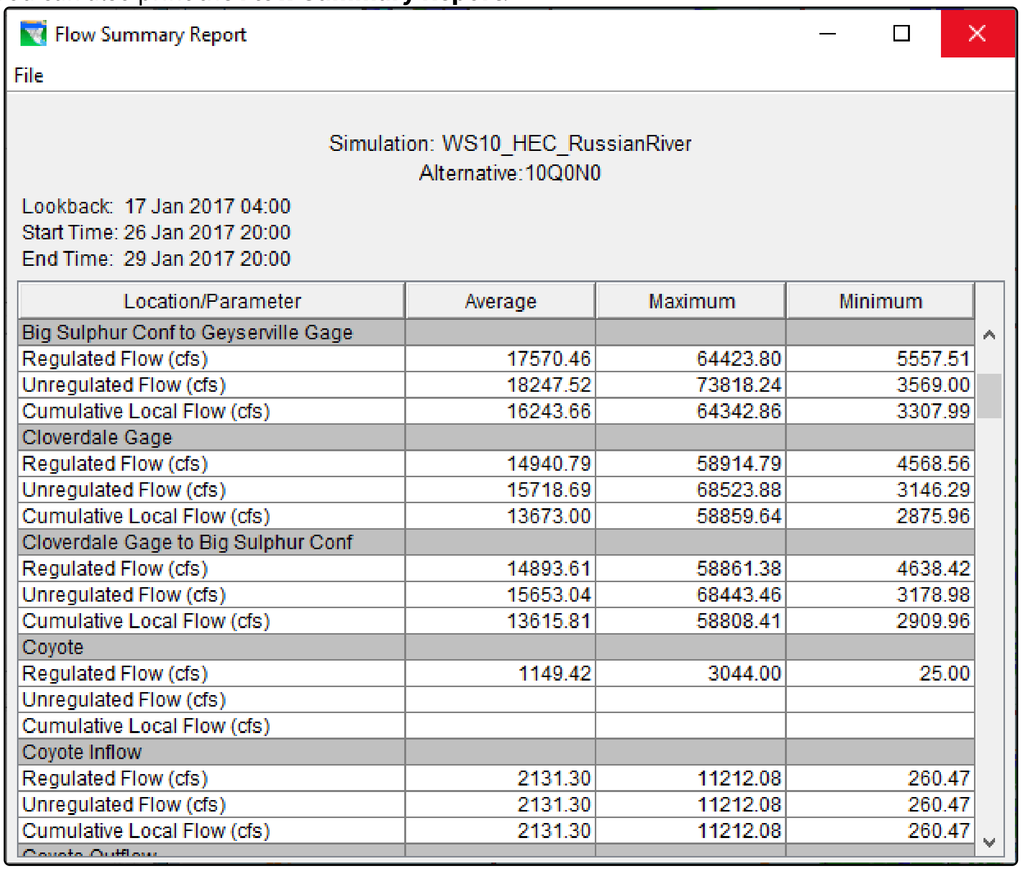
14.6.2 Flow Summary Report

The **Flow Summary Report** summarizes the flow at reservoirs and non-reservoir locations for the active forecast run. Results include regulated flow, unregulated flow, and cumulative local flow for each location, and are reported as the average, maximum, minimum values for each parameter.

To view the **Flow Summary Report**:

1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-ResSim model alternative (*RT-Normal*), from the **Forecast** tabs, click **Reports.**
2. Click **Flow Summary**, the **Flow Summary Report** (Figure 2) dialog will open. This report summarizes the flow at reservoirs and non-reservoir locations. Information includes regulated flow; unregulated flow; and cumulative local flow. For each parameter, the report displays the average, maximum, and minimum values. In addition, the report provides information on the time window for HEC-ResSim model alternative (key code *10Q0N0*) compute.

You can also print the **Flow Summary Report**.



Location/Parameter	Average	Maximum	Minimum
Big Sulphur Conf to Geyserville Gage			
Regulated Flow (cfs)	17570.46	64423.80	5557.51
Unregulated Flow (cfs)	18247.52	73818.24	3569.00
Cumulative Local Flow (cfs)	16243.66	64342.86	3307.99
Cloverdale Gage			
Regulated Flow (cfs)	14940.79	58914.79	4568.56
Unregulated Flow (cfs)	15718.69	68523.88	3146.29
Cumulative Local Flow (cfs)	13673.00	58859.64	2875.96
Cloverdale Gage to Big Sulphur Conf			
Regulated Flow (cfs)	14893.61	58861.38	4638.42
Unregulated Flow (cfs)	15653.04	68443.46	3178.98
Cumulative Local Flow (cfs)	13615.81	58808.41	2909.96
Coyote			
Regulated Flow (cfs)	1149.42	3044.00	25.00
Unregulated Flow (cfs)			
Cumulative Local Flow (cfs)			
Coyote Inflow			
Regulated Flow (cfs)	2131.30	11212.08	260.47
Unregulated Flow (cfs)	2131.30	11212.08	260.47
Cumulative Local Flow (cfs)	2131.30	11212.08	260.47

3. From the **File** menu, click **Close**, and the **Flow Summary Report** (Figure 2) dialog will close.

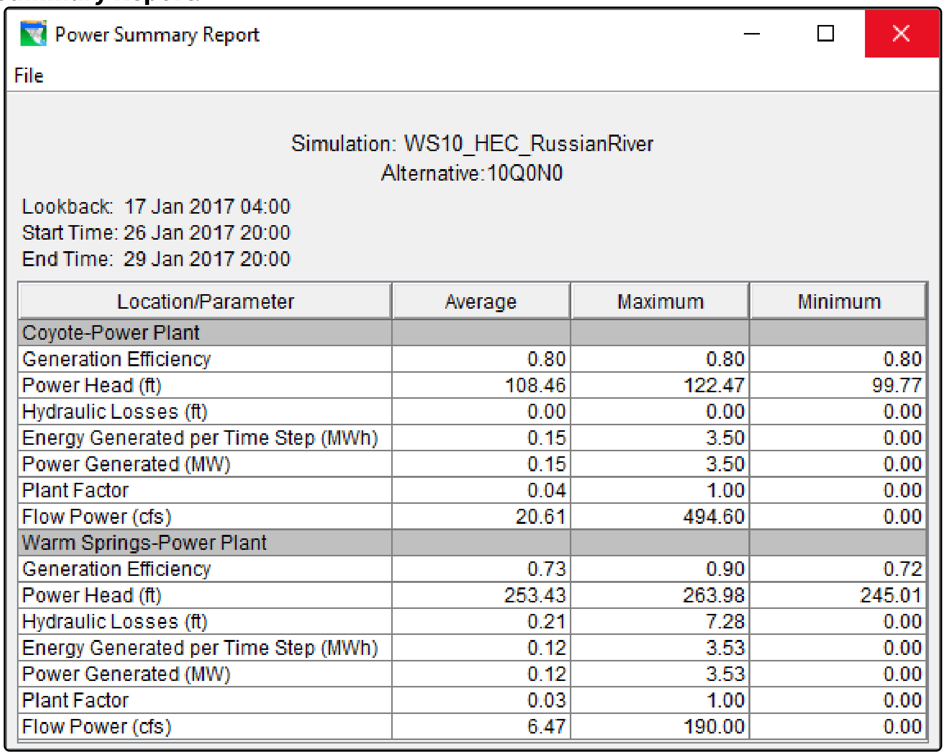
14.6.3 Power Summary Report

The **Power Summary Report** summarizes the hydropower information at a reservoir for the active forecast run. Results include generation efficiency, power head, hydraulic losses, energy generated per time step, power generated, plant factor, and flow power for each power plant, and are reported by the average, maximum, minimum values for each parameter. If the appropriate data (hydropower plant) has not been entered this report will be blank.

To view the **Power Summary Report**:

1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-ResSim model alternative (*RT-Normal*), right-click on the model alternative. From the shortcut menu, click **Power Summary**.
2. The **Power Summary Report** (Figure 3) dialog will open. This report summarizes the hydropower information for reservoirs in a watershed. Information includes regulated generation efficiency, power heard, hydraulic losses, energy generated (time step), power generated, plant factor, and flow power. For each parameter, the report displays the average, maximum, and minimum values. In addition, the report provides information on the time window for HEC-ResSim model alternative (key code *10Q0N0*) compute. You can also print the **Power**

Summary Report.



Location/Parameter	Average	Maximum	Minimum
Coyote-Power Plant			
Generation Efficiency	0.80	0.80	0.80
Power Head (ft)	108.46	122.47	99.77
Hydraulic Losses (ft)	0.00	0.00	0.00
Energy Generated per Time Step (MWh)	0.15	3.50	0.00
Power Generated (MW)	0.15	3.50	0.00
Plant Factor	0.04	1.00	0.00
Flow Power (cfs)	20.61	494.60	0.00
Warm Springs-Power Plant			
Generation Efficiency	0.73	0.90	0.72
Power Head (ft)	253.43	263.98	245.01
Hydraulic Losses (ft)	0.21	7.28	0.00
Energy Generated per Time Step (MWh)	0.12	3.53	0.00
Power Generated (MW)	0.12	3.53	0.00
Plant Factor	0.03	1.00	0.00
Flow Power (cfs)	6.47	190.00	0.00

3. From the **File** menu, click **Close**, and the **Power Summary Report** (Figure 3) dialog will close.

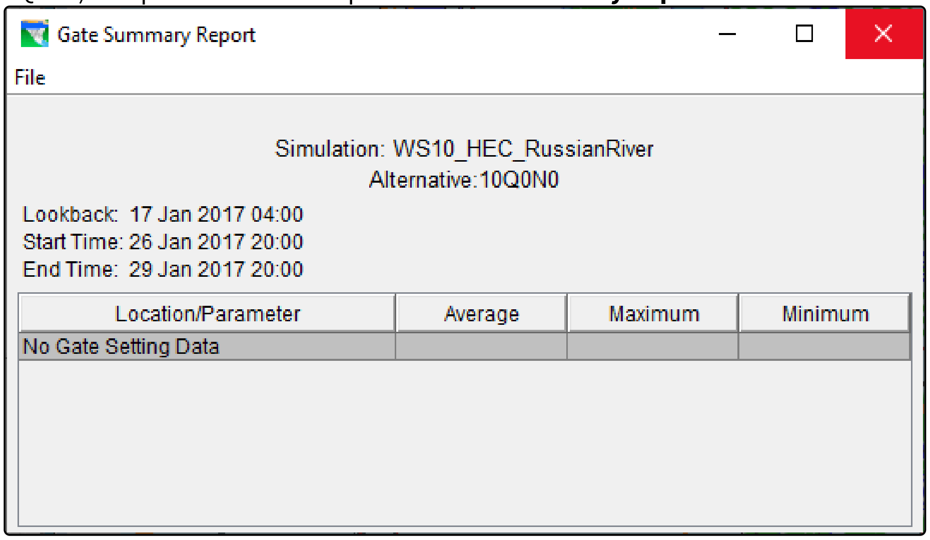
14.6.4 Gate Summary Report

The **Gate Summary Report** summarizes the gates defined for the reservoirs in the active forecast run. Reported is the average, maximum, minimum value for the gate opening at each reservoir where a gate has been defined. If the appropriate data (gates) has not been entered this report will be blank.

To view the **Gate Summary Report**:

1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-ResSim model alternative (*RT-Normal*), right-click on the model alternative. From the shortcut menu, click **Gates Summary**.
2. The **Gate Summary Report** (Figure 4) dialog will open. This report summarizes the gate information for reservoirs in a watershed. For each parameter, the report displays the average, maximum, and minimum values. In addition, the report provides information on the time window for the HEC-ResSim model alternative (key code

10Q0N0) compute. You can also print the **Gate Summary Report**.



3. From the **File** menu, click **Close**, and the **Gate Summary Report** (Figure 4) dialog will close.

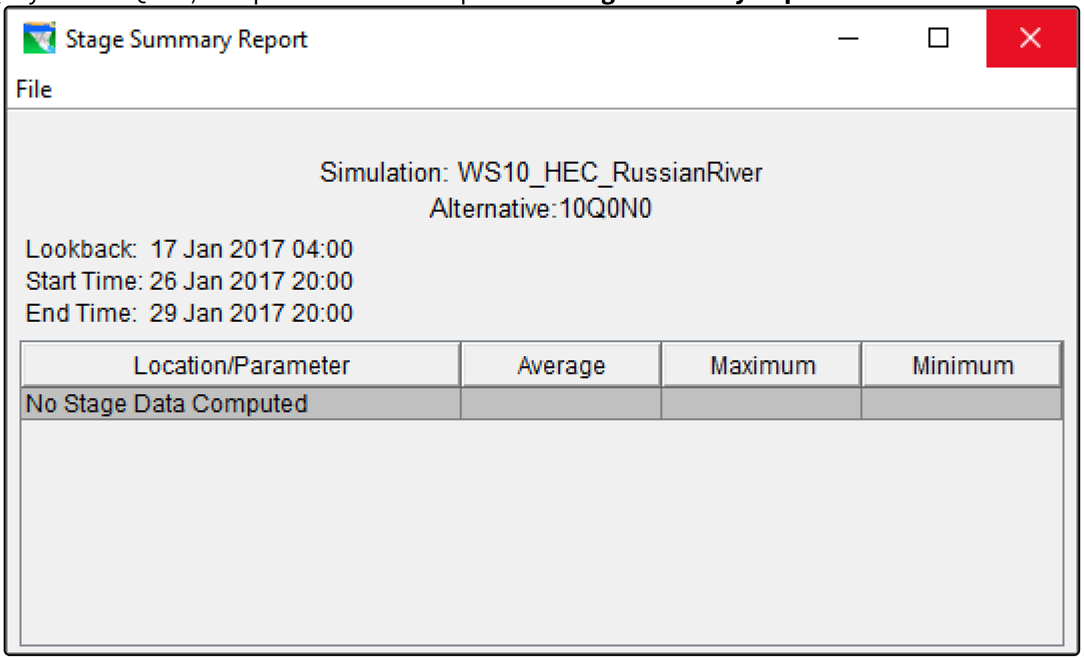
14.6.5 Stage Summary Report

The **Stage Summary Report** summarizes the stage at reservoirs and non-reservoir locations for the active forecast run. These results include the regulated stage, unregulated stage, and cumulative local stage for each location, and are reported by the average, maximum, minimum values for each parameter. If the appropriate data has not been entered (rating curves) this report will be blank.

To view the **Stage Summary Report**:

1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-ResSim model alternative (*RT-Normal*), right-click on the model alternative. From the shortcut menu, click **Stage Summary**.
2. The **Stage Summary Report** (Figure 5) dialog will open. This report summarizes the stage at reservoirs and non-reservoir locations in a watershed. Information includes regulated stage, unregulated stage, and cumulative local stage for each location. For each parameter, the report displays the average, maximum, and minimum values. In addition, the report provides information on the time window for the HEC-ResSim model alternative

(key code *10Q0N0*) compute. You can also print the **Stage Summary Report**.



3. From the **File** menu, click **Close**, and the **Stage Summary Report** (Figure 5) dialog will close.

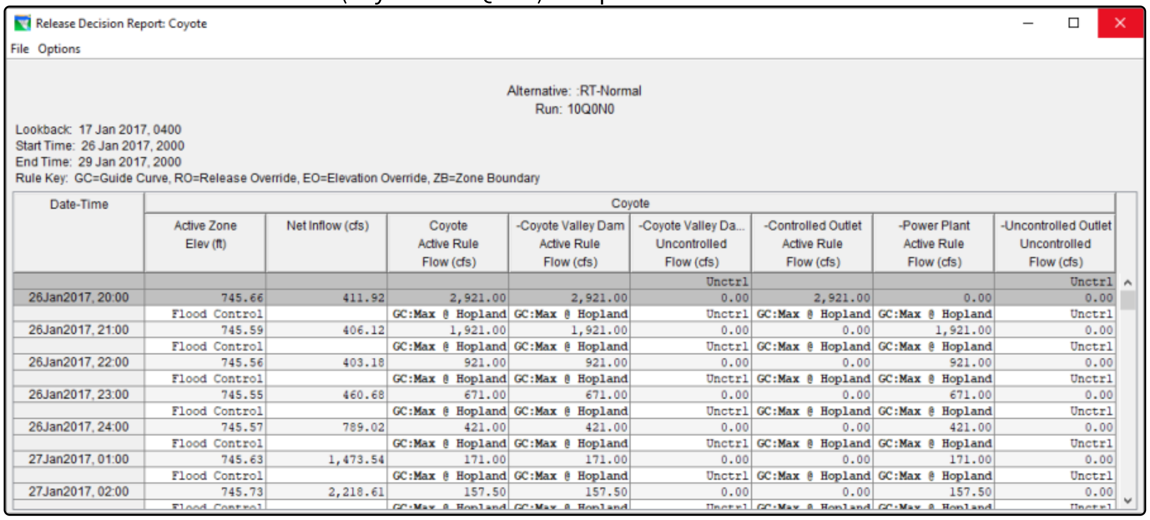
14.6.6 Release Decision Report

The **Release Decision Report** provides release decision information for reservoirs in a watershed. The release decision information is based on operational rules and physical constraints defined in an HEC-ResSim model alternative that was defined for a watershed.

To view the **Release Decision Report**:

1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-ResSim model alternative (*RT-Normal*), from the **Forecast** tabs, click **Reports**. You can also access this report from an HEC-ResSim reservoir model element. From the **Map Window**, locate an HEC-ResSim reservoir model element (*Coyote*), right-click on the model element, from the shortcut menu click **Release Decision Report**.
2. The **Release Decision Report** (Figure 6) dialog will open. This report summarizes the release decisions for a reservoir in a watershed. Information provided in this report is based on the operational rules and physical constraints defined in an HEC-ResSim model alternative. There is a legend for this report, so you can understand what the acronyms mean (*GC = Guide Curve*). In addition, the report provides information on the time window for

HEC-ResSim model alternative (key code *10Q0N0*) compute.



Release Decision Report: Coyote

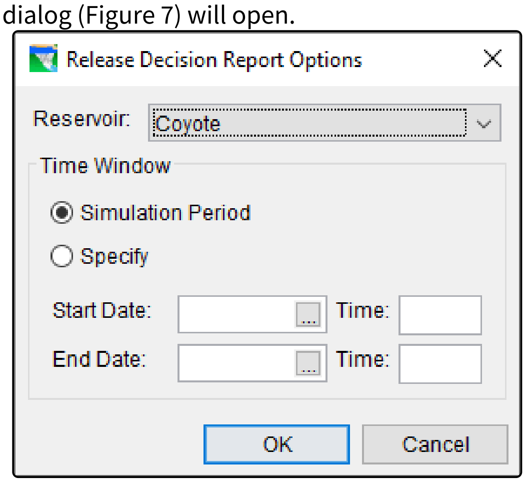
File Options

Alternative: :RT-Normal
Run: 10Q0N0

Lookback: 17 Jan 2017, 0400
Start Time: 26 Jan 2017, 2000
End Time: 29 Jan 2017, 2000
Rule Key: GC=Guide Curve, RO=Release Override, EO=Elevation Override, ZB=Zone Boundary

Date-Time	Coyote							
	Active Zone Elev (ft)	Net Inflow (cfs)	Coyote Active Rule Flow (cfs)	-Coyote Valley Dam Active Rule Flow (cfs)	-Coyote Valley Da... Uncontrolled Flow (cfs)	-Controlled Outlet Active Rule Flow (cfs)	-Power Plant Active Rule Flow (cfs)	-Uncontrolled Outlet Uncontrolled Flow (cfs)
					Unctrl			Unctrl
26Jan2017, 20:00	745.66	411.92	2,921.00	2,921.00	0.00	2,921.00	0.00	0.00
	Flood Control		GC:Max @ Hopland	GC:Max @ Hopland	Unctrl	GC:Max @ Hopland	GC:Max @ Hopland	Unctrl
26Jan2017, 21:00	745.59	406.12	1,921.00	1,921.00	0.00	0.00	1,921.00	0.00
	Flood Control		GC:Max @ Hopland	GC:Max @ Hopland	Unctrl	GC:Max @ Hopland	GC:Max @ Hopland	Unctrl
26Jan2017, 22:00	745.56	403.18	921.00	921.00	0.00	0.00	921.00	0.00
	Flood Control		GC:Max @ Hopland	GC:Max @ Hopland	Unctrl	GC:Max @ Hopland	GC:Max @ Hopland	Unctrl
26Jan2017, 23:00	745.55	460.68	671.00	671.00	0.00	0.00	671.00	0.00
	Flood Control		GC:Max @ Hopland	GC:Max @ Hopland	Unctrl	GC:Max @ Hopland	GC:Max @ Hopland	Unctrl
26Jan2017, 24:00	745.57	789.02	421.00	421.00	0.00	0.00	421.00	0.00
	Flood Control		GC:Max @ Hopland	GC:Max @ Hopland	Unctrl	GC:Max @ Hopland	GC:Max @ Hopland	Unctrl
27Jan2017, 01:00	745.63	1,473.54	171.00	171.00	0.00	0.00	171.00	0.00
	Flood Control		GC:Max @ Hopland	GC:Max @ Hopland	Unctrl	GC:Max @ Hopland	GC:Max @ Hopland	Unctrl
27Jan2017, 02:00	745.73	2,218.61	157.50	157.50	0.00	0.00	157.50	0.00

3. This report is for a single reservoir (*Coyote*) in the watershed for a certain time window. You can change the time window for the information being displayed, or switch to another reservoir in the watershed. From the **Release Decision Report** (Figure 6), from the **Options** menu, click **Edit Options**, the **Release Decision Report Options** dialog (Figure 7) will open.

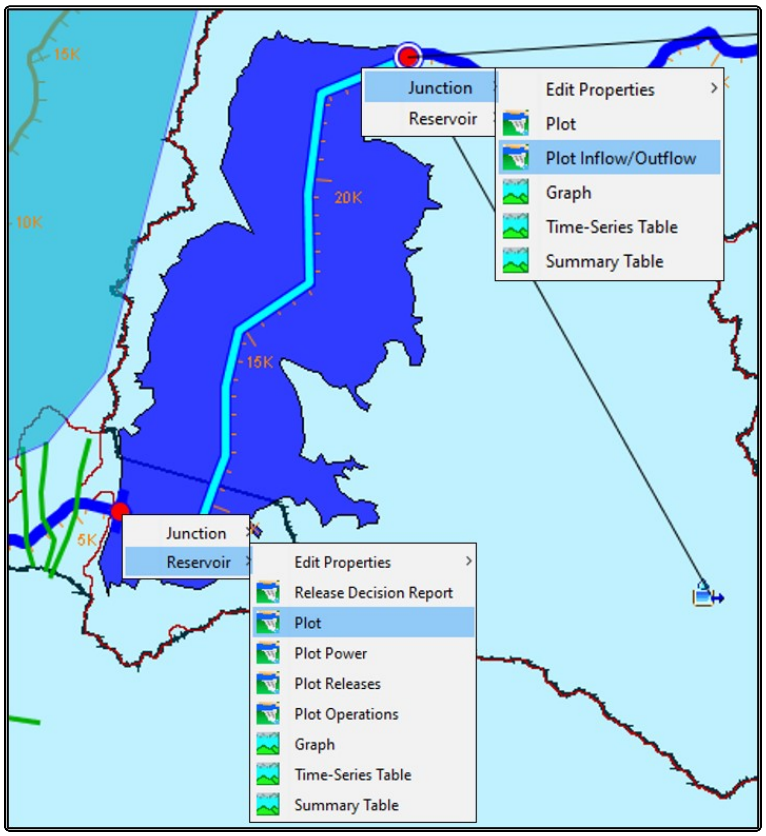


4. From the **Reservoir** list, select a different reservoir, click **OK**, the **Release Decision Report Options** dialog (Figure 7) will close. The **Release Decision Report** will now display results for the selected reservoir. You can also specify a different time window from the **Release Decision Report Options** dialog. From the **Time Window** box, select **Specify,** now you can enter different start and end dates.
5. You can also print the **Release Decision Report**. From the **File** menu, click **Close**, and the **Release Decision Report** (Figure 6) dialog will close.

14.6.7 Graphical Results

From the Modeling module, to review HEC-ResSim graphical results for individual model elements (e.g., reservoirs, reaches, junctions, etc.) from the **Forecast Runs Details** section, next to an HEC-ResSim model alternative, click ⊞ . The HEC-ResSim model elements will display. Right-click on an HEC-ResSim model element, from the shortcut menu, depending on the HEC-ResSim model element, there are several graphical results available for review. Another way to review HEC-ResSim model element results from the **Forecast Runs Detail** section, for an HEC-ResSim model alternative, select an individual model element. From the **Forecast** tabs, click **Reports**. From the **Reports** tab, there are several graphical results available for the selected HEC-ResSim model element. In addition, when some of the individual model elements are selected, in the **Map Window**, on the schematic, the selected individual model element is highlighted in yellow.

Another way to review HEC-ResSim graphical results from the Modeling module, is from the **Map Window**. From the **Map Window**, click (**Select Tool**), from the schematic, right-click on an HEC-ResSim model element. From the shortcut menu (Figure 8), you can review HEC-ResSim graphical results for the selected model element. Since the shortcut menu (Figure 8), will display any available model alternative results for the forecast run, HEC-ResSim results are identified with the HEC-ResSim icon ( ).



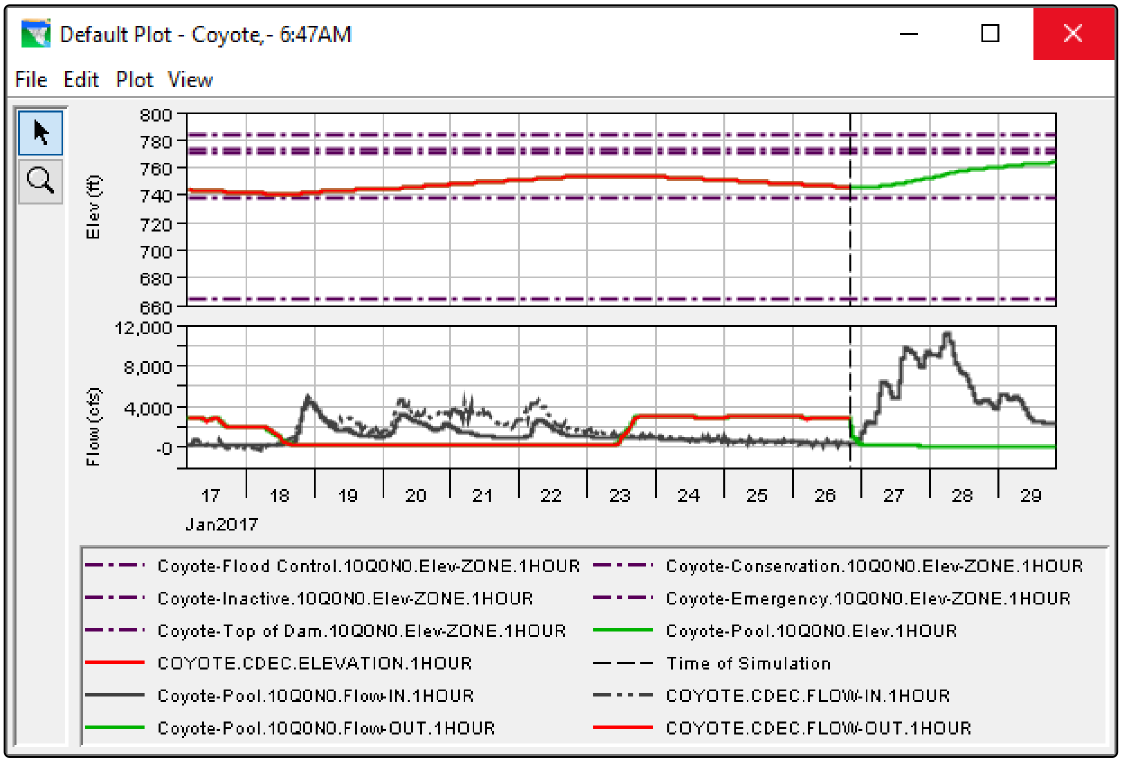
75 Figure 8 Map Window - HEC-ResSim Model Elements - Reservoir and Junction

14.6.7.1 Reservoirs

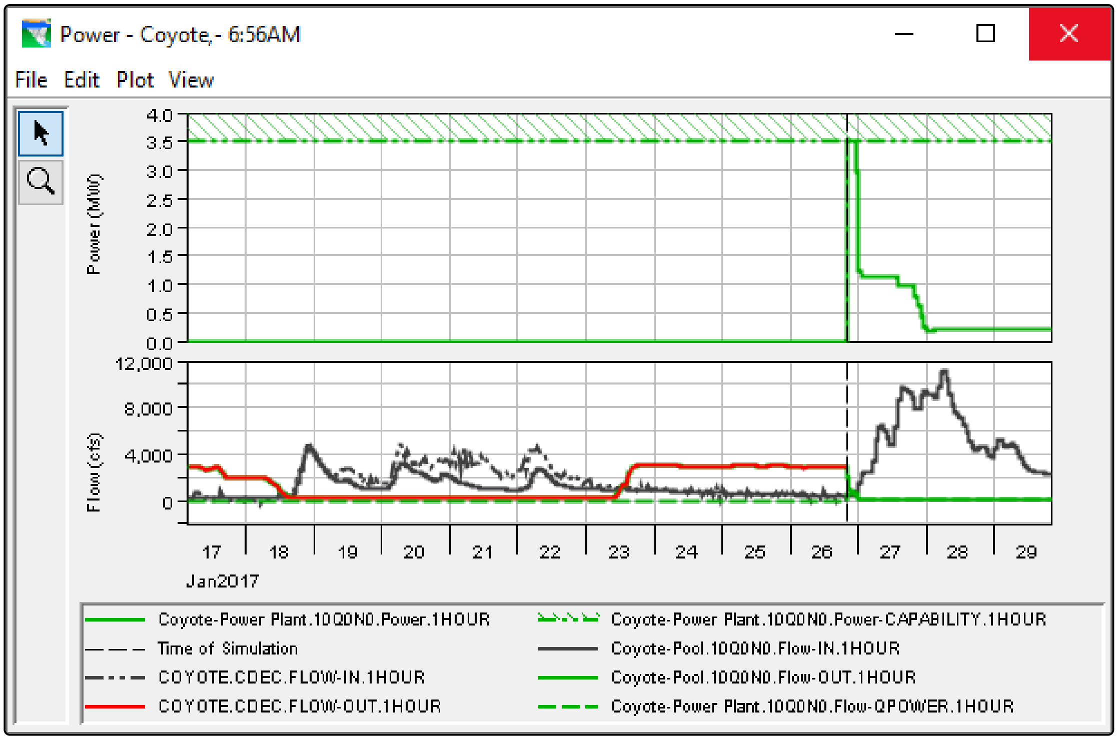
To view the graphical results from an HEC-ResSim reservoir:

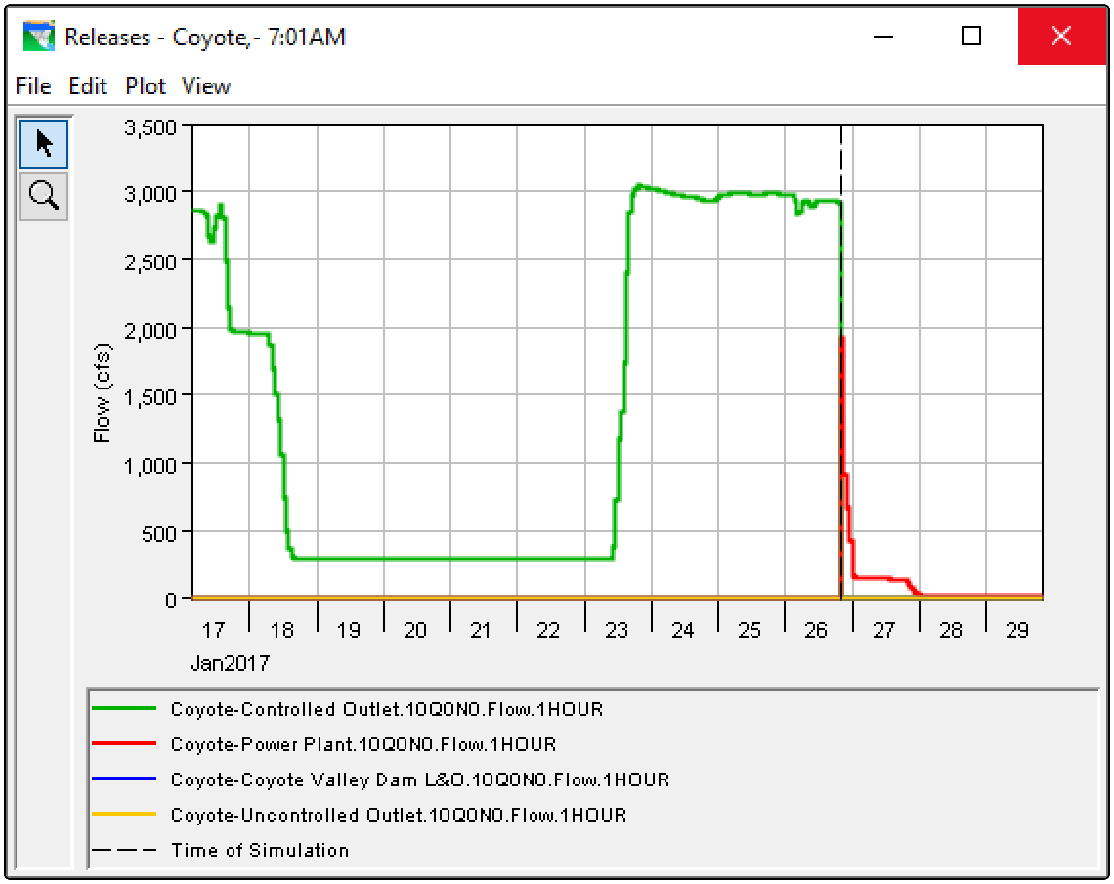
1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-ResSim model alternative (*RT-Normal*). Next to the selected HEC-ResSim model alternative, click . The HEC-ResSim model elements will display.

2. Select an HEC-ResSim reservoir model element (), from the **Forecast** tabs, click **Reports**. From the **Reports** tab, there are several graphical results available for a HEC-ResSim reservoir model element.
3. From the **Reports** tab, to view the default plot for an HEC-ResSim reservoir, click **Plot**. The **Default Plot** dialog will open (Figure 9). This plot displays various information about a reservoir (operations) for the active forecast run.



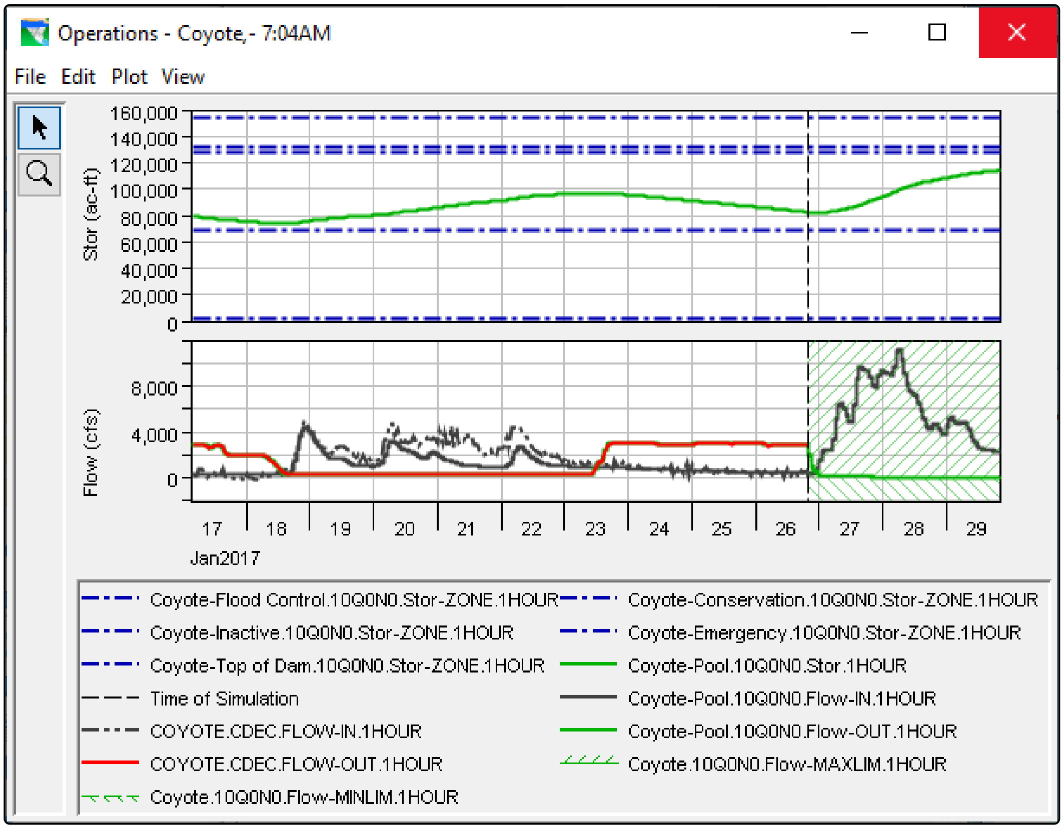
4. From the **Reports** tab, to view the power plot for an HEC-ResSim reservoir, click **Plot Power**. The **Power** plot dialog will open (Figure 10). This plot displays the hydropower information at a reservoir for the active forecast run.
5. From the **Reports** tab, to view the releases plot for an HEC-ResSim reservoir, click **Plot Releases**. The **Releases** plot dialog will open (Figure 11). This plot displays the release information at a reservoir for the active forecast run.


Power - Coyote,- 6:56AM
File Edit Plot View
Power (MW)
4.0
3.5
3.0
2.5
2.0
1.5
1.0
0.5
0.0
Flow (cfs)
12,000
8,000
4,000
0
17
18
19
20
21
22
23
24
25
26
27
28
29
Jan2017
Coyote-Power Plant.10Q0N0.Power.1HOUR
Coyote-Power Plant.10Q0N0.Power-CAPABILITY.1HOUR
Time of Simulation
Coyote-Pool.10Q0N0.Flow-IN.1HOUR
COYOTE.CDEC.FLOW-IN.1HOUR
Coyote-Pool.10Q0N0.Flow-OUT.1HOUR
COYOTE.CDEC.FLOW-OUT.1HOUR
Coyote-Power Plant.10Q0N0.Flow-QPOWER.1HOUR



6. From the **Reports** tab, to view the operations plot for an HEC-ResSim reservoir, click **Plot Operations**. The **Operations** plot dialog will open (Figure 12). This plot displays the operations information at a reservoir for the

active forecast run.



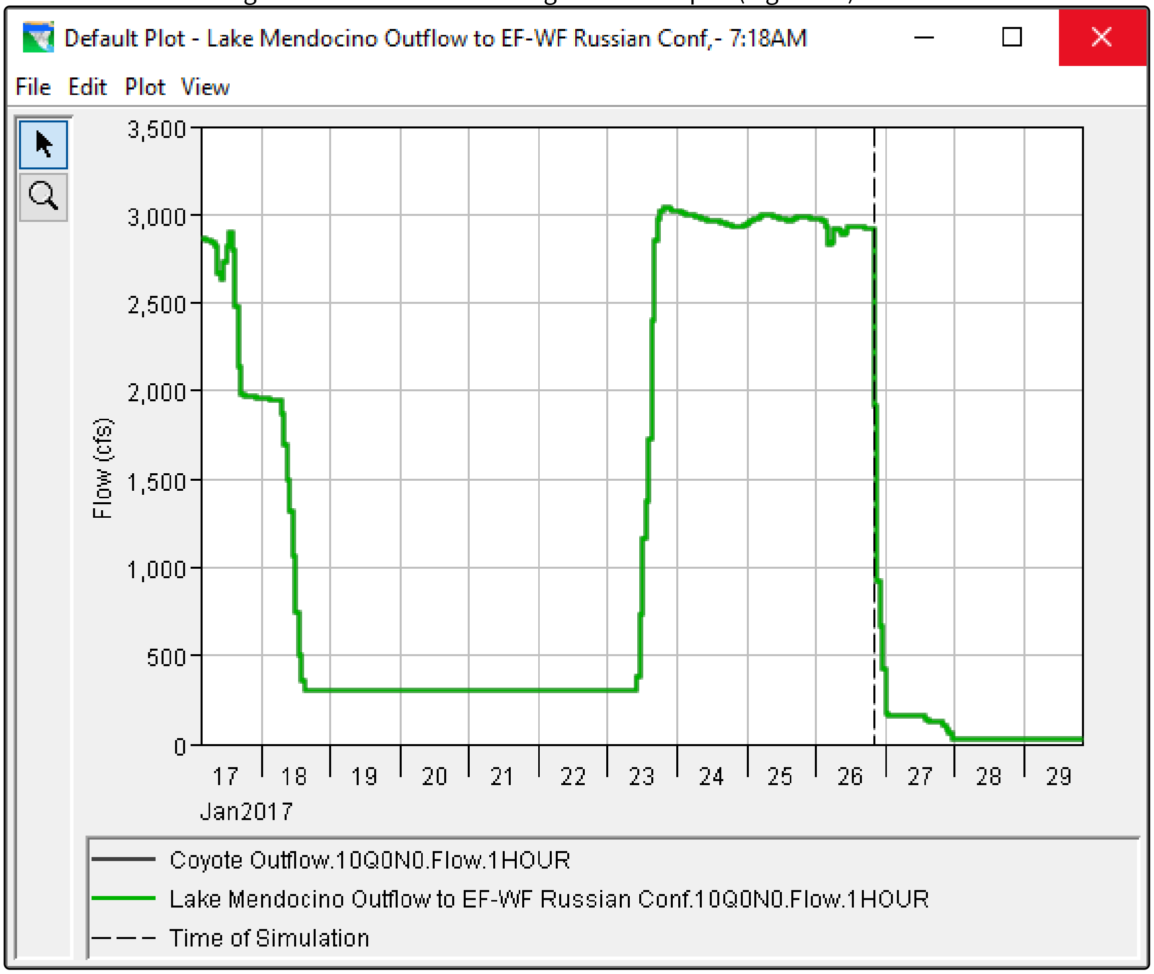
7. You can print, save as a graphics file, and view tabular data of the plot from the **File** menu (Figure 12). See Working with Plots for details on how you can manipulate plots available in HEC-RTS.
8. To close a plot dialog, from the **File** menu (Figure 12). click **Close**. The active plot dialog will close.

14.6.7.2 Routing Reaches

To view the graphical results from a HEC-ResSim routing reach:

1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-ResSim model alternative (*RT-Normal*). Next to the selected HEC-ResSim model alternative, click . The HEC-ResSim model elements will display.
2. Select an HEC-ResSim routing reach model element ( ), from the **Forecast** tabs, click **Reports**. From the **Reports** tab, click **Plot**.

3. The **Default Plot** dialog for an HEC-ResSim routing reach will open (Figure 13).



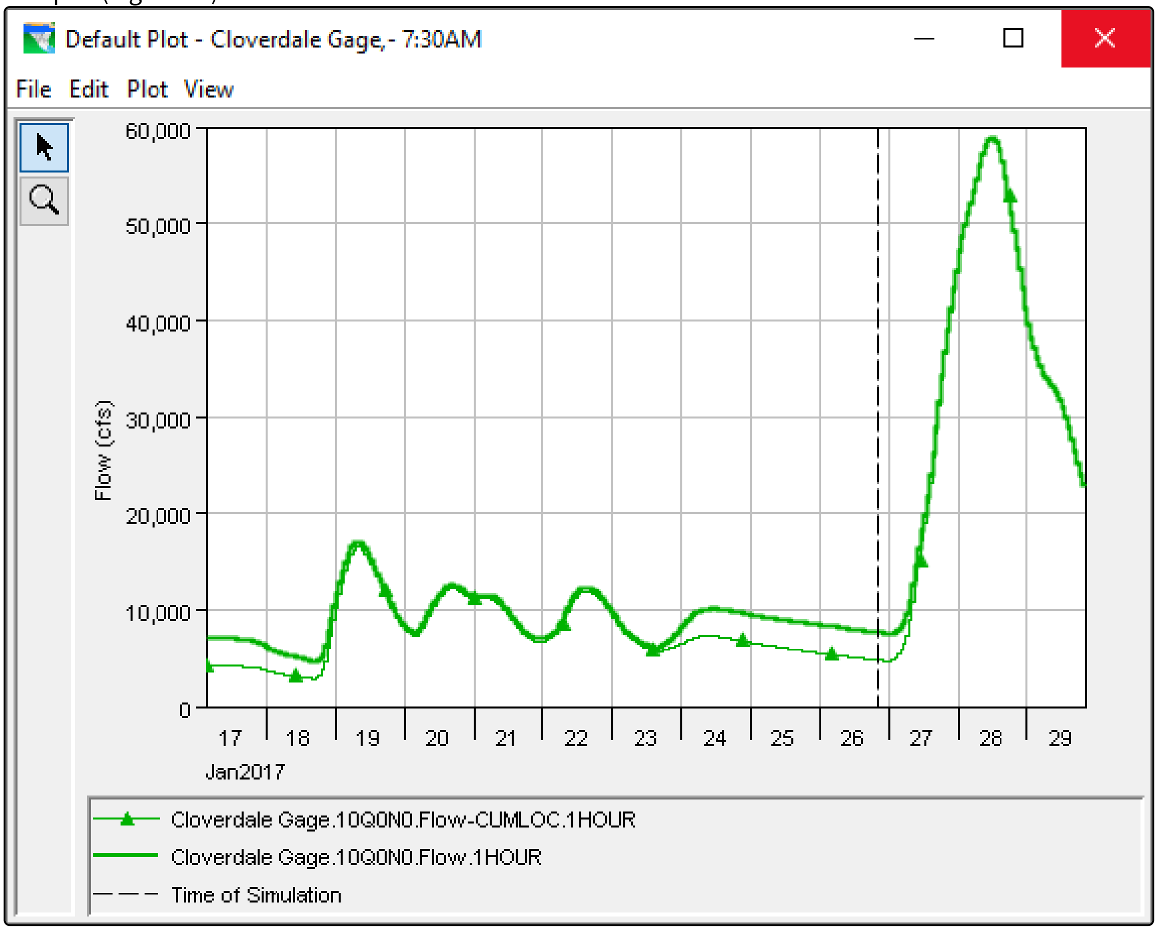
4. You can print, save as a graphics file, and view tabular data of the plot from the **File** menu (Figure 13). See Working with Plots for details on how you can manipulate plots available in HEC-RTS.
5. To close a plot dialog, from the **File** menu (Figure 13). click **Close**. The active plot dialog will close.

14.6.7.3 Junctions

To view the graphical results from an HEC-ResSim junction:

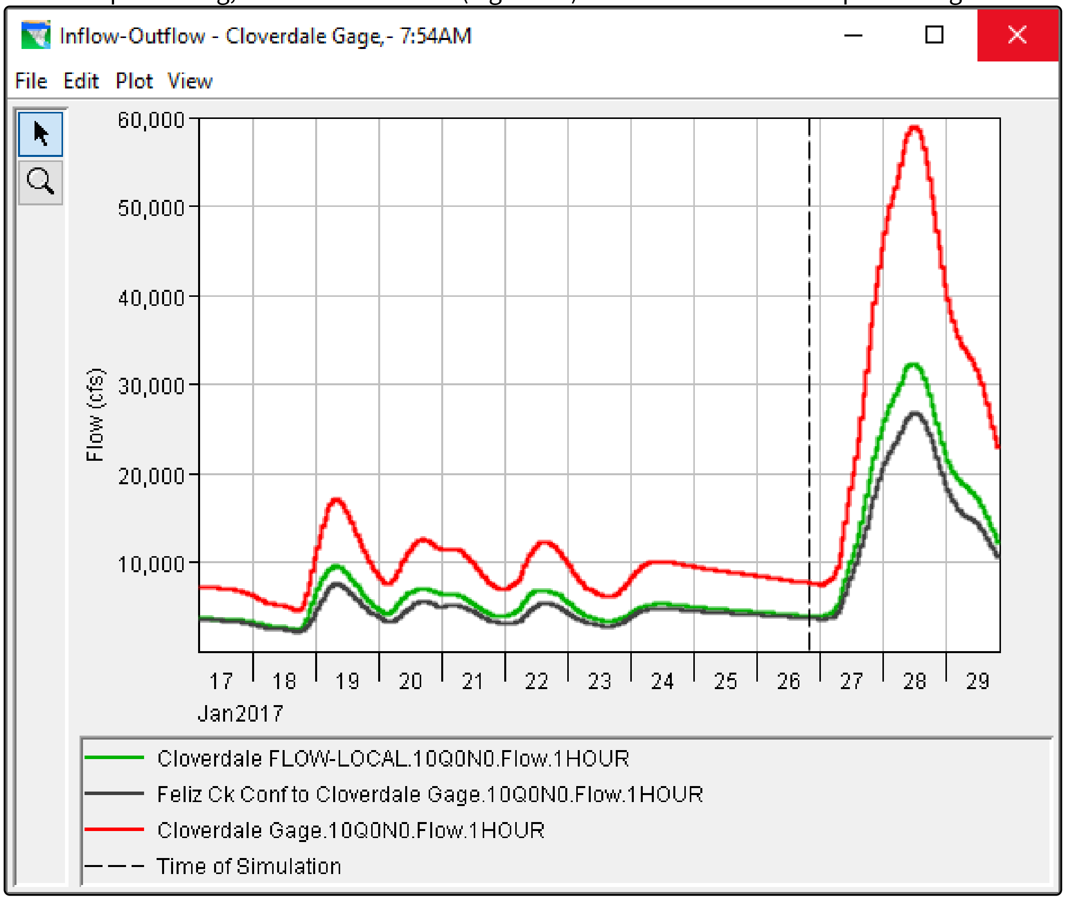
1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-ResSim model alternative (*RT-Normal*). Next to the selected HEC-ResSim model alternative, click . The HEC-ResSim model elements will display.
2. Select an HEC-ResSim junction model element ( ), from the **Forecast** tabs, click **Reports**. From the **Reports** tab, there are two graphical results available for an HEC-ResSim junction model element.

3. From the **Reports** tab, to view the default plot for an HEC-ResSim junction, click **Plot**. The **Default Plot** dialog will open (Figure 14).



4. From the **Reports** tab, to view the inflow-outflow plot for an HEC-ResSim junction, click **Plot Inflow/Outflow**. The **Inflow-Outflow Plot** dialog will open (Figure 15). This plot provides a graphical representation of flow into and the flow leaving the selected HEC-ResSim junction (*Cloverdale Gage*).
5. You can print, save as a graphics file, and view tabular data of the plot from the **File** menu (Figure 15). See Working with Plots for details on how you can manipulate plots available in HEC-RTS.

6. To close a plot dialog, from the **File** menu (Figure 15). click **Close**. The active plot dialog will close.

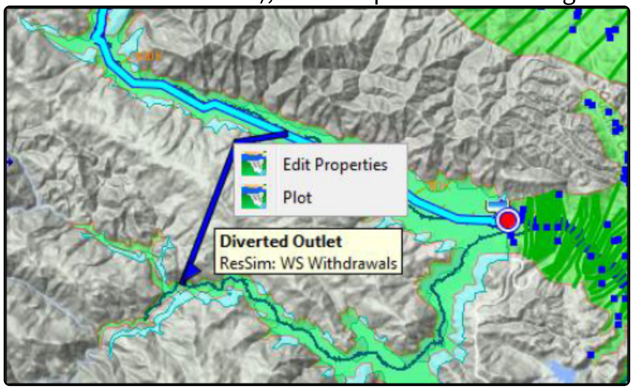


14.6.7.4 Diverted Outlets

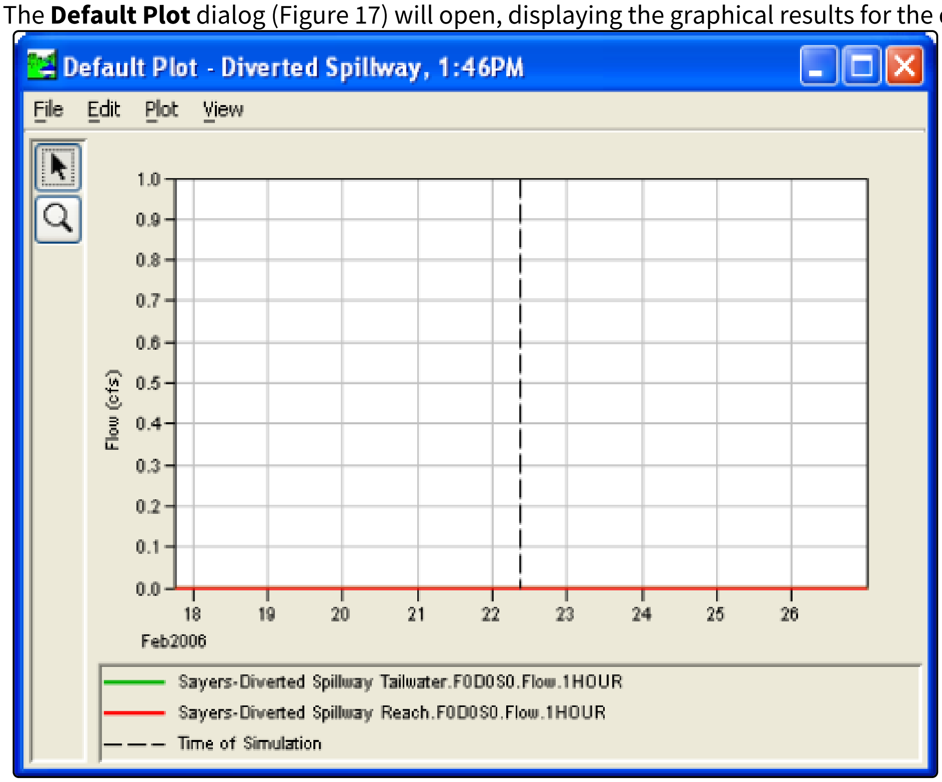
To view the graphical results from an HEC-ResSim diverted outlet:

1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. You may need to zoom in on the **Map window**, select the **Zoom Tool** and zoom-in on an area where there is an HEC-ResSim diverted outlet model element (line leaving a reservoir

pool with an arrowhead); an example is shown in Figure 16.



2. From the **Map Window** (Figure 16), click  (**Select Tool**), from the schematic, right-click on an HEC-ResSim diverted outlet model element. From the shortcut menu, click **Plot**.
3. The **Default Plot** dialog (Figure 17) will open, displaying the graphical results for the diverted outlet.



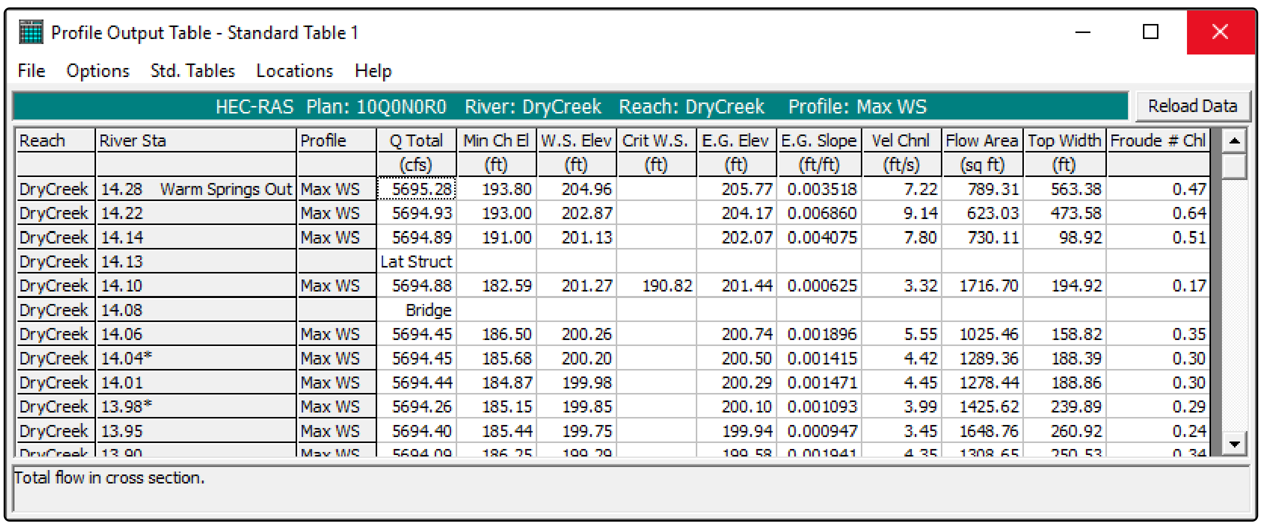
4. You can print, save as a graphics file, and view tabular data of the plot from the **File** menu (Figure 17). See Working with Plots for details on how you can manipulate plots available in HEC-RTS.
5. To close a plot dialog, from the **File** menu (Figure 17). click **Close**. The active plot dialog will close.

14.7 HEC-RAS Results

HEC-RAS may be used in HEC-RTS to simulate the behavior of channels in a watershed, given flow rates predicted with HEC-HMS. There are several reports available from HEC-RAS for the HEC-RAS model alternative, and for HEC-RAS model elements. From an HEC-RAS model alternative you have a series of reports, that provide you with results for key variables; distribution of flow (left and right overbank, main channel); steady flow results; unsteady flow results; rating curves; and, much more. There are two HEC-RAS model elements (cross sections, routing reach), you can review plots of cross sections; and, plots from routing reaches (profiles, XYZ).

14.7.1 Standard Table 1

The **Standard Table 1** is the default summary table (Figure 1). This table provides you a summary of key HEC-RAS output variables. To view the **Standard Table 1**:



Reach	River Sta	Profile	Q Total	Min Ch El	W.S. Elev	Crit W.S.	E.G. Elev	E.G. Slope	Vel Chnl	Flow Area	Top Width	Froude # Chl
			(cfs)	(ft)	(ft)	(ft)	(ft)	(ft/ft)	(ft/s)	(sq ft)	(ft)	
DryCreek	14.28 Warm Springs Out	Max WS	5695.28	193.80	204.96		205.77	0.003518	7.22	789.31	563.38	0.47
DryCreek	14.22	Max WS	5694.93	193.00	202.87		204.17	0.006860	9.14	623.03	473.58	0.64
DryCreek	14.14	Max WS	5694.89	191.00	201.13		202.07	0.004075	7.80	730.11	98.92	0.51
DryCreek	14.13		Lat Struct									
DryCreek	14.10	Max WS	5694.88	182.59	201.27	190.82	201.44	0.000625	3.32	1716.70	194.92	0.17
DryCreek	14.08		Bridge									
DryCreek	14.06	Max WS	5694.45	186.50	200.26		200.74	0.001896	5.55	1025.46	158.82	0.35
DryCreek	14.04*	Max WS	5694.45	185.68	200.20		200.50	0.001415	4.42	1289.36	188.39	0.30
DryCreek	14.01	Max WS	5694.44	184.87	199.98		200.29	0.001471	4.45	1278.44	188.86	0.30
DryCreek	13.98*	Max WS	5694.26	185.15	199.85		200.10	0.001093	3.99	1425.62	239.89	0.29
DryCreek	13.95	Max WS	5694.40	185.44	199.75		199.94	0.000947	3.45	1648.76	260.92	0.24
DryCreek	13.90	Max WS	5694.09	186.25	199.79		199.58	0.001941	4.35	1308.65	250.53	0.34

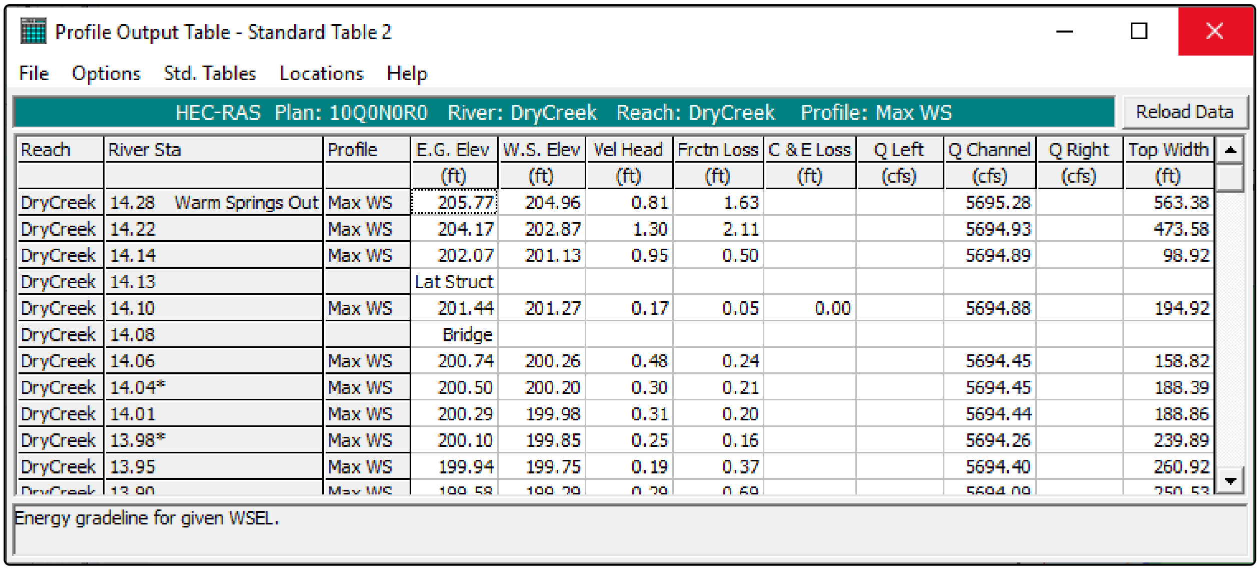
76 Figure 1 HEC-RAS - Profile Output Table - Standard Table 1 Dialog

1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-RAS model alternative (*Russian River Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **Summary Output Table**, the **Profile Output Table - Standard Table 1** dialog (Figure 1) opens. This report displays several hydraulic variables for stream stations. You can change certain parameters that are displayed, from the **Options** menu (Figure 1), you can choose which profiles and reaches to display, you can add a column to the report that includes the profile name, and you can choose which system units you wish to view the report in.
3. From the **File** menu, click **Exit**, and the **Profile Output Table - Standard Table 1** dialog (Figure 1) closes.

14.7.2 Standard Table 2

The **Standard Table 2** (Figure 2) is the second of the standard summary tables. This table provides information on the distribution of flow between the left overbank, main channel, and right overbank. This table also shows the friction losses, as well as contraction and expansion losses that occurred between each section. Energy losses displayed at a cross section are for the losses that occurred between that section and the next section downstream.

To view the **Standard Table 2**:


Profile Output Table - Standard Table 2

File Options Std. Tables Locations Help

HEC-RAS Plan: 10Q0N0R0 River: DryCreek Reach: DryCreek Profile: Max WS

Reload Data

Reach	River Sta	Profile	E.G. Elev (ft)	W.S. Elev (ft)	Vel Head (ft)	Frctn Loss (ft)	C & E Loss (ft)	Q Left (cfs)	Q Channel (cfs)	Q Right (cfs)	Top Width (ft)
DryCreek	14.28 Warm Springs Out	Max WS	205.77	204.96	0.81	1.63			5695.28		563.38
DryCreek	14.22	Max WS	204.17	202.87	1.30	2.11			5694.93		473.58
DryCreek	14.14	Max WS	202.07	201.13	0.95	0.50			5694.89		98.92
DryCreek	14.13		Lat Struct								
DryCreek	14.10	Max WS	201.44	201.27	0.17	0.05	0.00		5694.88		194.92
DryCreek	14.08		Bridge								
DryCreek	14.06	Max WS	200.74	200.26	0.48	0.24			5694.45		158.82
DryCreek	14.04*	Max WS	200.50	200.20	0.30	0.21			5694.45		188.39
DryCreek	14.01	Max WS	200.29	199.98	0.31	0.20			5694.44		188.86
DryCreek	13.98*	Max WS	200.10	199.85	0.25	0.16			5694.26		239.89
DryCreek	13.95	Max WS	199.94	199.75	0.19	0.37			5694.40		260.92
DryCreek	13.90	Max WS	199.58	199.29	0.29	0.69			5694.09		250.53

Energy gradeline for given WSEL.

77 Figure 2 HEC-RAS - Profile Output Table - Standard Table 2 Dialog

1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-RAS model alternative (*Russian River Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **Summary Output Table**, the **Profile Output Table - Standard Table 1** dialog (Figure 1) opens. From the **Std. Tables** menu, click **Standard Table 2**, the **Profile Output Table** - **Standard Table 2** dialog (Figure 2) will open. The report displays several hydraulic variables for stream stations. You can change certain parameters that are displayed, from the **Options** menu (Figure 2), you can choose which profiles and reaches to display, you can add a column to the report that includes the profile name, and you can choose which system units you wish to view the report in.
3. From the **File** menu, click **Exit**, and the **Profile Output Table - Standard Table 2** dialog (Figure 2) closes.

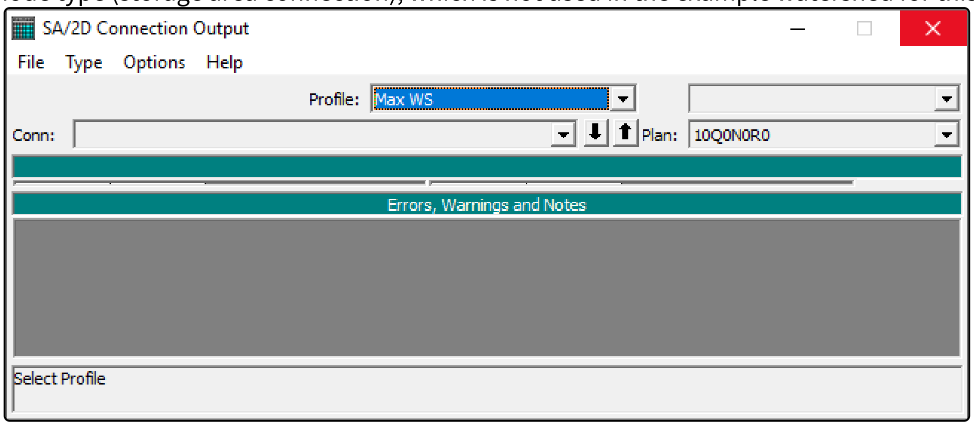
14.7.3 Detailed Output Table

Detailed output tables display hydraulic information at a single location for a single profile, for an HEC-RAS node type (i.e., cross section, culverts, bridges, etc.). For HEC-RTS version 3.1.1, the default node type is **SA/2D Connections**. For the example, in this User's Manual, for the HEC-RAS model alternative, there are no **SA/2D Connections** nodes. The user will need to select an appropriate node type to display information in a detailed output table.

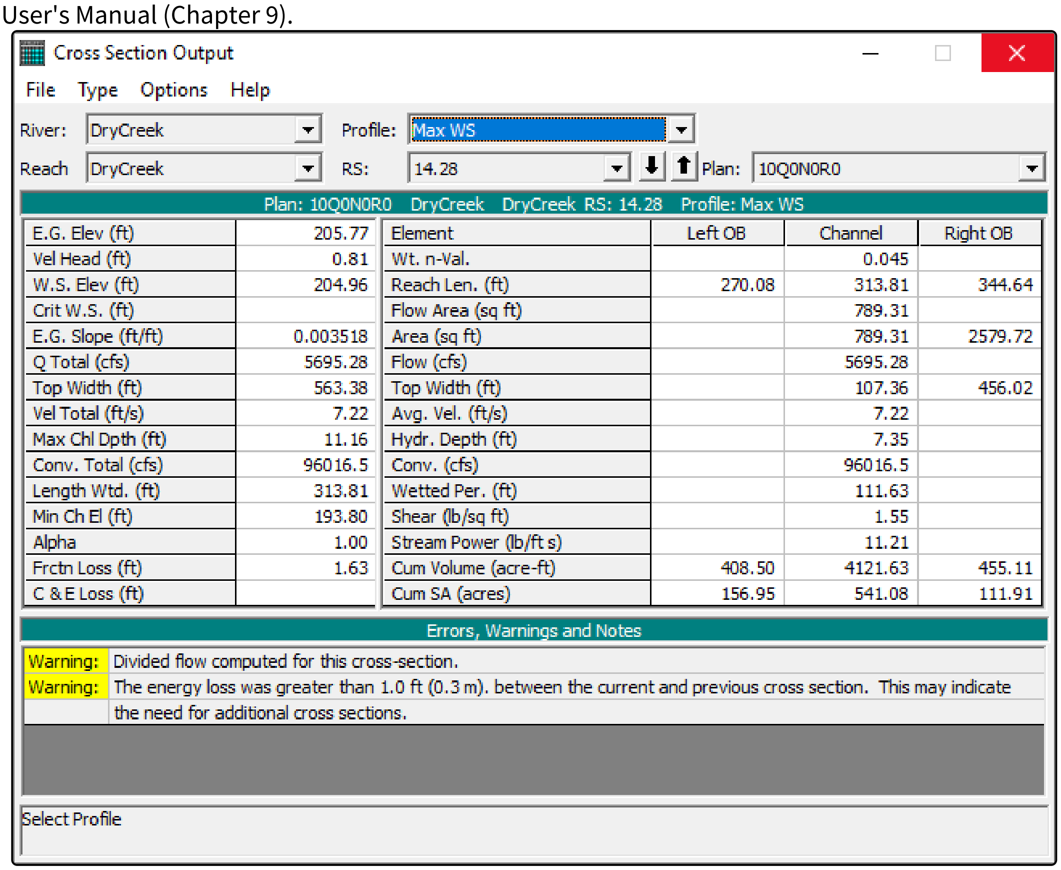
To view a **Detailed Output Table**:

1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-RAS model alternative (*Russian River Real-Time*), from the **Forecast** tabs, click **Reports**.

2. Click **Detailed Output Table**, the **SA/2D Connection Output** dialog (Figure 3) opens. This is the HEC-RAS default node type (storage area connection), which is not used in the example watershed for this User's Manual.



3. From the **Type** menu (Figure 3), click **Cross Sections**, the **Cross Section Output** dialog (Figure 4) will open. The report displays detailed hydraulic information about a single cross section (*14.28*) in the HEC-RAS model alternative. You can view information for other cross sections in the HEC-RAS model alternative, by manipulating the **River**, **Reach**, **Profile**, and **RS** lists (Figure 4). For further details on this HEC-RAS report, refer to the HEC-RAS User's Manual (Chapter 9).

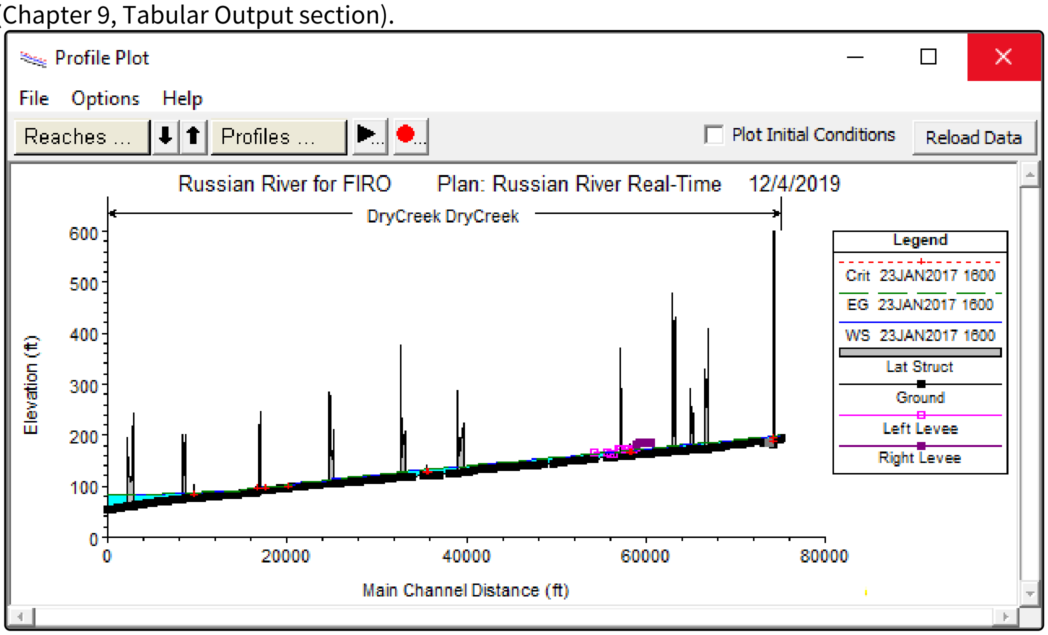


4. There are several other node types available in HEC-RAS - culvert, bridge, multiple opening, inline structure, lateral structure, storage area, storage area connection, pump stations, and flow distribution in cross sections. For further details, refer to the HEC-RAS User's Manual (Chapter 9).
5. From the **File** menu, click **Exit**, and the **Cross Section Output** dialog (Figure 4) closes.

14.7.4 Profile Plot

The result of a steady flow HEC-RAS model alternative is the calculation of water surface profiles. To view the water surface profile plot:

1. From the Modeling module, to view this report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-RAS model alternative (*Russian River Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **Profile Plot**, the **Profile Plot** dialog (Figure 5) opens. The **Profile Plot** displays the water surface profile for the first reach (routing) in the river system. To view other water surface profiles, you will need to manipulate the reaches and profiles. This manipulation is done by either clicking on **Reaches** or **Profiles** (Figure 5) and selecting what you want changed; change reaches by manipulating the up and down arrow to the right of the **Reaches** button; view profiles by using the animation controls; and, by changing various parameters that are available from the **Options** menu (Figure 5). For further details on this HEC-RAS report, refer to the HEC-RAS User's Manual (Chapter 9, Tabular Output section).

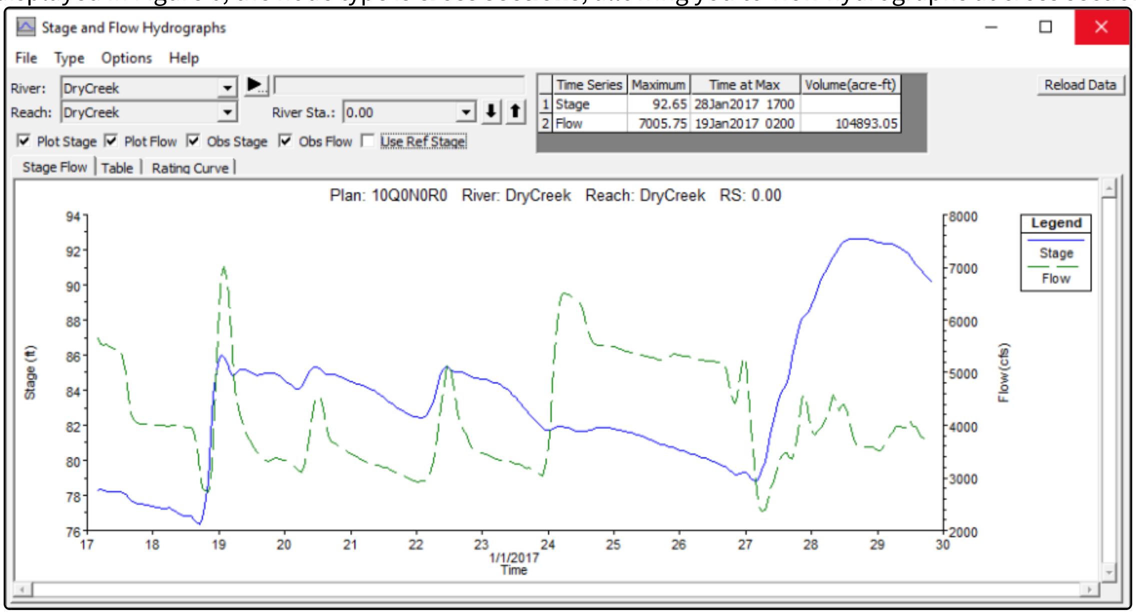


3. From the **File** menu, click **Exit**, and the **Profile Plot** dialog (Figure 5) closes.

14.7.5 Stage & Flow Hydrograph

During an HEC-RAS unsteady flow compute, stage and flow hydrographs are generated. To view the stage and flow hydrographs for a selected HEC-RAS model:

1. From the Modeling module, to view stage and flow hydrographs, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-RAS model alternative (*Russian River Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **Stage & Flow Hydrograph**, the **Stage and Flow Hydrographs** dialog (Figure 6) opens. You can plot stage, plot flow, plot observed stage, plot observed flow, and plot stage using a reference stage. By default, as displayed in Figure 6, the node type is **cross sections**, allowing you to view hydrographs at cross sections only.



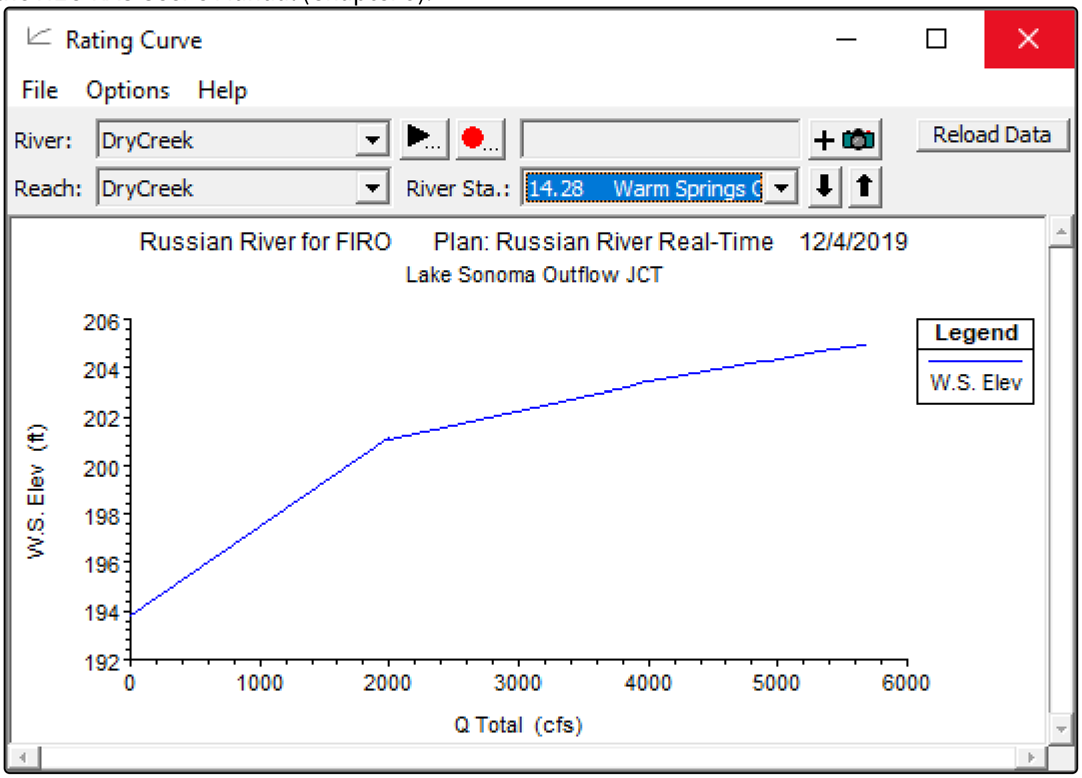
3. There are several other node types available in HEC-RAS - bridges/culverts, inline structures, lateral structures, storage areas, storage area connections, pump stations, ground water interactions, and storage area connections flow are with BC lines. You can view information by manipulating the **River**, **Reach**, and **River Sta.** lists (Figure 6). For further details on this HEC-RAS report, refer to the HEC-RAS User's Manual (Chapter 9).
4. Additional, from the **Stage and Flow Hydrographs** dialog (Figure 6), besides viewing data in a plot, from the three available tabs you can also view data in a tabular form (**Table**), and plot a rating curve of the selected event (**Rating Curve**). For further details on this HEC-RAS report, refer to the HEC-RAS User's Manual (Chapter 9).
5. From the **File** menu, click **Exit**, and the **Stage and Flow Hydrographs** dialog (Figure 6) closes.

14.7.6 Rating Curve Plots

During an HEC-RAS compute, if information for rating curves has been defined, rating curves can be viewed. To view rating curves for a selected HEC-RAS model alternative:

1. From the Modeling module, to view rating curves, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-RAS model alternative (*Russian River Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **Rating Curve**, the **Rating Curve** dialog (Figure 7) opens. The **Rating Curve** plot displays the water surface elevation versus flow rate for the water surface profiles that were computed. To view other rating curves, manipulate the **River**, **Reach**, and **River Sta.** lists (Figure 7). For further details on this HEC-RAS report, refer to

the HEC-RAS User's Manual (Chapter 9).

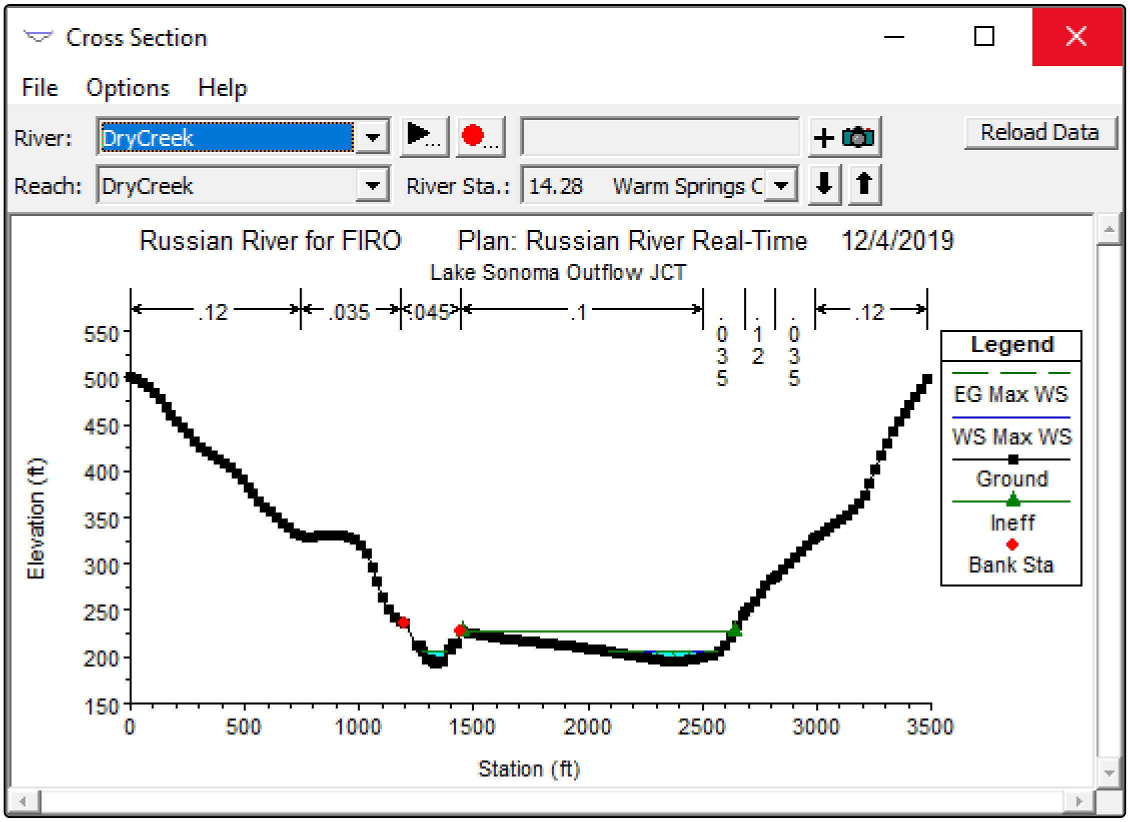


3. From the **File** menu, click **Exit**, and the **Rating Curve** dialog closes (Figure 7).

14.7.7 Cross Section Plot

During an HEC-RAS compute, at each defined cross section, data is generated. A cross section plot will display results from an HEC-RAS compute. To view cross section information for a selected HEC-RAS model alternative:

1. From the Modeling module, to view cross section information, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-RAS model alternative (*Russian River Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **Cross Section Plot**, the **Cross Section** dialog (Figure 8) opens. The **Cross Section** plot displays the computed results (i.e., maximum water surface elevation; energy grade) at the selected cross section (*14.28*). To view other cross sections, manipulate the **River**, **Reach**, and **River Sta.** lists (Figure 8). For further details on this HEC-RAS report, refer to the HEC-RAS User's Manual (Chapter 9).



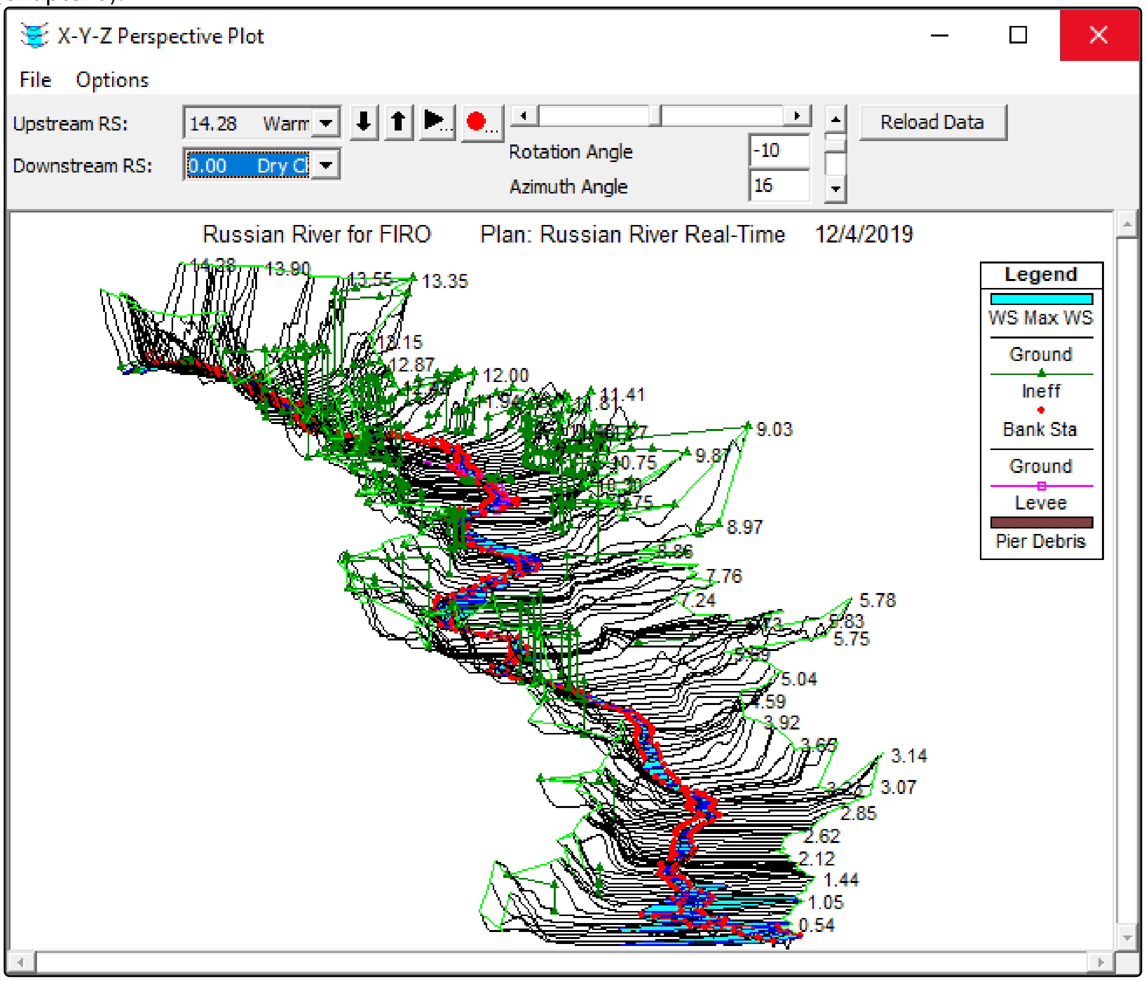
3. From the **File** menu, click **Exit**, and the **Cross Section** dialog closes (Figure 8).

14.7.8 X-Y-Z 3-D Plots

Once an HEC-RAS compute is complete, you can review multiple cross section within a reach from an X-Y-Z 3-D plot for the selected HEC-RAS model alternative. To view an X-Y-Z 3D plot:

1. From the Modeling module, to view multiple cross section within a reach, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-RAS model alternative (*Russian River Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **X-Y-Z 3-D Plot**, the **X-Y-Z Perspective Plot** dialog (Figure 9) opens. The **X-Y-Z Perspective Plot** displays multiple cross section within a reach. You can select which reaches to be plotted; the range of the cross sections (**Upstream RS**, **Downstream RS** lists); and, which HEC-RAS pans and profiles will be displayed. For further details on this HEC-RAS report, refer to the HEC-RAS User's Manual

(Chapter 9).



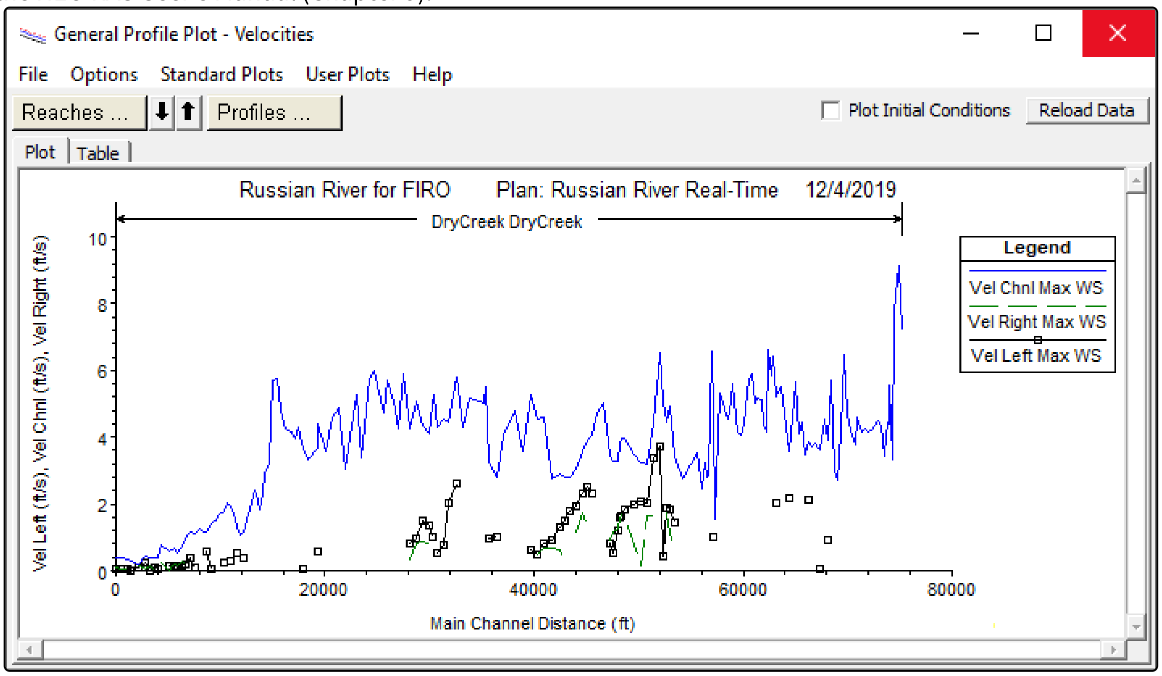
3. From the **File** menu, click **Exit**, and the **X-Y-Z Perspective Plot** dialog closes (Figure 9).

14.7.9 General Profile Plot

Once an HEC-RAS compute is complete, for a profile, you might want to plot variables other than water surface profile. Any variable that is computed at a cross section can be displayed in a profile. To view a general profile plot:

1. From the Modeling module, to view a general profile plot, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-RAS model alternative (*Russian River Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **General Profile Plot**, the **General Profile Plot - Velocities** dialog (Figure 10) opens. The **General Profile Plot - Velocities** displays velocity versus distance for a profile by default. There are predefined plots from the **Standard Plots** menu (Figure 10) the you can plot (i.e., flow, area, top width, weighted n, Froude number, hydraulic depth, shear, surface area, volume, steam power). For further details on this HEC-RAS report, refer to

the HEC-RAS User's Manual (Chapter 9).



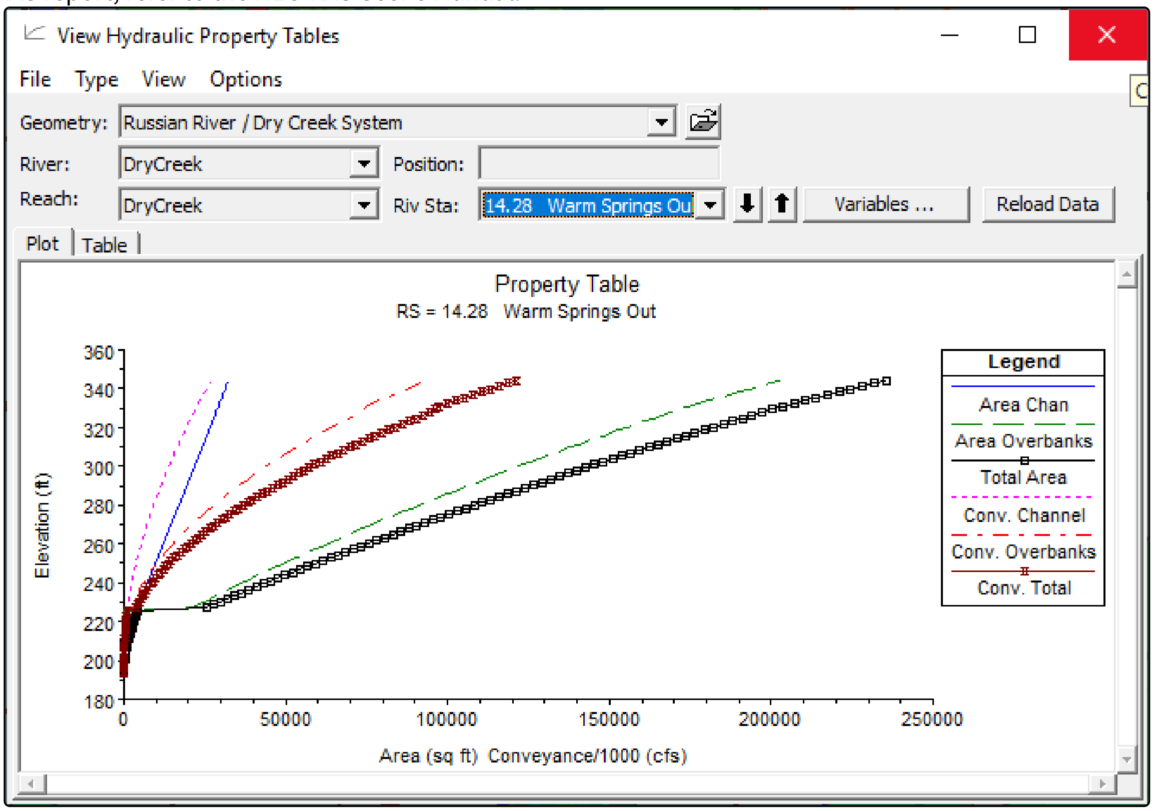
3. You can also create your own plots, from the **Options** menu (Figure 10), click **Plot Variables**. The **Y Axis Variable** dialog will open, from the **Y Axis Variable** dialog you choose which variables to plot and then save the plot (**User Plots**) for future use. For further details on this HEC-RAS report, refer to the HEC-RAS User's Manual (Chapter 9).
4. From the **File** menu, click **Exit**, and the **General Profile Plot - Velocities** dialog closes (Figure 10).

14.7.10 Hydraulic Properties Tables

Once an HEC-RAS compute is complete, you can view the computed curves based on the hydraulic table parameters that have been entered for the cross section and structures, for the selected HEC-RAS model alternative. To view hydraulic table curves:

1. From the Modeling module, to view hydraulic table curves, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-RAS model alternative (*Russian River Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **Hydraulic Properties Table**, the **View Hydraulic Property Tables** dialog (Figure 11) opens. The **View Hydraulic Property Tables** default plot displays curves for each variable that you have selected by cross section (*14.28*). Click **Variables** (Figure 11), from the available list you can turn on/off what variables to plot. To view other cross sections, manipulate the **River**, **Reach**, and **Riv Sta** lists (Figure 11). For further details on this HEC-

RAS report, refer to the HEC-RAS User's Manual.



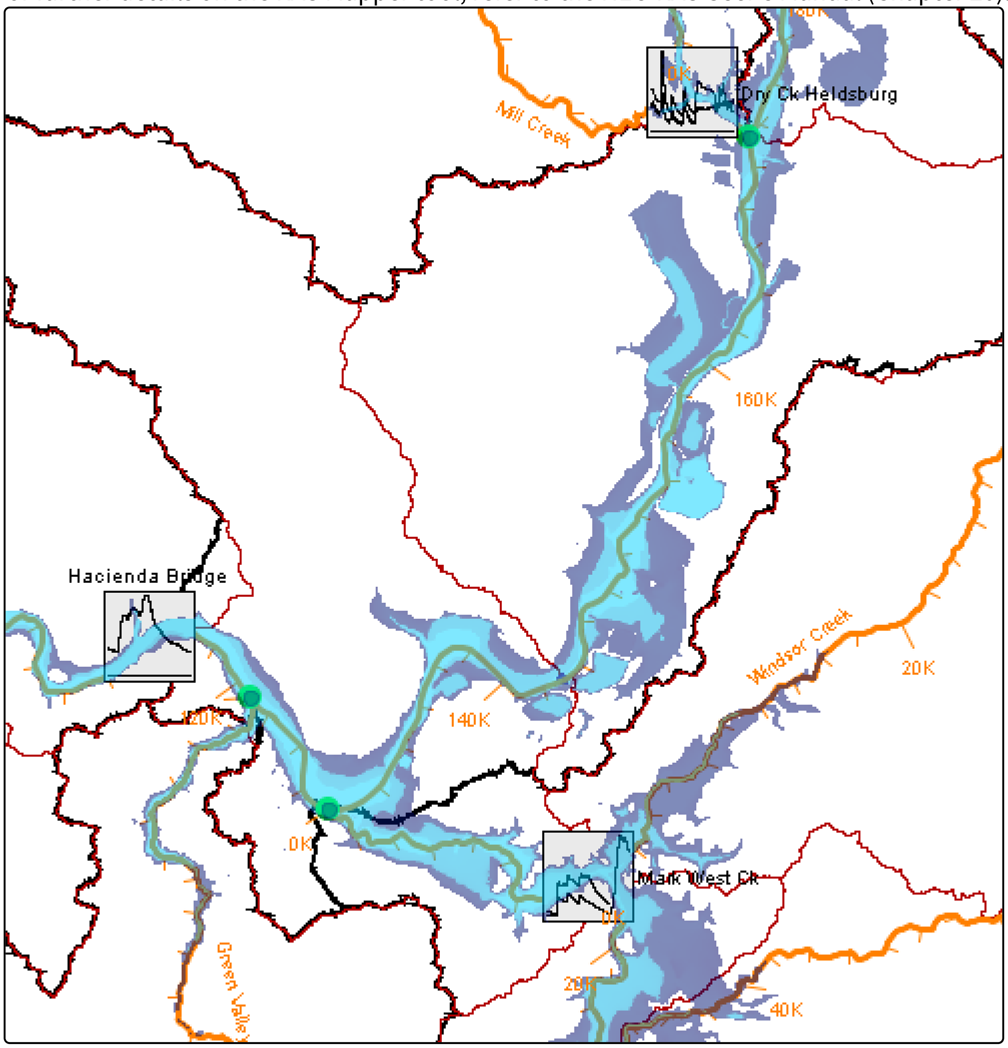
3. From the **Type** menu (Figure 11), you can change the type of values that are displayed. You have a choice between cross sections, internal boundaries, and storage area connections. For further details on this HEC-RAS report, refer to the HEC-RAS User's Manual (Chapter 9). From the **File** menu, click **Exit**, and the **View Hydraulic Property Tables** dialog closes (Figure 11).

14.7.11 RAS Mapper Results

Within the HEC-RAS software application, when performing an unsteady flow analysis, RAS Mapper is a tool that you can use to be build various type of inundation maps (Depth(max), Velocity(max), WSE(max), D*V(max), Arrival Time). If the HEC-RAS model alternative in an HEC-RTS forecast run, includes running RAS Mapper, and an HEC-RTS forecast compute is successfully, then you will have inundation map(s) that you could view in the **Map Window**. To view RAS Mapper inundation maps:

1. From the Modeling module, to view RAS Mapper inundation maps, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-RAS model alternative (*Russian River Real-Time*), from the **Forecast** tabs, click **Reports.**
2. From the **Reports** section, there are three buttons – **Depth (Max)**, **D_V(Max)**, and **Arrival Time (2ft hrs)**. These buttons are toggles, which will cause the RAS Mapper inundation maps to display in the **Map Window** (Figure 12). For example, in Figure 12, you have clicked **Arrival Time (2ft hrs)**, which will cause the associated inundation map to display. If you click **Arrival Time (2ft hrs)** again, the inundation map will no longer display.

For further details on the RAS Mapper tool, refer to the HEC-RAS User's Manual (Chapter 20).



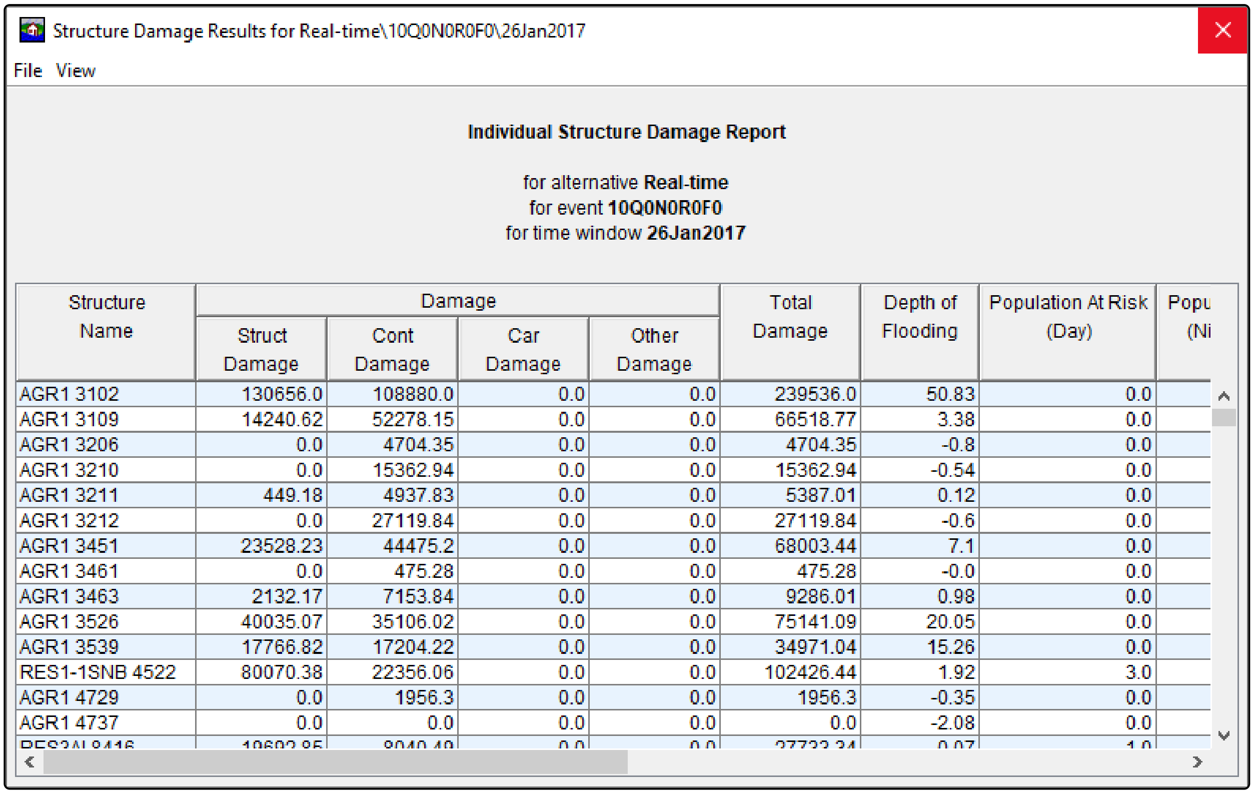
14.8 HEC-FIA Results

HEC-FIA may be used in HEC-RTS to predict the consequences associated with conditions predicted with HEC-HMS, HEC-ResSim, and HEC-RAS. This section briefly describes the HEC-FIA reports accessible from the CAVI. For more detail about the reports, please see the HEC-FIA users manual.

14.8.1 Individual Structure Damage Report

The **Individual Structure Damage Report** (Figure 1) provides information by individual structure, this information includes structure, content, car, other, total damage for each structure, depth of flooding, population at risk (day and night), percent not mobilized, percent caught evacuating, percent clear, and loss of life (day and night) information is also available for each structure. If your alternative is set to run a deterministic computation, this will display a single deterministic answer. However, if your HEC-FIA model alternative is set to run a Monte Carlo computation, this report will display the average for all iterations.

To view the **Individual Structure Damage Report**:


Structure Damage Results for Real-time\10Q0N0R0F0\26Jan2017

File View

Individual Structure Damage Report

for alternative **Real-time**
for event **10Q0N0R0F0**
for time window **26Jan2017**

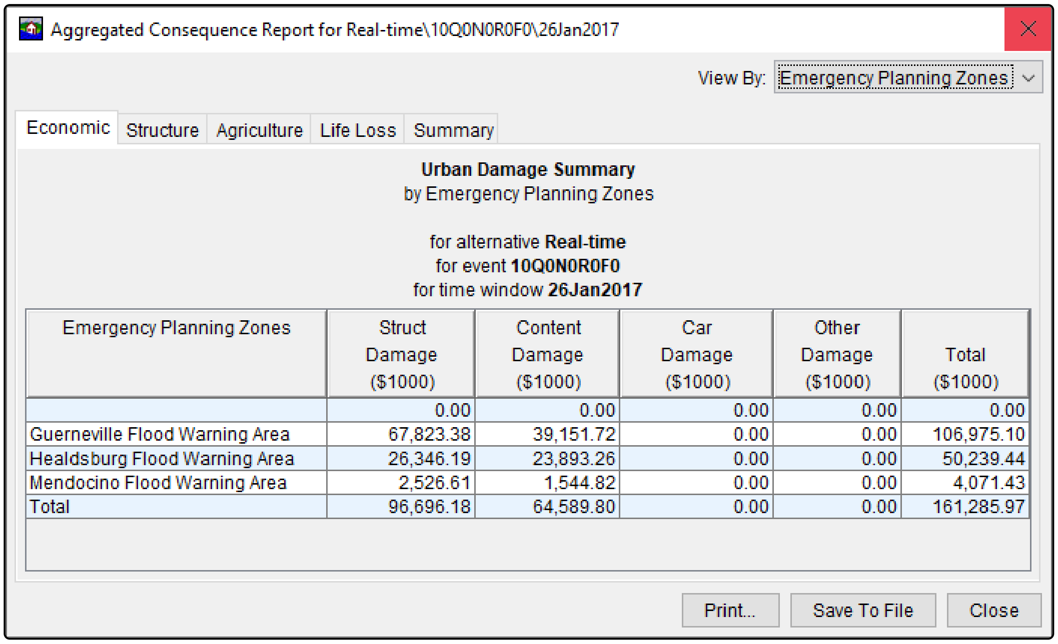
Structure Name	Damage				Total Damage	Depth of Flooding	Population At Risk (Day)	Popu (Ni
	Struct Damage	Cont Damage	Car Damage	Other Damage				
AGR1 3102	130656.0	108880.0	0.0	0.0	239536.0	50.83	0.0	
AGR1 3109	14240.62	52278.15	0.0	0.0	66518.77	3.38	0.0	
AGR1 3206	0.0	4704.35	0.0	0.0	4704.35	-0.8	0.0	
AGR1 3210	0.0	15362.94	0.0	0.0	15362.94	-0.54	0.0	
AGR1 3211	449.18	4937.83	0.0	0.0	5387.01	0.12	0.0	
AGR1 3212	0.0	27119.84	0.0	0.0	27119.84	-0.6	0.0	
AGR1 3451	23528.23	44475.2	0.0	0.0	68003.44	7.1	0.0	
AGR1 3461	0.0	475.28	0.0	0.0	475.28	-0.0	0.0	
AGR1 3463	2132.17	7153.84	0.0	0.0	9286.01	0.98	0.0	
AGR1 3526	40035.07	35106.02	0.0	0.0	75141.09	20.05	0.0	
AGR1 3539	17766.82	17204.22	0.0	0.0	34971.04	15.26	0.0	
RES1-1SNB 4522	80070.38	22356.06	0.0	0.0	102426.44	1.92	3.0	
AGR1 4729	0.0	1956.3	0.0	0.0	1956.3	-0.35	0.0	
AGR1 4737	0.0	0.0	0.0	0.0	0.0	-2.08	0.0	

78 Figure 1 HEC-FIA - Individual Structure Damage Report

1. From the Modeling module, to view the **Individual Structure Damage Report**, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-FIA model alternative (*Real-Time*), from the **Forecast** tabs, click **Reports.**
2. Click **Individual Structure Damage Report**, the **Individual Structure Damage Report** (Figure 1) opens. The **Individual Structure Damage Report** displays by structure the damages, flooding depth, life loss and other consequences associated with the defined forecast. For further details on this HEC-FIA report, refer to the HEC-FIA User's Manual.
3. From the **File** menu, click **Close**, and the **Individual Structure Damage Report** closes (Figure 1).

14.8.2 Aggregated Consequence Damages

The **Aggregated Consequence Report** (Figure 2) allows you to view damage by emergency planning zones, damage categories, and structure occupancy types. The report provides an **Urban Damage Summary**; **Structures Flooded Summary**; **Agriculture Summary**; **Life Loss Report**, and, a **Summary Report**.



79 Figure 2 HEC-FIA - Aggregated Consequence Report - Urban Damage Summary

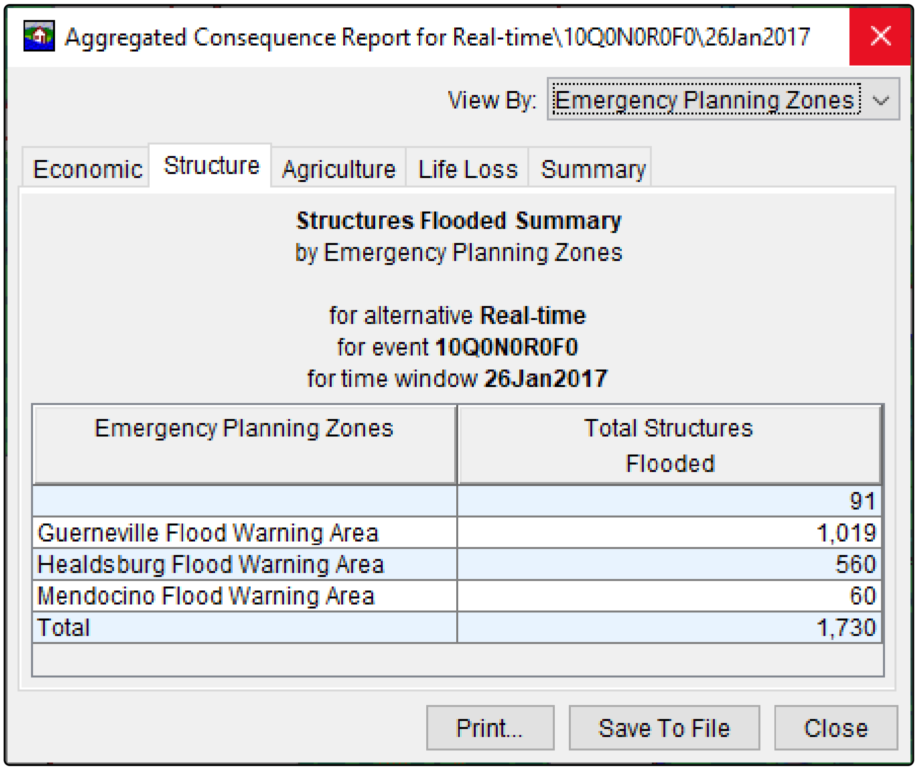
To view the **Urban Damage Summary**:

1. From the Modeling module, to view urban damage summary, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-FIA model alternative (*Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **Aggregated Consequence Damages**, the **Urban Damage Summary** (Figure 2) opens. The **Urban Damage Summary** report displays, by default the **Economic** damage (**Economic** tab) by **Emergency Planning Zones** (**View By**). This report provides structure damage, content damage, car damage, other damage, and total damage by emergency planning zones.
3. From the **View By** list (Figure 2) you can also view the **Urban Damage Summary** by **Damage Category** and **Occupancy Type**.
4. To close the **Urban Damage Summary**, click **Close**, and the **Aggregated Consequence Report** will close (Figure 2).

To view the **Structures Flooded Summary**:

1. From the Modeling module, to view structures flooded summary, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-FIA model alternative (*Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **Aggregated Consequence Damages**, the **Urban Damage Summary** (Figure 2) opens. Click the **Structure** tab, the **Structures Flooded Summary** displays (Figure 3), by **Emergency Planning Zones**. This report provides structure damage, content damage, car damage, other damage, and total damage by emergency planning

zones.

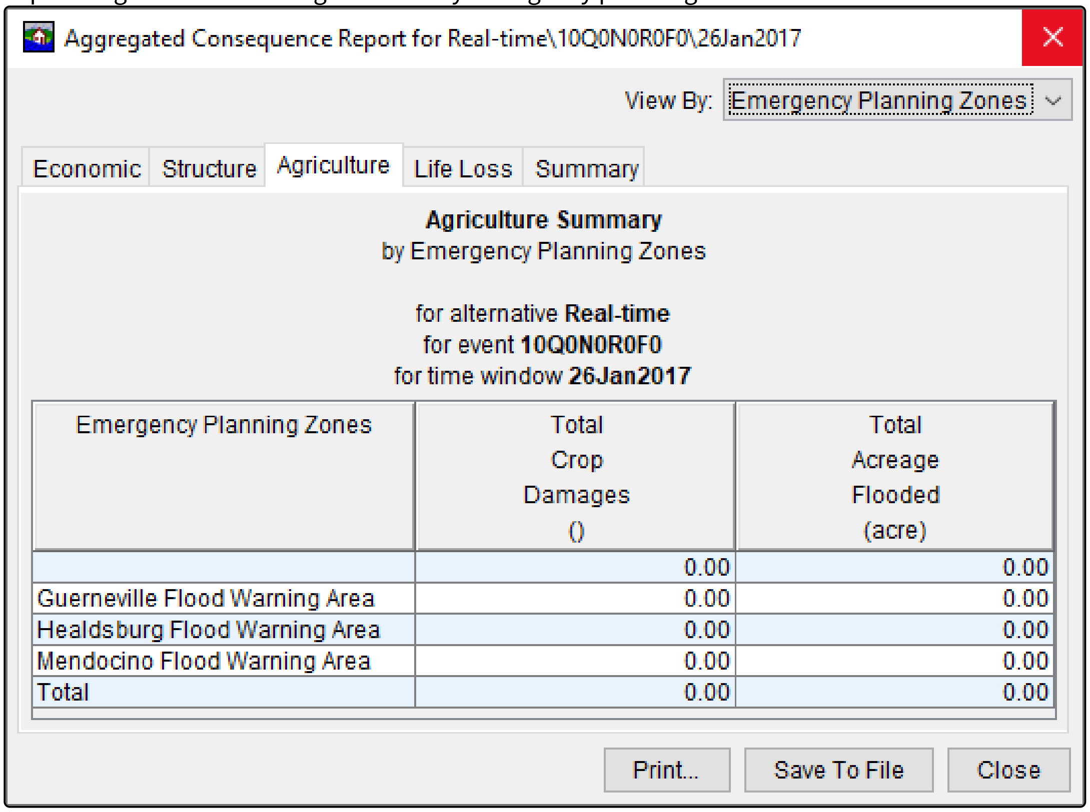


3. From the **View By** list (Figure 3) you can also view the **Structures Flooded Summary** by **Damage Category** and **Occupancy Type**.
4. To close the **Structures Flooded Summary**, click **Close**, and the **Aggregated Consequence Report** will close (Figure 2).

To view the **Agriculture Summary**:

1. From the Modeling module, to view agriculture summary, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-FIA model alternative (*Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **Aggregated Consequence Damages**, the **Urban Damage Summary** (Figure 2) opens. Click the **Agriculture** tab, the **Agriculture Summary** displays (Figure 4), by **Emergency Planning Zones**. This report provides total

crop damage and total acreage flooded by emergency planning zones.

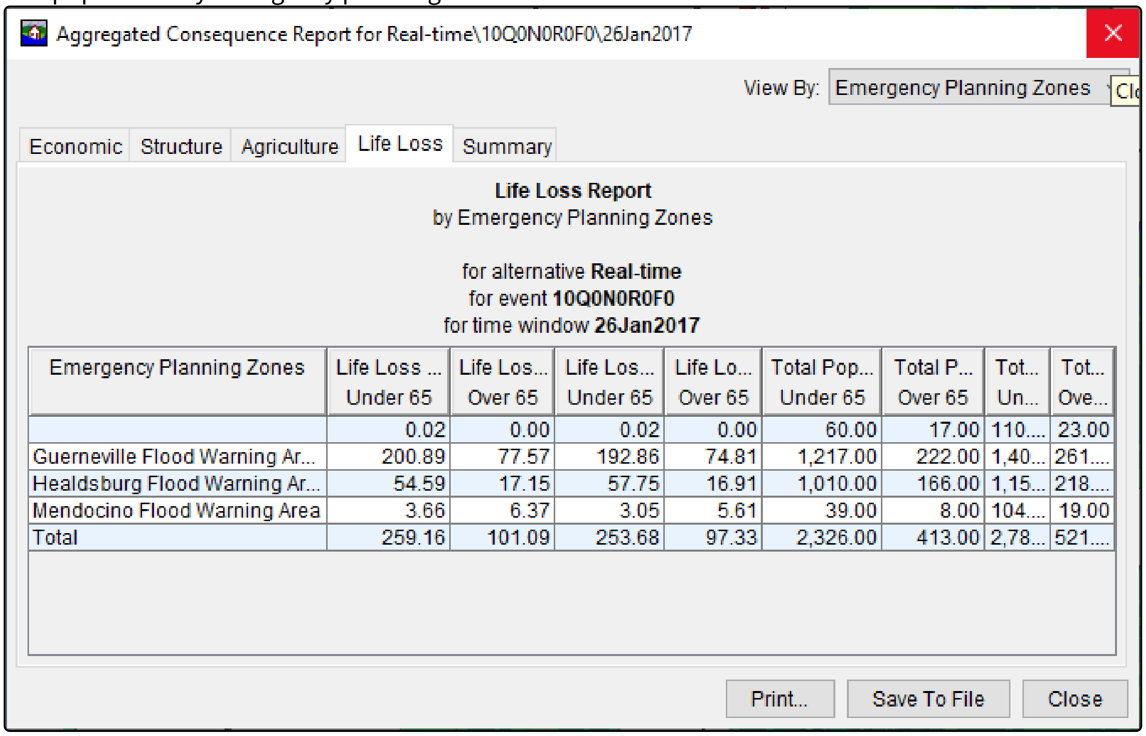


3. From the **View By** list (Figure 4) you can also view the **Agriculture Summary** by **Damage Category** and **Occupancy Type**.
4. To close the **Agriculture Summary**, click **Close**, and the **Aggregated Consequence Report** will close (Figure 4).

To view the **Life Loss Report**:

1. From the Modeling module, to view the life loss report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-FIA model alternative (*Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **Aggregated Consequence Damages**, the **Urban Damage Summary** (Figure 2) opens. Click the **Life Loss** tab, the **Life Loss Report** displays (Figure 5), by **Emergency Planning Zones**. This report provides life loss and

total population by emergency planning zones.

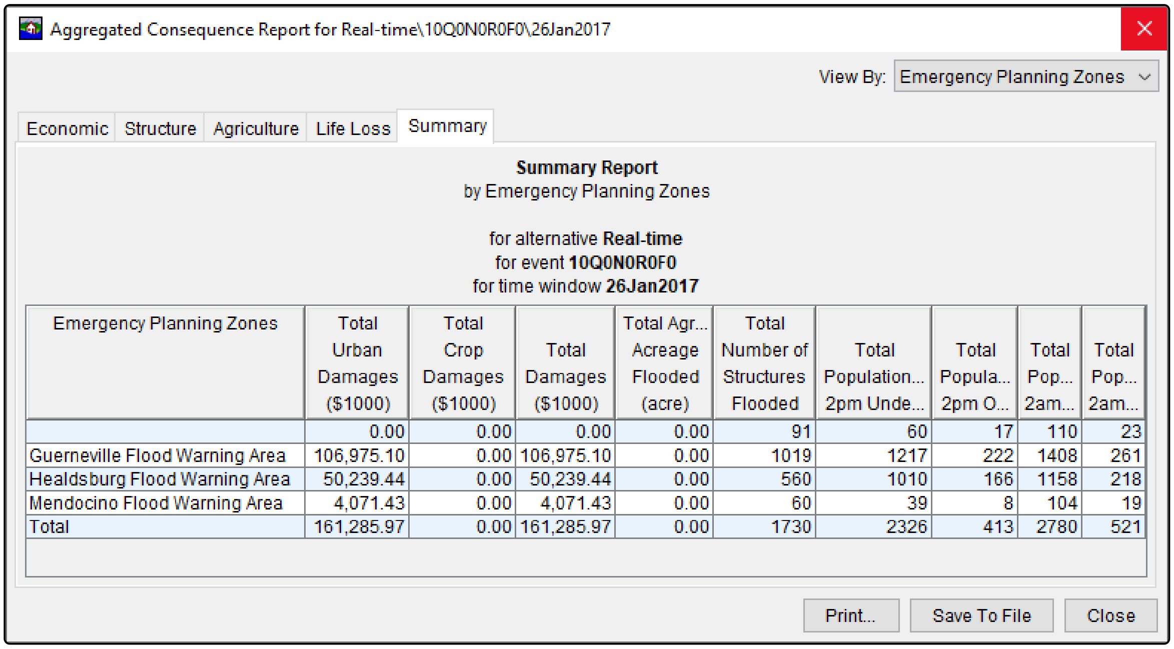


Emergency Planning Zones	Life Loss ... Under 65	Life Los... Over 65	Life Los... Under 65	Life Lo... Over 65	Total Pop... Under 65	Total P... Over 65	Tot... Un...	Tot... Ove...
	0.02	0.00	0.02	0.00	60.00	17.00	110....	23.00
Guerneville Flood Warning Ar...	200.89	77.57	192.86	74.81	1,217.00	222.00	1,40...	261....
Healdsburg Flood Warning Ar...	54.59	17.15	57.75	16.91	1,010.00	166.00	1,15...	218....
Mendocino Flood Warning Area	3.66	6.37	3.05	5.61	39.00	8.00	104....	19.00
Total	259.16	101.09	253.68	97.33	2,326.00	413.00	2,78...	521....

3. From the **View By** list (Figure 5) you can also view the **Life Loss Report** by **Damage Category** and **Occupancy Type**.
4. To close the **Life Loss Report**, click **Close**, and the **Aggregated Consequence Report** will close (Figure 5).

To view the **Summary Report**:

1. From the Modeling module, to view the summary report, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-FIA model alternative (*Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **Aggregated Consequence Damages**, the **Urban Damage Summary** (Figure 2) opens. Click the **Summary** tab, the **Summary Report** displays (Figure 6), by **Emergency Planning Zones**. This report provides information on urban damage, agricultural damage, total damage, acreage flooded, structures floods, and population impacted.
3. From the **View By** list (Figure 6) you can also view the **Summary Report** by **Damage Category** and **Occupancy Type**.



4. To close the **Summary Report**, click **Close**, and the **Aggregated Consequence Report** will close (Figure 6).

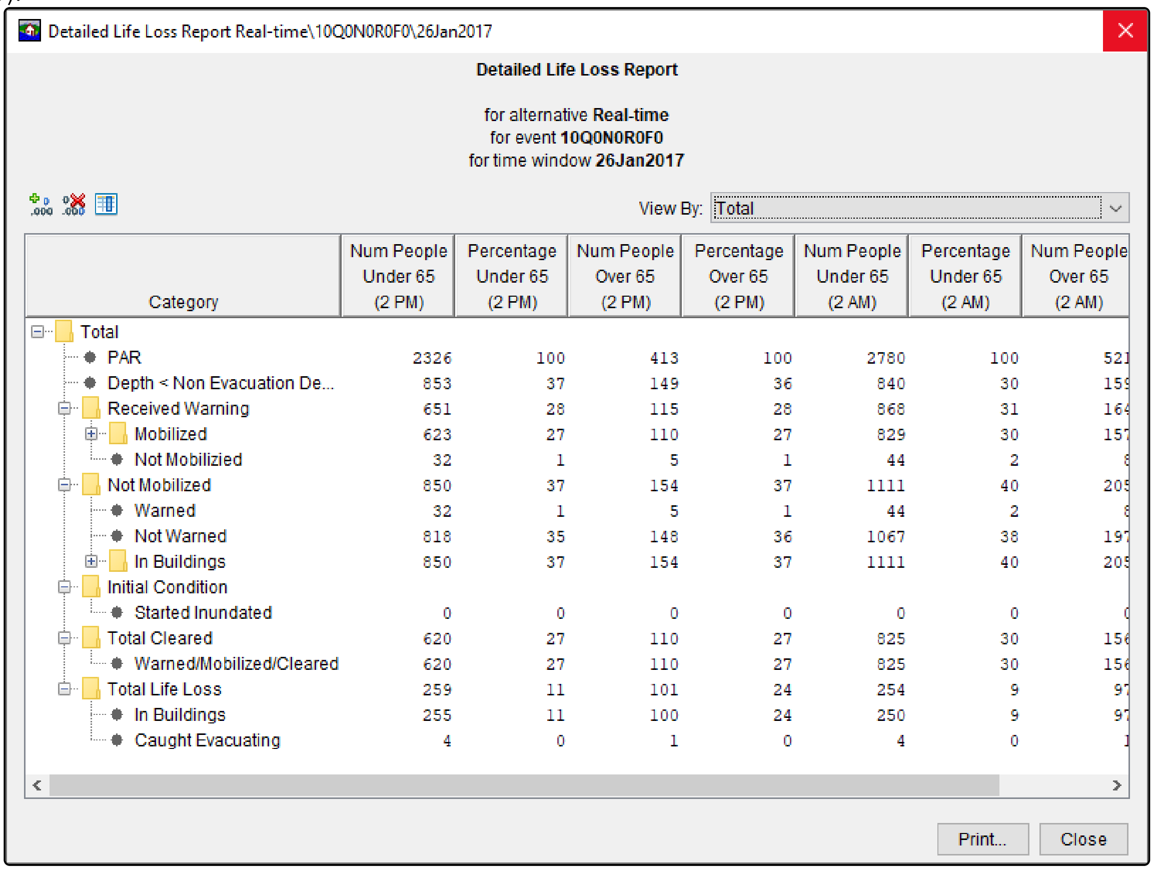
14.8.3 Detailed Life Loss Report

The **Detailed Life Loss Report** is dependent on the HEC-FIA model alterative being setup to compute life loss. This report provides information about how HEC-FIA calculated life loss for the watershed.

To view the **Detailed Life Loss Report**:

1. From the Modeling module, to view the **Detailed Life Loss Report**, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-FIA model alternative (*Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **Life Loss**, the **Detailed Life Loss Report** (Figure 7) displays by **Total**. The **Detailed Life Loss Report** provides information about how HEC-FIA calculated life loss for the watershed. For further details on this HEC-FIA report, refer to the HEC-FIA User's Manual.
3. From the **View By** list (Figure 7) you can also view the **Detailed Life Loss Report** by **Emergency Planning Zone**, **Damage Category**, and **Occupancy Type**.

4. To close the **Detailed Life Loss Report**, click **Close**, and the **Detailed Life Loss Report** dialog will close (Figure 7).



Category	Num People Under 65 (2 PM)	Percentage Under 65 (2 PM)	Num People Over 65 (2 PM)	Percentage Over 65 (2 PM)	Num People Under 65 (2 AM)	Percentage Under 65 (2 AM)	Num People Over 65 (2 AM)
Total							
PAR	2326	100	413	100	2780	100	521
Depth < Non Evacuation De...	853	37	149	36	840	30	159
Received Warning	651	28	115	28	868	31	164
Mobilized	623	27	110	27	829	30	157
Not Mobilizied	32	1	5	1	44	2	8
Not Mobilized	850	37	154	37	1111	40	205
Warned	32	1	5	1	44	2	8
Not Warned	818	35	148	36	1067	38	197
In Buildings	850	37	154	37	1111	40	205
Initial Condition							
Started Inundated	0	0	0	0	0	0	0
Total Cleared	620	27	110	27	825	30	156
Warned/Mobilized/Cleared	620	27	110	27	825	30	156
Total Life Loss	259	11	101	24	254	9	97
In Buildings	255	11	100	24	250	9	97
Caught Evacuating	4	0	1	0	4	0	1

14.8.4 ECAM Reports

The **ECAM Report** is dependent on the HEC-FIA model alterative being setup to compute life loss and ECAM. For the example HEC-RTS watershed being used in this User's Manual, the HEC-FIA model alternative (*Real-Time*) is not setup for an ECAM compute, therefore no results.

To view ECAM reports:

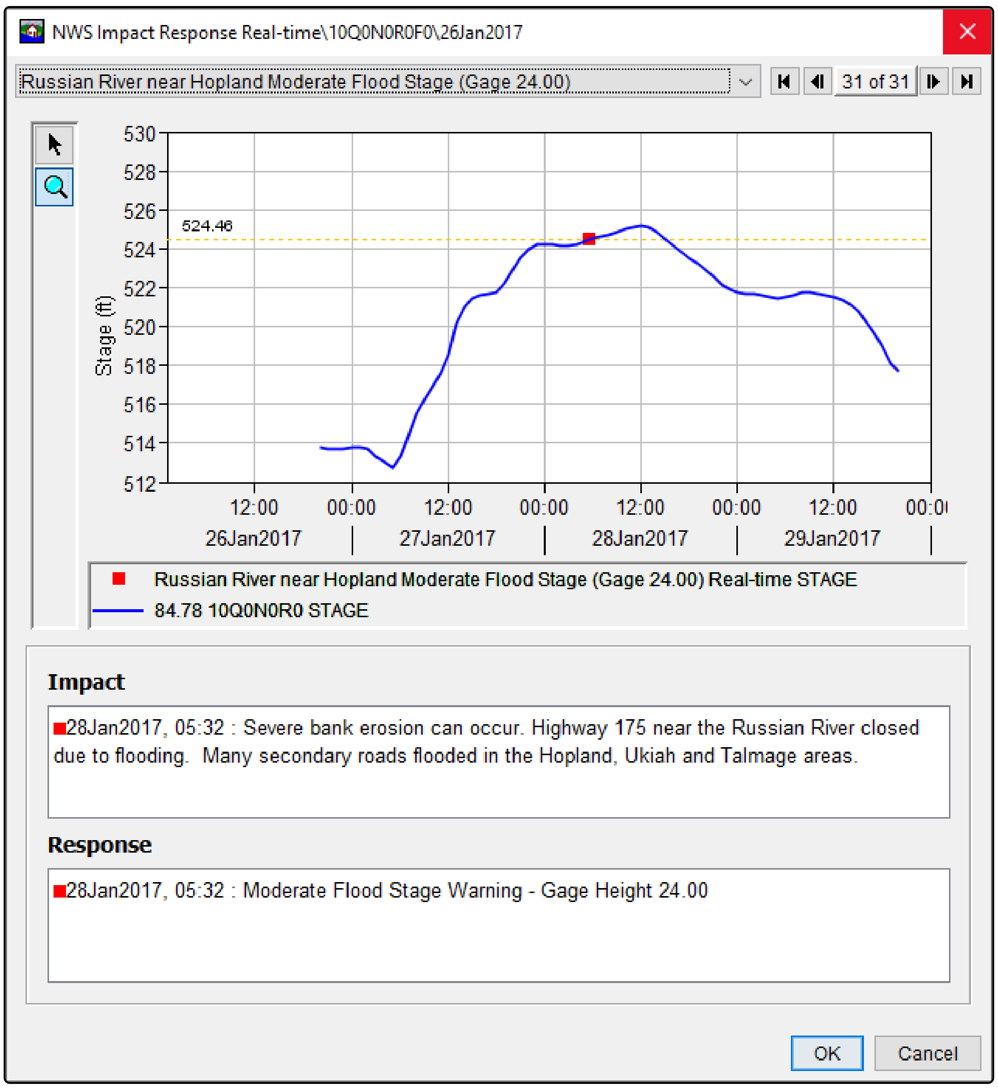
1. From the Modeling module, to view the ECAM reports, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-FIA model alternative (*Real-Time*), from the **Forecast** tabs, click **Reports**.
2. There are two ECAM reports (tabs) - **Output** and **Employment**. When the **Output** tab is selected, the **ECAM Indirect Economics Output Report** displays. The **ECAM Indirect Economics Output Report** provides information about the output of the economy, by sector (e.g., forestry, food processing, general manufacturing, etc.) for a selected county (**County** list). When the **Employment** tab is selected, the **ECAM Indirect Economics Employment Report** displays. The **ECAM Indirect Economics Employment Report** provides information representing the changes in labor employed by sector (e.g., forestry, food processing, general manufacturing, etc.) for a selected county (**County** list).

3. To close either ECAM report, click **Close**, and the ECAM report will close. For further details on this HEC-FIA report, refer to the HEC-FIA User's Manual.

14.8.5 Impact Response By Rule Report

The **Impact Response By Rule Report** is dependent on the HEC-FIA model alterative having an impact response element defined. This report displays the hydrograph at the location of the rule, and the threshold elevation relative to the hydrograph. The **Impact Response By Rule Report** displays each rule independently.

To view the **Impact Response by Rule Report** (Figure 8):



80 Figure 8 HEC-FIA - Impact Response by Rule Report

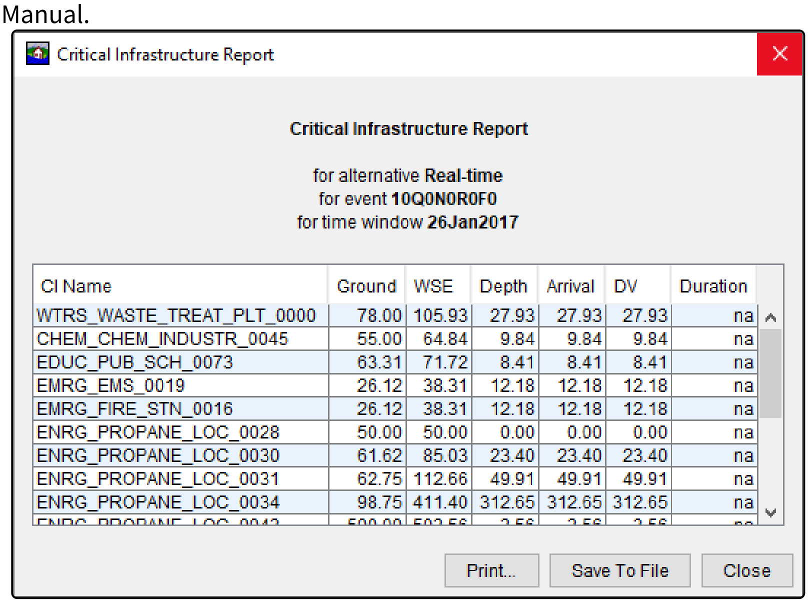
1. From the Modeling module, to view the **Impact Response by Rule Report**, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-FIA model alternative (*Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **Impact Response By Rule**, a dialog will open (Figure 8). From the list at the top of the dialog (Figure 8), select which rule you would like to review (e.g., *Russian River near Hopland Moderate Flood Stage (Gage 24.00)*). A plot of the hydrograph at the location of the rule is displayed (Figure 8). In addition, both the **Impact** and **Response** boxes provide details. For further details on this HEC-FIA report, refer to the HEC-FIA User's Manual.
3. To close the **Impact Response By Rule Report**, click **OK**, and the dialog will close (Figure 8).

14.8.6 Critical Infrastructure Report

The **Critical Infrastructure Report** is dependent on the HEC-FIA model alterative having a critical infrastructure inventory defined. The report only includes critical infrastructure elements which were inundated.

To view the **Critical Infrastructure Report**:

1. From the Modeling module, to view the **Critical Infrastructure Report**, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-FIA model alternative (*Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **Critical Infrastructure**, the **Critical Infrastructure Report** (Figure 9) displays. For each critical infrastructure, the **Critical Infrastructure Report** provides information about elevations at ground, water surface, depth, arrival, and duration. For further details on this HEC-FIA report, refer to the HEC-FIA User's Manual.



3. To close the **Critical Infrastructure Report**, click **Close**, and the **Critical Infrastructure Report** will close (Figure 9).

14.8.7 Flood Damage Reduction Reports

The flood damage reduction reports are dependent on the HEC-FIA model alterative being setup with the necessary hydraulic data and holdout inundation configuration. For the example HEC-RTS watershed being used in this User's Manual, the HEC-FIA model alternative (*Real-Time*) is not setup properly for generating flood damage reduction reports.

To view flood damage reduction reports:

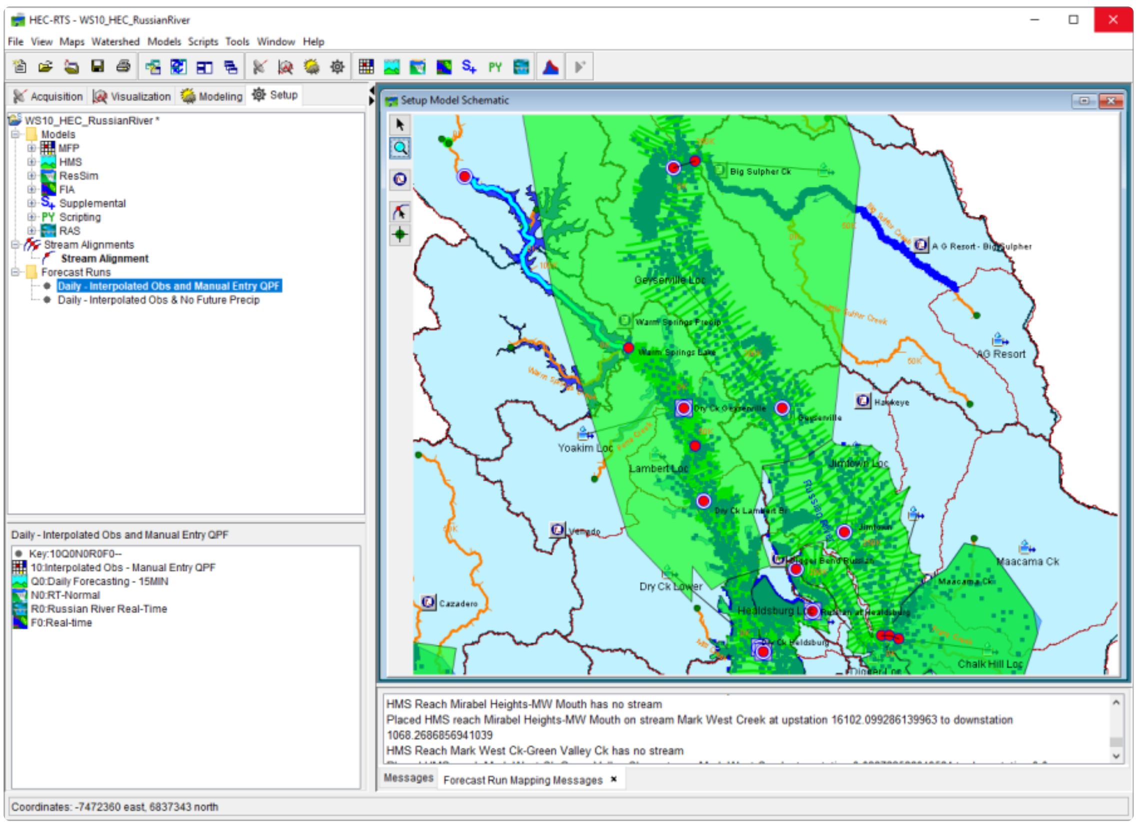
1. From the Modeling module, to view the flood damage reduction reports, make sure you have an active forecast and at least one forecast run selected (checkbox) for viewing results. From the **Forecast Runs Detail** section, select an HEC-FIA model alternative (*Real-Time*), from the **Forecast** tabs, click **Reports**.
2. Click **Flood Damage Reduction Aggregated Report**, the **Flood Damage Reduction Aggregated Report** displays. The **Flood Damage Reduction Aggregated Report** by default displays the computed impacts by the holdout distribution area and by reservoirs for the structure damage reduction allocation.
3. Click **Flood Damage Reduction Project Report**, the **Flood Damage Reduction Project Report** displays. The **Flood Damage Reduction Project Report** by default displays the computed impacts by project and by reservoirs for the structure damage reduction allocation. For each project damage reduced is reported, and whether a project is a USACE project or not.
4. To close either flood damage reduction reports, click **Close**, and the flood damage reduction report will close. For further details on this HEC-FIA report, refer to the HEC-FIA User's Manual.

15 Configuring HEC-RTS

In this chapter, we describe a workflow to configure the model alternatives within HEC-RTS. This workflow description assumes previous development of the HEC-RTS study and the individual models in their native software applications (HEC-HMS, HEC-ResSim, HEC-RAS, and HEC-FIA) are complete.

15.1 Setting Up Your HEC-RTS Study

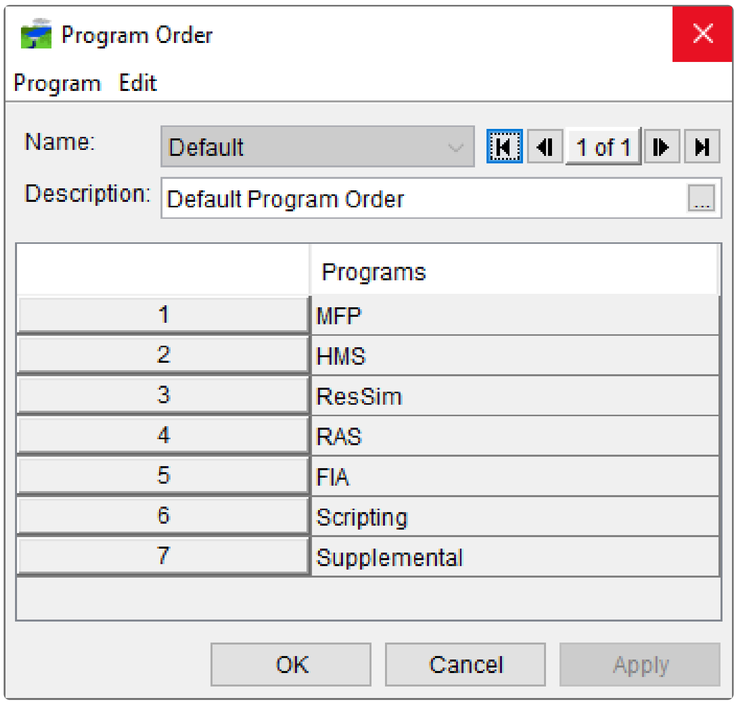
In HEC-RTS, you setup a watershed to reflect a collection of data, information, application models and images that represent a watershed and water control features within a watershed. The basic setup of your HEC-RTS study is done in the **Setup** module (Figure 1) of HEC-RTS. You will setup items that describe a watershed's physical arrangement. When you create a new watershed, a directory structure is generated where all files associated with the watershed are stored.



Once you have created a new watershed, you are able to import maps from external sources, specify the unit system and time zone for the watershed, add layers containing additional information about the watershed such as streams. This chapter will discuss the main items that you will be doing in the **Setup** module: create watersheds; define the computing sequence for forecast runs; create time series icons; create time series layers; import and define keys for model alternatives; create forecast runs; setup the model linking between model alternatives in a forecast run; and, manage extract and post groups. For further details, refer to Setup Module.

15.2 Defining the Order of Application Programs

The order of program execution within HEC-RTS is a fundamental part of the watershed setup, which you defined from the **Setup** module. You need to decide on the program order (Figure 1) early on in your setup, and if you change the order after a forecast has been configured, then the flow of data will be disrupted and results will be invalidated. Refer to Program Order for additional details for defining the order for application programs.



15.3 Developing Time Series Icons

You develop time series icons in the **Map Window** of the **Setup** module. Time series icon are used by the other modules to display data and information availability. The information can be time series data, images, web pages, webcams, scripts, and document files. You can link time series icons to data from DSS files. Time series icons provide you easy access to analyzing incoming data and for providing location specific information.

You can represent time series icons as symbols, thumbnail plots, or color-coded data quality bars. You should define and setup the time series icons early in the watershed development so data can be readily viewed. Occasionally, you may need to make changes to the time series icons to accommodate additional data development. Refer to Time Series Icons and Time Series Icons Layers for additional information describing time series icons.

15.4 Developing Model Alternatives

A **model alternative** is a single configuration with a specific set of input and parameters. A second model alternative would be defined by a different configuration, different input, or different parameters. Since model alternatives do not include time-specific data, you can use a model alternative with any time window.

Initially, you develop alternatives using the software of the native model (e.g., HEC-HMS, HEC-ResSim, HEC-RAS, and HEC-FIA). Then, you import these alternatives into HEC-RTS. Once the model alternatives have been imported, from the **Setup** module you can configure and edit specific model alternatives for use in HEC-RTS.

For example, since MFP deals with future precipitation, you can configure model alternatives to use in conjunction with HEC-HMS to compute a variety of future precipitation scenarios. Since HEC-HMS simulates watershed response to precipitation, then you can use HEC-HMS alternatives to forecast future flows. These future flows can be used with the reservoir software application HEC-ResSim, to compute reservoir releases. Regulated flows and stages from HEC-ResSim can be input to HEC-RAS and HEC-FIA model alternatives. Basically, all individual model alternatives created in the native programs are available to HEC-RTS.

In the **Setup** module, you will create model alternative keys that will be used in creating a forecast run. Each key is a single alpha-numeric identifier that represents a model alternative. For example, the Key N might represent normal conditions, the Key W might represent wet conditions, and the Key D might represent dry conditions. Refer to Model Alternatives and Forecast Runs for additional information on model alternatives.

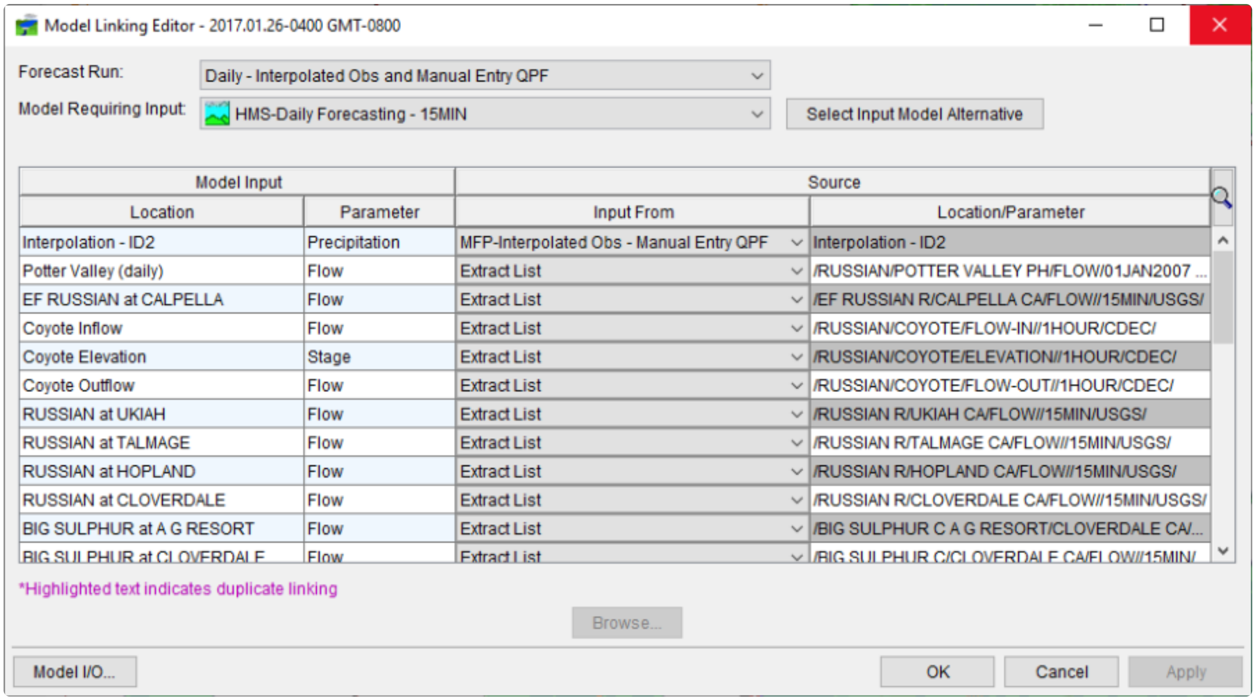
15.5 Creating Forecast Runs

A **forecast run** is a single forecast scenario defined by a specific set of data, information, and model alternatives. You create forecast runs in the **Setup** module. You use the previously defined model alternative keys to create a forecast run key that consists of a collection of the model alternative keys.

As an example, you might name a forecast run *MFP-HMS-ResSim*. For this example, the forecast run key might be **R**0**B**0**N**0---- which could reflect **R**adar precipitation data for MFP, **B**ase data for HEC-HMS, and **N**ormal Operations for HEC-ResSim. In this example, the dashes indicate that HEC-RAS and HEC-FIA are not being run. Refer to Model Alternatives and Forecast Runs for additional information on forecast runs.

15.6 Linking Models

After a forecast run has been created you need to link the models, which is accomplished through the **Model Linking Editor** (Figure 1). From this editor you select a forecast run you wish to configure. Then you select each model alternative that is associated with the selected forecast run. For each model alternative you will determine where you will be getting data from another model alternative. For example, in Figure 1, you specify which model alternative the input comes from and the DSS pathname associated with the model results. For example, you may set the precipitation input data for HEC-HMS to come from an MFP precipitation grid. Refer to Model Linking Editor for additional information on model linking.



15.7 Data Extract and Posting

Once the models have been linked the next step is to configure the data extract groups. A data extract group specifies the location of the gridded and time series data required for the different model alternatives. For example, this could be gridded precipitation data in DSS or observed flow data from another DSS file. Once you have completed computing your forecast, you might want to post results back to the HEC-RTS DSS study file. This task is completed by creating post groups that specify what time series data needs to be save. Refer to Data Extract and Post Editors for additional information.

16 Setup Module

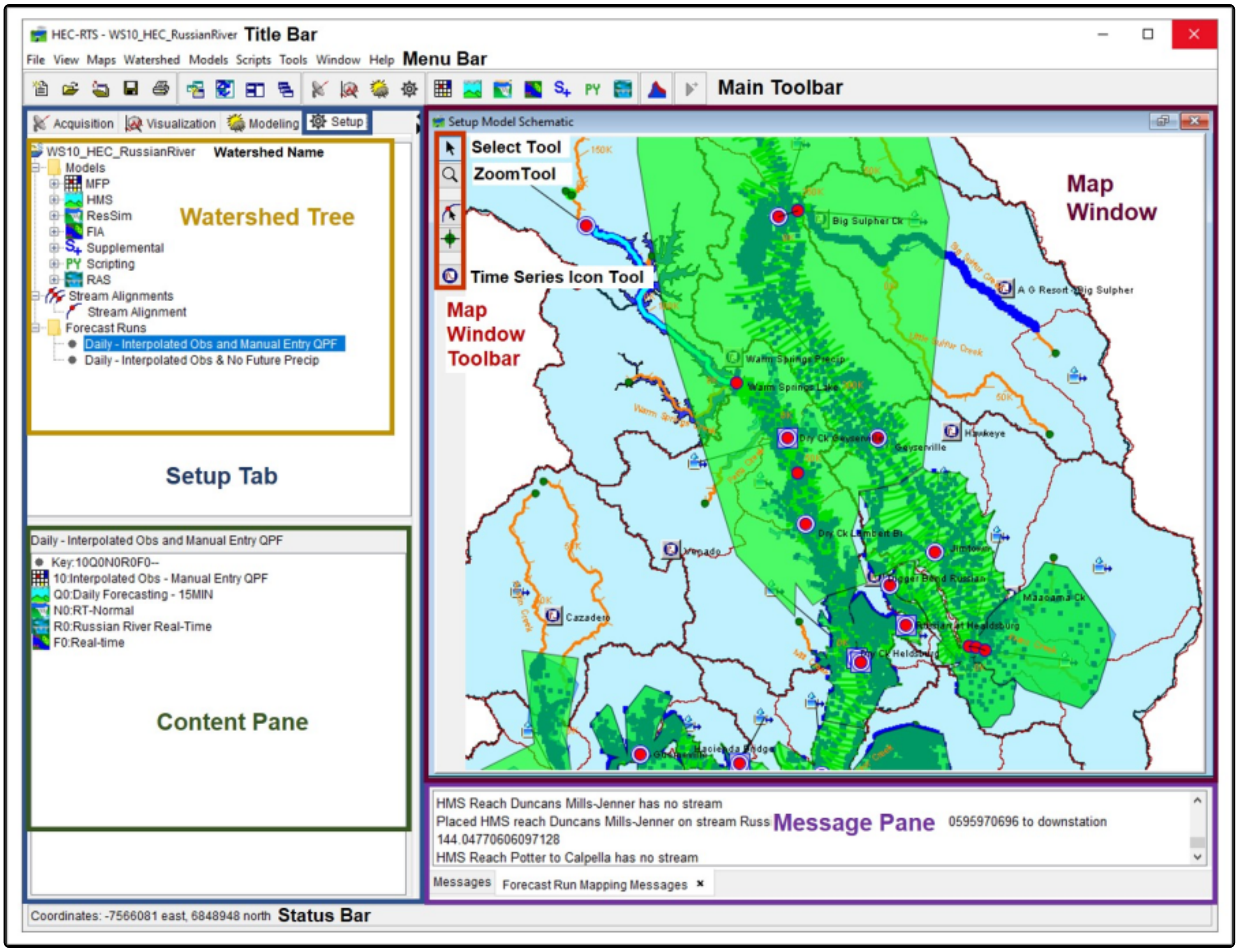
In HEC-RTS, a **watershed** is a set of data, information, models, and images that represent watershed lands and the channels, gages, and water control features within the watershed. The **Setup** module is the grouping of commands for watershed setup. These are commands for configuring inputs, models, and outputs that describe a watershed's behavior. You develop a visual representation of the watershed to display in HEC-RTS that is map-based. This chapter describes the Setup module, which includes an overview of the Setup module interface, how to create a new watershed, and reports on watershed setup.

In the Setup module, you configure a watershed's physical arrangement, including maps and geo-extents, time series icons, and other geo-referenced data. Once you have setup a watershed, you will be able to use the data you have defined within HEC-ResSim, HEC-FIA, and other geo-referenced HEC-RTS applications.

Because the order of program execution (Program Order) is a fundamental part of the watershed setup, you should define it early in the process of constructing datasets for your watershed. You establish the program order in the Setup module. This involves determining the set of programs that HEC-RTS might execute, the order of program execution, and the data flow for forecasting. You may also define supplemental software applications and add the supplemental software applications to the program order. A **supplemental program** is a software application or script that you develop and by default is not included with HEC-RTS.

16.1 Setup Module Overview

The Setup module is the grouping of commands for configuring an HEC-RTS watershed. From the HEC-RTS main window, click the **Setup** tab (Figure 1). A detailed discussion of common screen components is presented in HEC-RTS Interface. The **Setup** tab (Figure 1) displays the name of the watershed, the tree of the watershed, and the schematic of the watershed. The following sections provide an overview of the Setup module interface.



81 Figure 1 Setup Module

16.1.1 Menu Bar

The following is an overview of the **Menu Bar** (Figure 1) for the Setup module. The **File**, **View**, **Maps**, **Scripts**, **Tools**, **Window**, and **Help** menus and their commands are common to all modules and are discussed in HEC-RTS Main Window.

Watershed	From this menu, you can view the watershed properties; view a tree of the watershed files; manage stream reach aliases; view a list of the streams in the watershed; import or export stream alignment files (refer to Stream Alignment); import a CWMS 2.1 watershed; and, team modeling. The available commands are: **Watershed Properties**, **Files**, **Stream Reach Alias Editor**, **List of Streams**, **Stream Alignment** (refer to Stream Alignment), **Import 2.1 Watershed**, and **Team**.

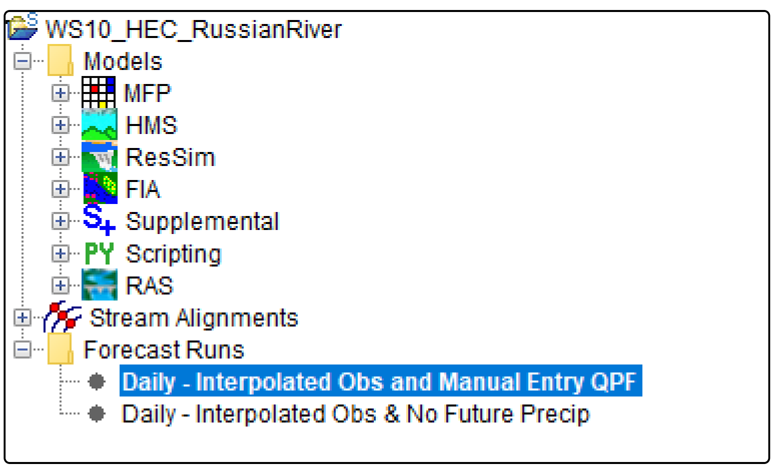
Model s	From this menu you can review the steps of the Setup module; establish the program order; import model alternatives; specify model alternative keys; configure forecast runs; setup model linking; and, access the extract and post editors. Available commands are: **Next Steps**, **Program Order**, **Import**, **Model Alternative Keys**, **Forecast Runs**, **Model Linking**, **Edit Extract**, **Edit Post**, and commands related to models configured in the watershed (e.g., **HMS**).

16.1.2 Map Window

The map window in the Setup module (Figure 1) displays a schematic of the watershed. Depending on where you are at in the creation of a watershed, the map window will display the current status of the watershed. From the **Watershed Tree**, you can also influence what is displayed in the Setup module map window. The map window tools allow the user to access and configure elements (time series icons, stream alignment, stream nodes) within your watershed. A description of the map and stream alignment tools is included in Map Window.

16.1.3 Watershed Tree

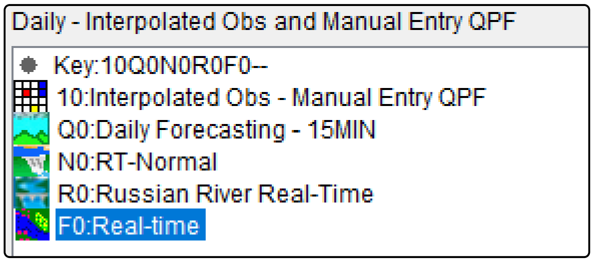
The **Watershed Tree** (Figure 2) provides a visual of the selected watershed in a tree format. At the top of the tree is the name of the watershed (*WS10_HEC_Russian River*). There are three main nodes (folders) to the watershed tree – **Models**, **Stream Alignments**, and **Forecast Runs**. You can expand (⊞) or collapse (⊟) the folders to view further details about the selected watershed. For example, in Figure 1, the **Forecast Runs** folder has been expanded, which displays the available forecast runs for the watershed. Double-click on a forecast run (*Daily – Interpolated Obs and Manual Entry QPF*), the map window (Figure 1) will now display the schematic for the selected forecast run.



82 Figure 2 Watershed Tree

16.1.4 Content Pane

The **Content Pane** (Figure 3) displays information based on selections made in the **Watershed Tree**. For example, in Figure 2, a forecast run (*Daily – Interpolated Obs and Manual Entry QPF*) has been selected. The **Content Pane** (Figure 3) displays information about the forecast run that includes the name of the forecast run, the key code of the forecast run, and the model alternatives that make-up the forecast run.



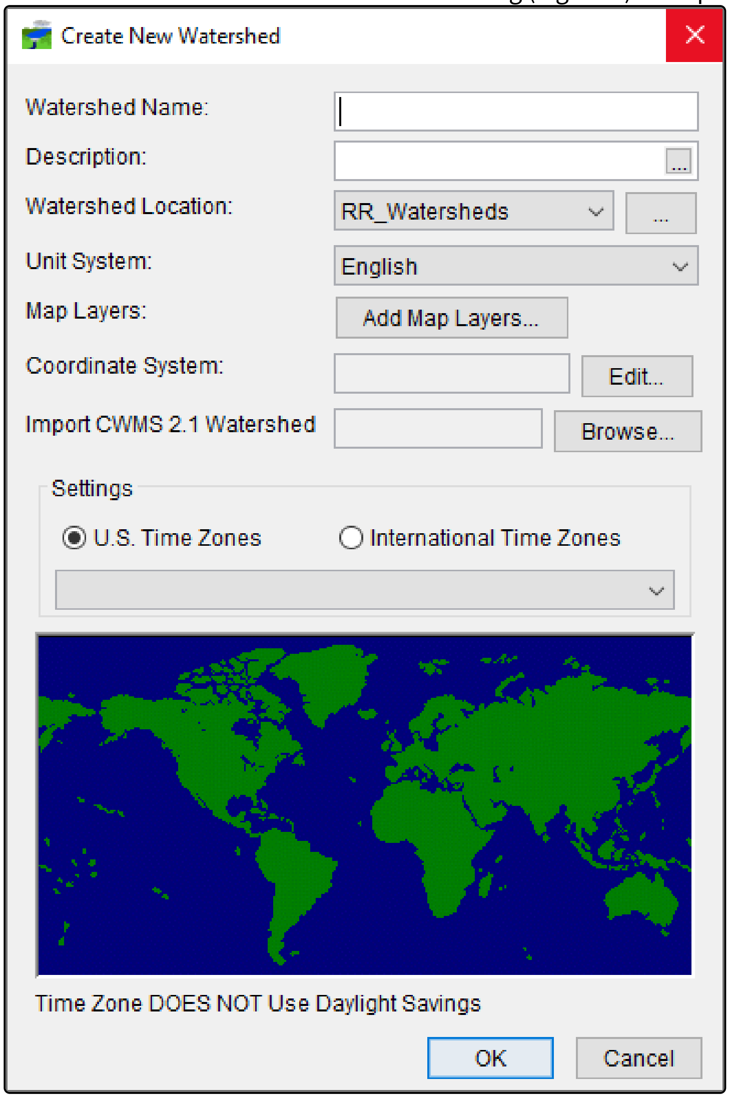
83 Figure 3 Content Pane

16.2 Creating a Watershed

The first thing you must do when creating a watershed is to give the watershed a name and description, specify the watershed's location, and establish the system of units and the time zone for the watershed. Once you have configured the new watershed, HEC-RTS generates a new directory hierarchy in the specified watershed directory. The new watershed becomes active in the main window, and the tools needed to configure watershed components become available. At this point you may continue to configure the new watershed by adding layers, setting preferences for the map window, creating time series icons, establishing the database connectivity, and configuring models and model alternatives.

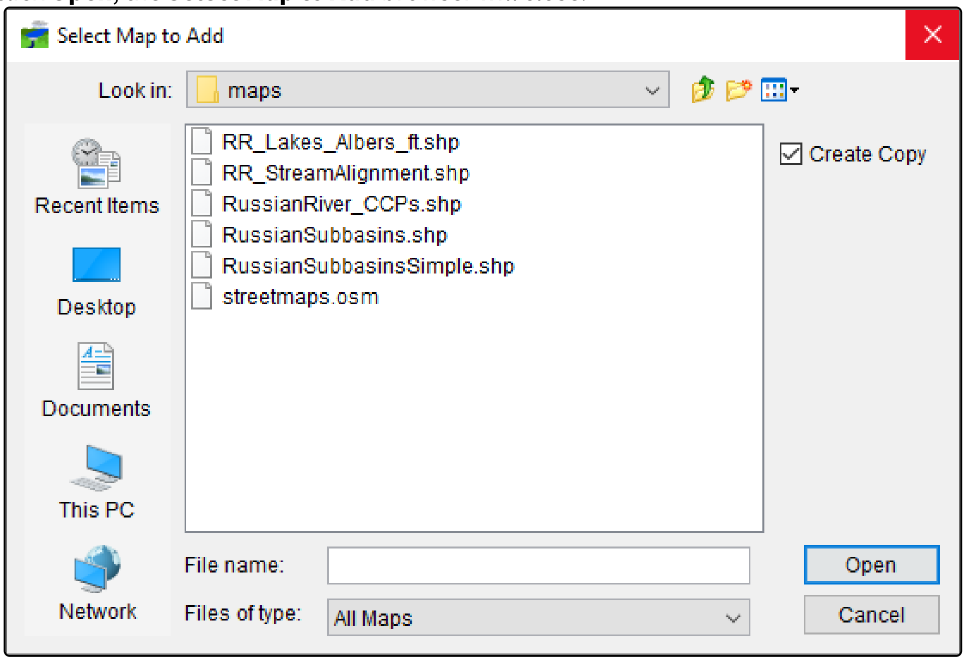
To create a watershed:

1. From the HEC-RTS main window, click on the **Setup** tab (Setup Module), from the **File** menu, click **New Watershed**. The **Create New Watershed** dialog (Figure 1) will open.

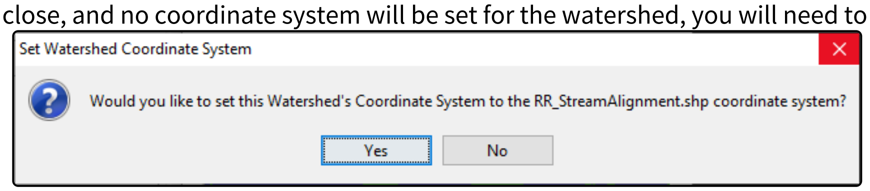


2. From the **Watershed** box (Figure 1), enter a name for the watershed. The watershed name must be less than thirty-two (32) characters and cannot contain any of the following characters: , **| / : * ? " < > -** {}.
3. Enter a description for the new watershed in the **Description** box that clearly explains what it is.
4. From the **Watershed Location** list, select the watershed location to save the new watershed or add a new location.
5. For system units, select either **English** (U.S. customary units) or **SI** (System International) from the **Unit System** list. System units cannot be changed once a watershed is created.
6. To add map layers to a watershed, click **Add Map Layers** button, a **Select Map to Add** browser (Figure 2) will open. To save the selected map layers to the watershed's *map* folder, be sure that **Create Copy** (Figure 2) has been selected. Browse to the location where map layers for the watershed are located. Select a map layer(s),

click **Open**, the **Select Map to Add** browser will close.

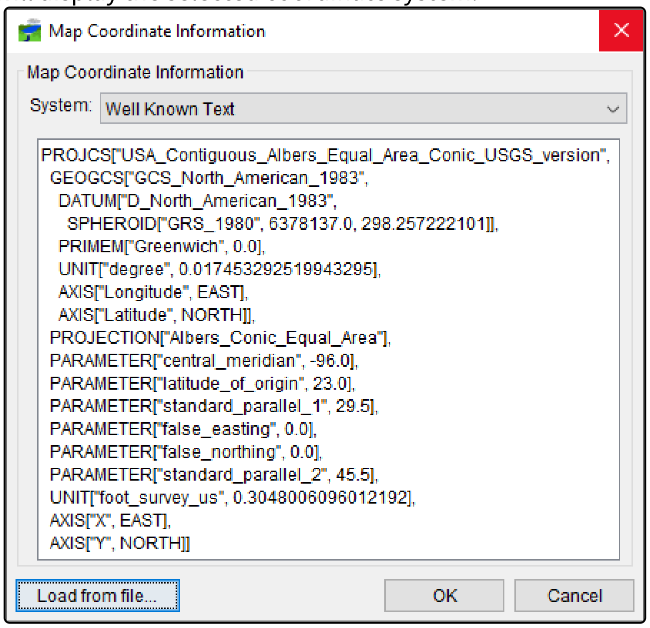


7. A **Set Watershed Coordinate System** window (Figure 3) will open. This window asks would you like to set the watershed's coordinate system to the selected map layer's coordinate system. If you click **Yes**, the **Set Watershed Coordinate System** window will close, and in the **Coordinate System** box (Figure 1), the watershed coordinate system is displayed, skip Step 8. If you click **No**, the **Set Watershed Coordinate System** window will close, and no coordinate system will be set for the watershed, you will need to do Step 8.

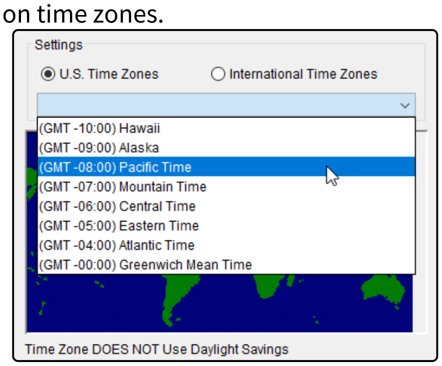


8. To set the coordinate system of the watershed, click **Edit** (Figure 1), the **Map Coordinate Information** dialog will open (Figure 4). Click **Load from file**, an **Open** browser will open. Browse to the location of watershed maps, click on a ***.prj* file that is associated with a map layer that is in the correct coordinate system. Click *Open**, the **Open** browser will close, and the **Map Coordinate Information** dialog (Figure 4) will display the coordinate system. Click **OK**, the **Map Coordinate Information** dialog will close, and the **Coordinate System** box (Figure 1)

will display the selected coordinate system.

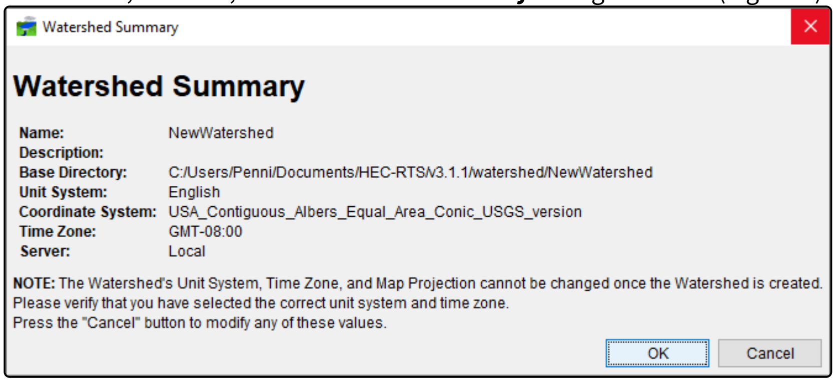


9. For the watershed time zone, by default, **U.S. Time Zones** is selected. From the list (Figure 5), there are eight choices. Select the time zone appropriate for your watershed. See Specifying Time Zones for further discussion on time zones.

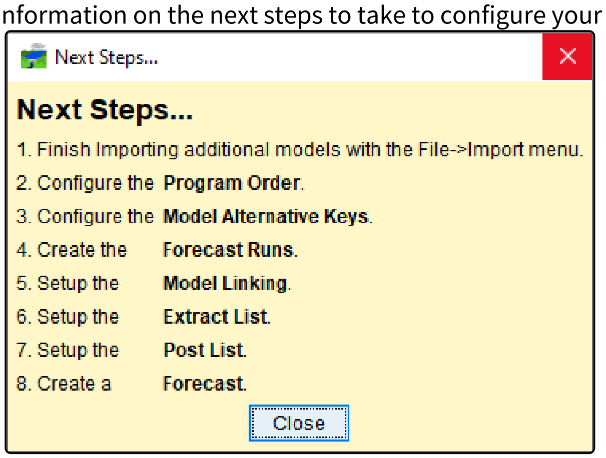


10. Click **OK**, the **Create New Watershed** dialog (Figure 1) will close. A **Watershed Summary** dialog will open (Figure 6), which will provide you with information about the watershed that is being created. Review this

information, click **OK**, the **Watershed Summary** dialog will close (Figure 6).



11. The watershed will be created and the **Next Steps** dialog (Figure 7) will open. This dialog provides with information on the next steps to take to configure your new watershed.



16.3 Import a CWMS Version 2.1 Watershed

An option of importing a CWMS Version 2.1 watershed into HEC-RTS Version 3.1.1 is available during the creation of a watershed. From the **Create New Watershed** dialog, next to the **Import CWMS 2.1 Watershed** box, click **Browse**, a **Select Workspace File** browser will open (like Select Map to Add Browser). Browse to the location of a CWMS Version 2.1 watershed, select a ***.wksp*, click *Select**. The **Select Workspace File** browser will close, from the **Create New Watershed** dialog click **OK**. The **Watershed Summary** dialog will open, click **OK**. The **Import CWMS 2.1 Watershed** dialog will open, when the import is complete, click **Close**. The watershed will be created and the **Next Steps** dialog will display. This dialog provides with information on the next steps to take to configure your new watershed. Further details on importing a CWMS watershed is provided in Model Alternatives and Forecast Runs.

16.4 Specifying Units

All modeling data and information, time series data, geometric and reservoir information must use the same system units. Therefore, it is crucial that you are certain of the units when you specify either **English** or **SI** as the unit system for your watershed when you create the watershed. The watershed system units **cannot be changed** once a watershed has been created.

16.5 Specifying Time Zones

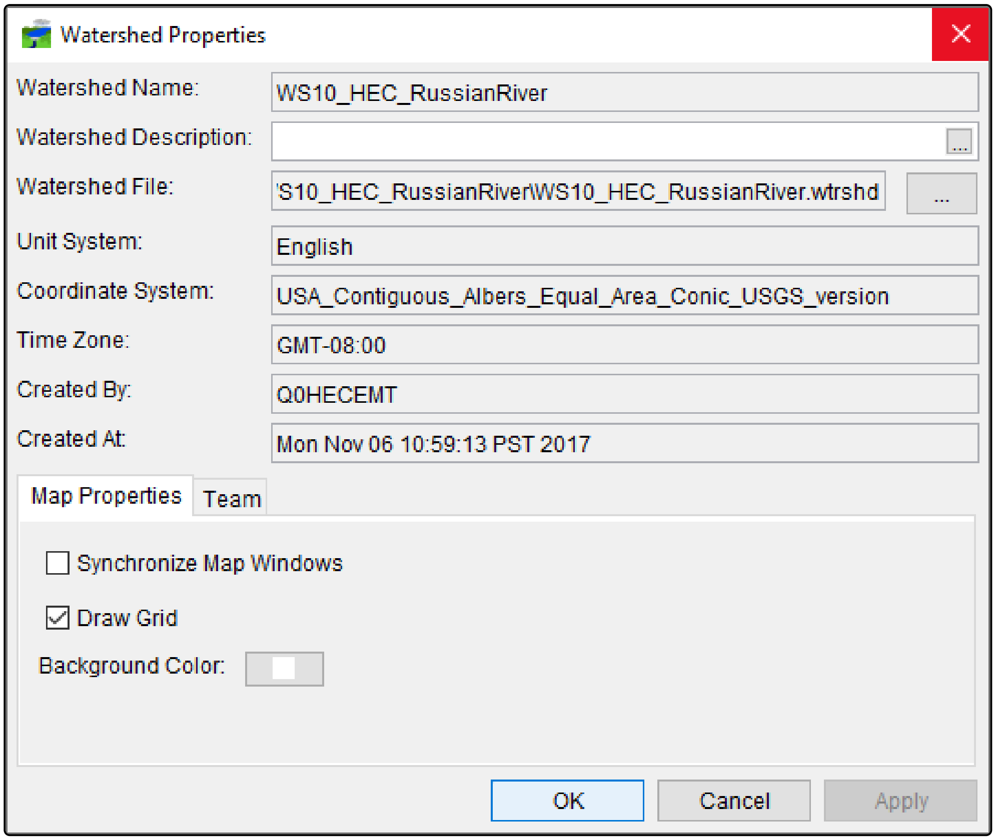
When a new watershed is configured, you must select the appropriate time zone that is representative of the watershed. This typically would be the time zone for the outlet or a major portion of the watershed. Internally, HEC-RTS will store time series data using the fixed "standard time offset from GMT" for the selected watershed time zone. Thus, internally, the data will be held in standard time only. However, when viewing data or model results, you may select local or any other time zone for display purposes (which provides standard and daylight time labels, as appropriate, throughout the year). The default time zone setting is **U.S. Time Zones**. If your watershed is outside the United States, you can change your time zone setting. In the **Create New Watershed** dialog, select the **International Time Zones**. The list will now contain time zones for the world.

NOTE: You **cannot change** the watershed time zone once a watershed has been created. However, at any time you can customize the display of data from the **View** menu (HEC-RTS Main Window).

16.6 Watershed Properties

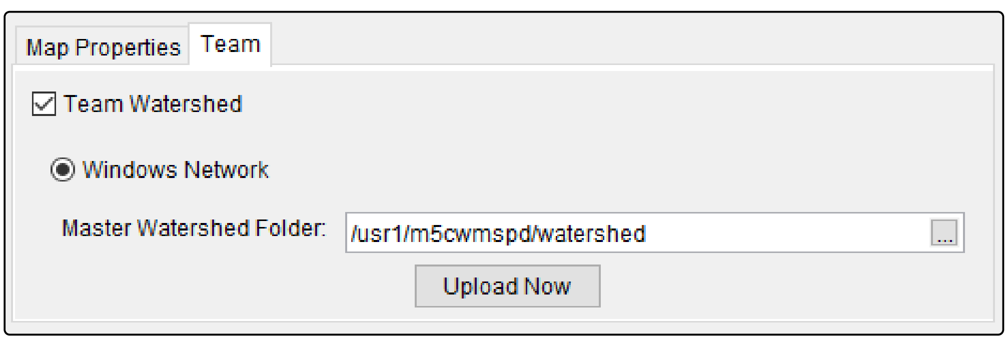
Once you have your watershed created, you can review the basic properties of the watershed. From the Setup module, from the **Watershed** menu, click **Watershed Properties**. The **Watershed Properties** dialog (Figure 1) will open. Information about the watershed is provided and includes the watershed name; watershed description; the watershed filename (*WS10_HEC_RussianRiver.wtrshd*); selected unit system; coordinate system for the watershed; watershed time zone; user who created the watershed; and, the date and time when the watershed was created.

To the right of the **Watershed File** box (Figure 1), click ..., a **Windows File Explorer** will open, displaying the directory and file structure for the watershed. The **Watershed Properties** dialog (Figure 1) also contains two tabs - **Map Properties** and **Team**. The **Map Properties** tab is selected by default, this allows you to synchronize or not synchronize map windows; to either display a grid or not on map window; and to change the background color of map windows. The default background color of map windows is white, to change the color, to the right of **Background Color** (Figure 1), click , a **Color Chooser** will open which will allow you to change the background color of map windows.



84 FIgure 1 Watershed Properties Dialog

The **Team** tab (Figure 2) is setup for the HEC-RTS Team Modeling concepts for collaboration between modelers working with the same watershed and forecast. From this tab, if you want the watershed to be available to other modelers, you will select **Team Watershed**, and define the network location of the master watershed. Further information and details about the team modeling concept in HEC-RTS is provided in Team Modeling.



85 Figure 2 Watershed Properties Dialog - Team Tab

16.7 Setup Module - Team Modeling

Team modeling facilitates collaboration between modelers working with the same watershed and forecasts. Multiple modelers may be sharing responsibility within a complex watershed containing several models. One modeler may only be responsible for the HEC-HMS model, while another modeler may be responsible for HEC-ResSim and HEC-RAS models. Team modeling allows each modeling team member to contribute expertise and participate in real-time synchronized activities. Team modeling consists of a master watershed and its associated forecasts, which are stored in a shared location (such as a networked drive). Team members share data by uploading and downloading changes to and from a master watershed, interacting with a local copy that resides on their computer. When watershed files are updated on the master watershed, team members working on the same watershed are notified of the changes by a real-time notification system. For further details on team modeling refer to Team Modeling.

16.8 Configuring a Watershed

Now that a watershed has been created you will need to configure the watershed. Configuration steps include: define the program order; create time series icons; create time series layers; import model alternatives; define keys for model alternatives; create forecast runs; setup the model linking between model alternatives in a forecast run; and, manage extract and post groups. The following sections provide further information on the configuration of your watershed.

16.8.1 Program Order

The order of program execution within HEC-RTS is a fundamental part of the watershed setup, which you define from the Setup module. You need to decide on the program order early on in your setup, and if you change the order after a forecast has been configured, then the flow of data will be disrupted, and results will be invalidated. Refer to Program Order for additional details for defining the order for application programs.

16.8.2 Developing Time Series Icons

You develop time series icons in the **Map Window** of the Setup module. Time series icons are used by the other modules to display data and information availability. The information can be time series data, images, web pages, webcams, scripts, and document files. You can link time series icons to data from DSS files. Time series icons provide you easy access to analyzing incoming data and for providing location specific information. Refer to Time Series Icons and Time Series Icons Layers for additional information describing time series icons.

16.8.3 Import Model Alternatives

There are two methods for importing model alternatives into your waters. Import a CWMS Version 2.1 Watershed describes importing an entire watershed, which includes model alternatives. From the Setup module, you can import model alternatives separately for each software application that will be used in your watershed. From the Setup module, from the **Models** menu, point to **Import**, click on the software application that you want to import a model alternative into your watershed. Another way is from the **Watershed Tree**, under the **Models** folder, right-click on the software application that you want to import a model alternative into your watershed. Refer to Model Alternatives and Forecast Runs for further details.

16.8.4 Create Model Alternative Keys

A **model alternative** is a single configuration with a specific set of input and parameters. A second model alternative would be defined by a different configuration or different input or parameters. Since model alternatives do not include time-specific data, you can use a model alternative with any time window. In the Setup module, you will create model alternative keys that will be used in creating a forecast run. Each key is a single alpha-numeric identifier that represents a model alternative. Refer to Model Alternatives and Forecast Runs for additional information.

16.8.5 Create Forecast Runs

A **forecast run** is a single forecast scenario defined by a specific set of data, information, and model alternatives. You create forecast runs in the Setup module. You use the previously defined model alternative keys to create a forecast run key that consists of a collection of the model alternative keys. Refer to Model Alternatives and Forecast Runs for additional information.

16.8.6 Link Models Alternatives

After a forecast run has been created you need to link the models, which is accomplished through the **Model Linking Editor**. From this editor you select a forecast run you wish to configure. Then you select each model alternative that is associated with the selected forecast run. For each model alternative you will determine where you will be getting data from another model alternative or from an extract list. Refer to Model Linking Editor for additional information.

16.8.7 Data Extract and Posting

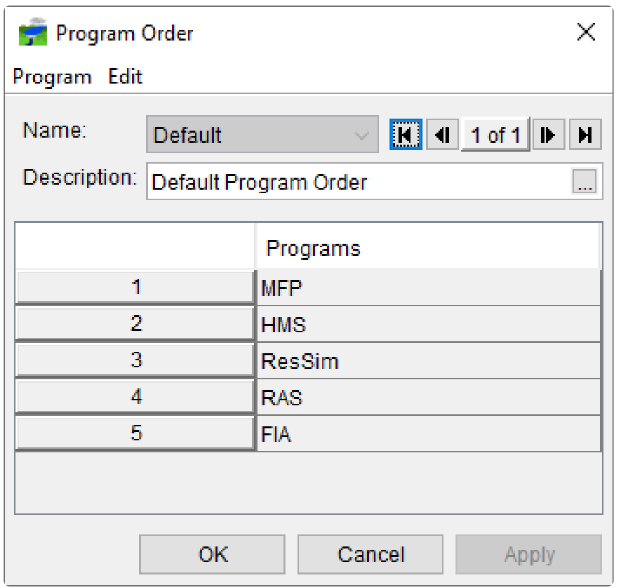
Once the models have been linked the next step is to configure the data extract groups. A data extract group specifies the location of the gridded and time series data required for the different model alternatives. For example, this could be gridded precipitation data in DSS or observed flow data from the HEC-RTS main DSS file. Once you have completed computing your forecast, you might want to post results back to the HEC-RTS main DSS file. This task is completed by creating post groups that specify what time series data needs to be save. Refer to Data Extract and Post Editors for additional information.

17 Program Order

HEC-RTS allows for a flexible set of software applications (programs), computation order, and data flow for forecasting. The order of execution of software applications is a fundamental part of the watershed setup. You can also define supplemental programs and add them to the computation order so that they are executed from HEC-RTS. The program order is defined in the Setup module. This chapter provides details on the program order.

17.1 Establishing Program Order

HEC-RTS generates a default program order (Figure 1) that contains HEC software applications. If you change the order of programs after a forecast run has been configured, you may disrupt the orderly flow of data and the results and invalidate any existing work. For HEC-RTS Version 3.1.1, during the creation of a forecast run, the only program order that you can select is the default program order.



17.2 Program Order Dialog

To change, delete, or add software applications to a watershed's program order, you use the **Program Order** dialog. The **Program Order** dialog also establishes computation order and data flow for forecasting. In addition, the **Program Order** dialog provides the mechanism for defining supplemental programs, adding the supplemental programs to the program order, and establishing data flow for supplemental programs. To access the **Program Order** dialog, from the HEC-RTS main window, click on the **Setup** tab, from the **Model** menu, click **Program Order**. The **Program Order** dialog will open.

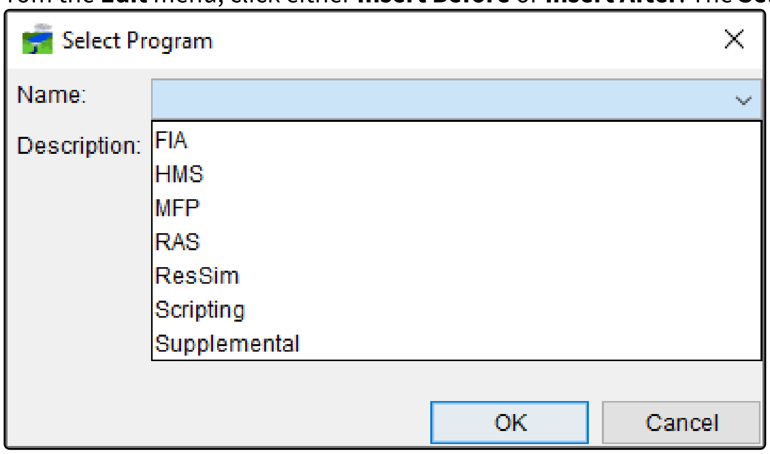
17.3 HEC Software Applications in the Program Order

By default, the program order for a watershed is: **MFP**, **HMS**, **ResSim**, **RAS**, and **FIA**. Additional software applications or plugins may also be added or removed from the program order, along with additional HEC software applications.

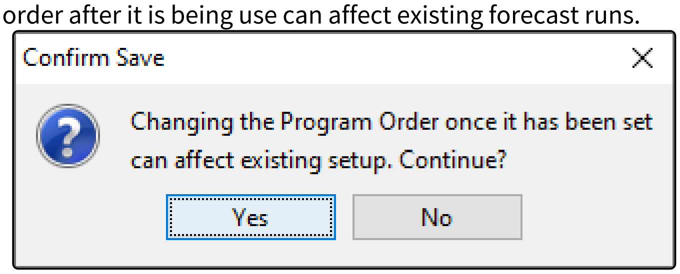
17.3.1 Add an HEC Program

The default program order is displayed in the table on the **Program Order** dialog. To add an HEC software application:

1. Click a row on the table of the **Program Order** dialog where you would like to add the program.
2. From the **Edit** menu, click either **Insert Before** or **Insert After**. The **Select Program** dialog opens (Figure 1).



3. From the **Name** list, select a program, click **OK**. The **Select Program** dialog will close, and the selected software application will display in location you selected in the table.
4. Click **Apply**, a **Confirm Save** window will display (Figure 2). This window is warning you that changing a program order after it is being use can affect existing forecast runs.



5. Click **Yes**, the **Confirm Save** window will close. You now have added a software application to the program order.

17.3.2 Remove an HEC Program

To remove an HEC program from the program order:

1. Click a row on the table of the **Program Order** dialog where you would like to remove a program.
2. From the **Edit Menu**, click **Remove**. A **Confirm Removal** window will display asking you if it is okay to remove the program from the program order. Click **Yes**, the **Confirm Removal** window will close.

3. The selected row has been deleted from the table. A **Confirm Save** window will display (Figure 2). This window is warning you that changing a program order after it is being use can affect existing forecast runs.
4. Click **Yes**, the **Confirm Save** window will close. You now have removed a software application from the program order.

17.4 Supplemental Program in the Program Order

You can define supplemental programs that can be added to the program order. A **supplemental program** is a software application that is not included with HEC-RTS. A supplemental program is usually something that has been created by you or your office for use in water control management forecasting. For example, an office might have a locally developed software application that is used to model the unsteady flow regime of one or more rivers. This software application, and the river model associated with it, could be used in addition to or in place of HEC-RAS in the program order.

17.4.1 Supplemental Program Interface

A supplemental program must adhere to a specific interface for communicating with HEC-RTS during the execution of a forecast. This interface includes how to receive arguments for the supplemental program from HEC-RTS, how to output messages to the forecast compute log, and how to inform HEC-RTS about the exit status of the supplemental program.

17.4.1.1 Receiving Arguments

During a forecast computation, HEC-RTS executes a supplemental program by running its executable with a single command line parameter. The parameter is the fully qualified name of a file that contains the arguments for the supplemental program. This arguments file contains standard HEC-RTS arguments in *keyword*=*value* format – one keyword/value pair per line – plus any alternative-specific arguments defined for the supplemental program model alternative being executed.

The standard HEC-RTS argument keywords available in the arguments file are listed:

ComputeType	Type of computation, "Forecast" for HEC-RTS
ProgramName	Supplemental program name
ProgramAlternative	Supplemental program model alternative name
ModelAlternative	Model alternative identifier
Alternative	Forecast run identifier
RunTimeWindow	Lookback, forecast, and end times of forecast

IOTimeStep	Interval of data in a forecast DSS file
TimeZoneOffset	Offset in hours from UTC to watershed time zone
InputPosition	Number of characters contained in the **InputFPart** name
ModelPosition	Number of characters contained in the **OutputFPart** name
ForecastPath	Forecast directory of current forecast
ForecastDSSFile	Fully qualified name of forecast HEC-DSS file
InputFPart	The F pathname part of data to be read from the forecast HEC-DSS file
OutputFPart	The F pathname part of data to be written to the forecast HEC-DSS file
User	The name of the HEC-RTS user executing the forecast
ClientDisplay	The X-Windows server address of the user's display
ForceRecompute	"true" or "false" specifying whether the supplemental program must be executed – if "false", the supplemental program can decide whether it should execute

An example of a **Supplemental Program Arguments File** is shown in Figure 1.

17.4.1.2 Outputting Messages to the Forecast Compute Log

The supplemental program architecture allows the executable to output normal messages and error messages to the forecast compute log by writing these messages to a supplemental program log file.

```
ComputeType=Forecast
ProgramName=SupplementalTest
ProgramAlternative=NormalTest
ModelAlternative=20D0N0X0------
Alternative=20D0N0X0------
RunTimeWindow=26 August 2006, 24:00; 30 August 2006, 03:00; 3September
2006, 24:00
IOTimeStep=1 Hour(s)
TimeZoneOffset=-5 Hour(s)
InputPosition=4
ModelPosition=6
ForecastPath=/usr1/q0cwmsts/watershed/2006.08.30-0300/BaldEagle
ForecastDSSFile=/usr1/q0cwmsts/watershed/2006.08.30-
0300/BaldEagle/forecast.dss
InputFPart=20D0
OutputFPart=20D0N0
User=q0hecmdp-hec65.64201#1
ClientDisplay=153.77.12.12:0
ForceRecompute=false
```

The supplemental program log file:

- Has the same name as the arguments file name (passed as the only command line parameter to the supplemental program executable) with the "**.args**" extension replaced by "**.log**".
- May contain special message type lines that instruct HEC-RTS how to process the following lines:
    - **[No Message]** specifies that the following lines will not be included in the compute log
    - **[Log Message]** specifies that the following lines will be treated as a normal compute log message
    - **[Error Message]** specifies that the following lines will be treated as a compute log error message
- The message type lines are case insensitive, but must include the brackets ({*}\[* and *\]{*}) without any adjoining whitespace and may not appear on a line with any other text. The message type remains in force until another message type line is encountered. An implicit *\[No* *Message\]* line is processed at the beginning of the log file, so text in the log file before a message type line will not be included in the compute log.

17.4.1.3 Informing HEC-RTS of the Exit Status of a Supplemental Program

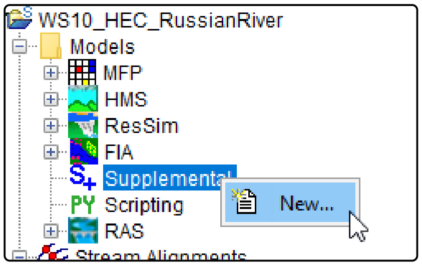
HEC-RTS uses the exit code returned from the executable to the operating system to determine the status of the supplemental program execution. The exit code is interpreted as follows:

0	The supplemental program completed successfully
1	The supplemental program skipped computation - the executable should not return this value if the **ForceRecompute** argument is set to "true"
other	An error occurred during execution - the executable should output some diagnostic message(s) to the compute log if it returns a value other than 0 or 1

17.4.2 Create a Supplemental Program

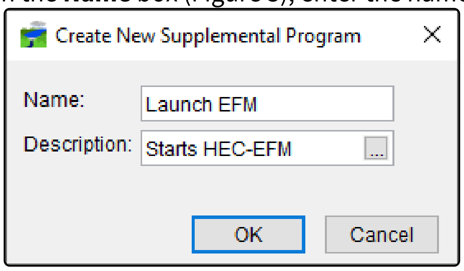
To create a supplemental program:

1. From the **Watershed Tree** (Figure 2); right-click **Supplemental**, from the shortcut menu, click **New**. The **Create New Supplemental Program** dialog opens (Figure 3).

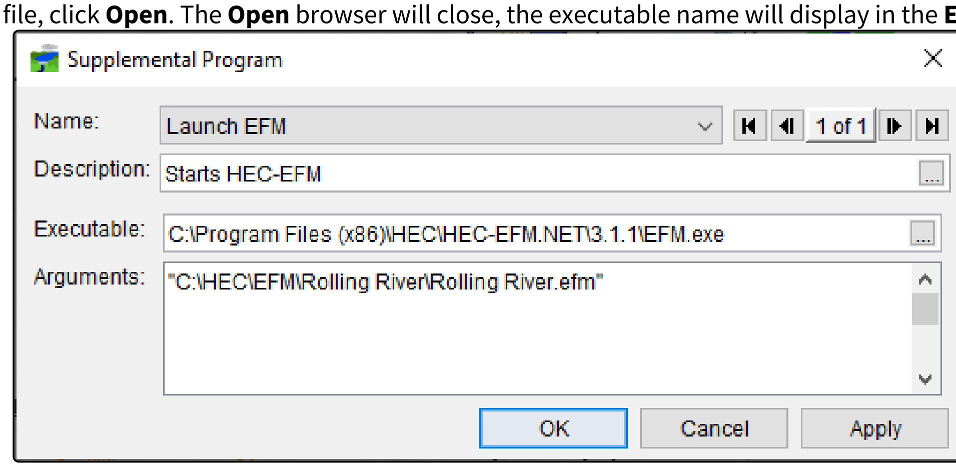


86 Figure 2 Watershed Tree - Models Folder - Supplemental Shortcut Menu

2. In the **Name** box (Figure 3), enter the name of the supplemental program.



3. Enter a description in the **Description** box and click **OK**.
4. The **Supplemental Program** dialog (Figure 4) opens. In the **Supplemental Program** dialog, specify the location of the software application that implements the supplemental program in the **Executable** box or click  . An **Open** browser will open, browse to the location of the software application's executable, select the appropriate file, click **Open**. The **Open** browser will close, the executable name will display in the **Executable** box (Figure 4).



87 Figure 4 Supplemental Program Dialog

5. If needed, enter command line arguments in the **Arguments** box (Figure 4). Click **OK**, the **Supplemental Program** dialog closes. From the **Watershed Tree**, from the **Models** folder, expand **Supplemental**, the name of the supplemental program will display.

17.4.3 Add a Supplemental Program - Program Order

Once you have defined a supplemental program, you can add it to the program order. To do this, follow the instructions for adding an HEC program in HEC Software Applications in the Program Order. Instead of selecting an HEC program, you will select the supplemental program that you created and the appropriate input for the program.

17.4.4 Remove a Supplemental Program - Program Order

The steps to remove a supplemental program from the program order are the same as for removing an HEC program (HEC Software Applications in the Program Order).

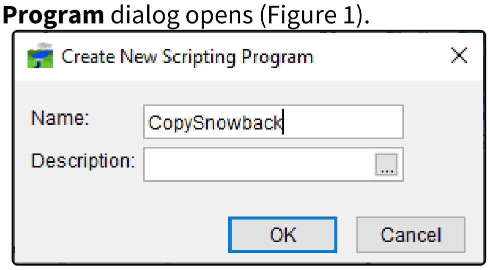
17.5 Scripting in the Program Order

If you want to add a script to the computing sequence you must add the script to the program order.

17.5.1 Create a Scripting Program

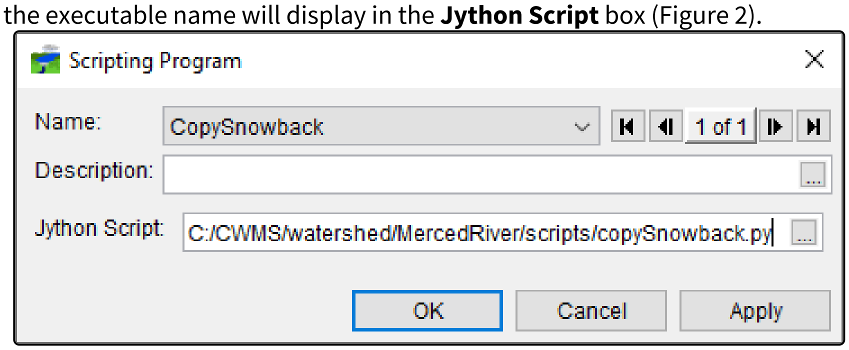
To create a scripting program:

1. From the **Watershed Tree**; right-click **Scripting**, from the shortcut menu, click **New**. The **Create New Scripting Program** dialog opens (Figure 1).



88 Figure 1 Create New Scripting Program Dialog

2. In the **Name** box (Figure 1), enter the name of the script.
3. Enter a description in the **Description** box and click **OK**.
4. The **Scripting Program** dialog (Figure 2) opens. In the **Scripting Program** dialog, specify the location of the script that implements the scripting program in the **Jython Script** box or click ... . An **Open** browser will open, browse to the location of the script (***.py*), select the script file, click *Open**. The **Open** browser will close, and the executable name will display in the **Jython Script** box (Figure 2).



17.5.2 Add a Scripting Program – Program Order

Once you have defined a supplemental program, you can add it to the program order (Program Order Dialog). To do this, follow the instructions for adding an HEC program in HEC Software Applications in the Program Order. Instead of selecting an HEC program, you will select the supplemental program that you created and the appropriate input for the program.

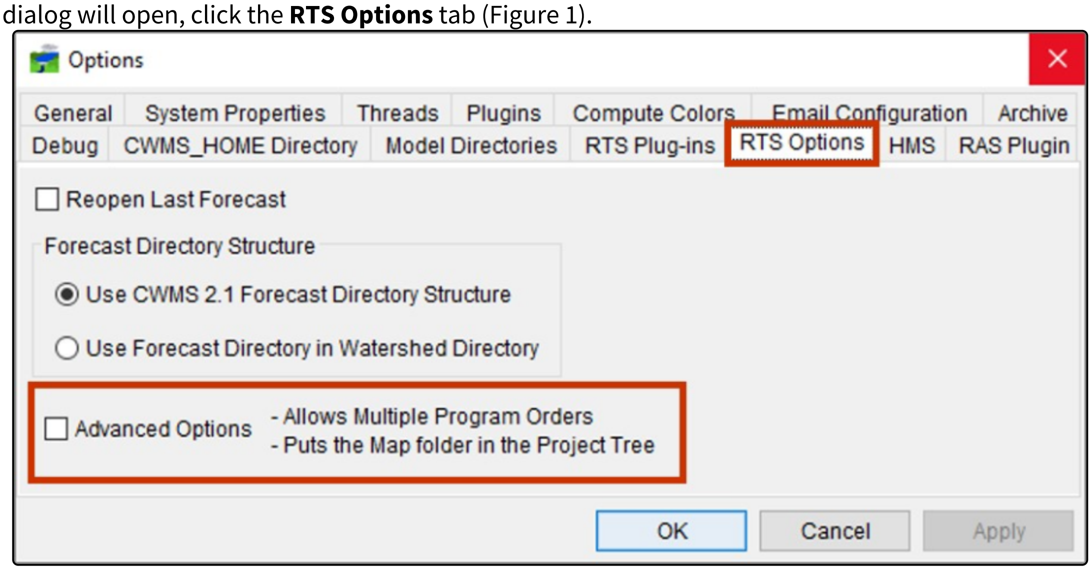
17.5.3 Remove a Scripting Program – Program Order

The steps to remove a supplemental program from the program order are the same as for removing an HEC program (HEC Software Applications in the Program Order).

17.6 Multiple Program Orders

By default, HEC-RTS allows only a single program order to be used. If desired, the user can enable the use of multiple program orders (RTS Options Tab). To enable multiple program orders:

1. From the HEC-RTS main window, click on the **Setup** tab, from the **Tools** menu, click **Options**. The **Options** dialog will open, click the **RTS Options** tab (Figure 1).



89 Figure 1 Options Dialog - RTS Options Tab - Advanced Options

2. On the **RTS Options** page (Figure 1), **Advanced Options** is not selected. To enable **Multiple Program Orders,** select **Advanced Options** (this also adds the **Map** folder to the **Watershed Tree**.
3. Click **OK**, the **Options** dialog (Figure 1) will close, and now you will be able to define multiple program orders for your watershed.

Care must be exercised when using multiple programs orders. When a model alternative is linked to a program for its input information, it remembers the position of the model it is receiving its data from, not the name or program of the specified model alternative. This can lead to potential conflicts if a model alternative is used in two different program orders. The recommended practice when using multiple programs orders is to configure a different model alternative for use in each different program order.

18 Time Series Icons

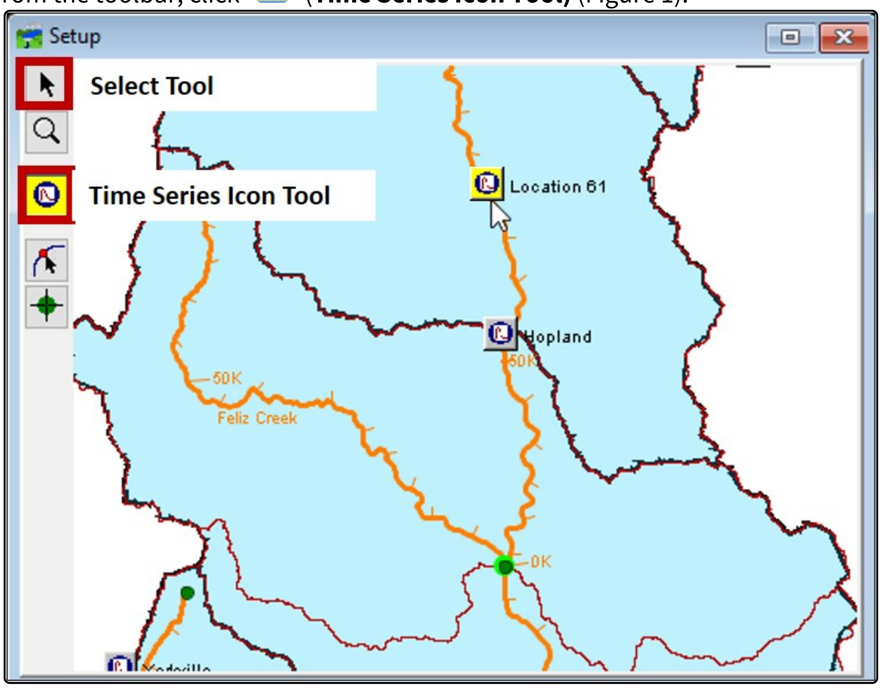
Time series icons are displayed in the **Map Window** of the HEC-RTS main window and indicate sites for which data and information are available. The information can be time series data, images, websites/webpages, webcams, scripts, spreadsheets, or other files. Time series data for the icons can be linked to data from DSS files. Time series data can appear as thumbnail plots or color bars. The icons provide easy access for analyzing data and providing location-specific information (e.g., frequently updated pictures of gages, reports on the location).

In the **Setup** module, you will use the **Time Series Icon Tool** to create and manipulate time series icons. Map Window provides a description of the **Time Series Icon Tool**.

18.1 Create a Time Series Icon

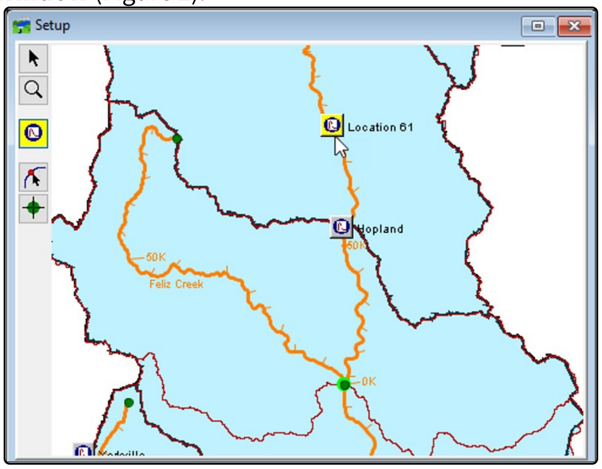
To create a time series icon:

1. From the HEC-RTS main window, open a watershed, click the **Setup** tab, from the **Map Window** that displays, from the toolbar, click (**Time Series Icon Tool)** (Figure 1).



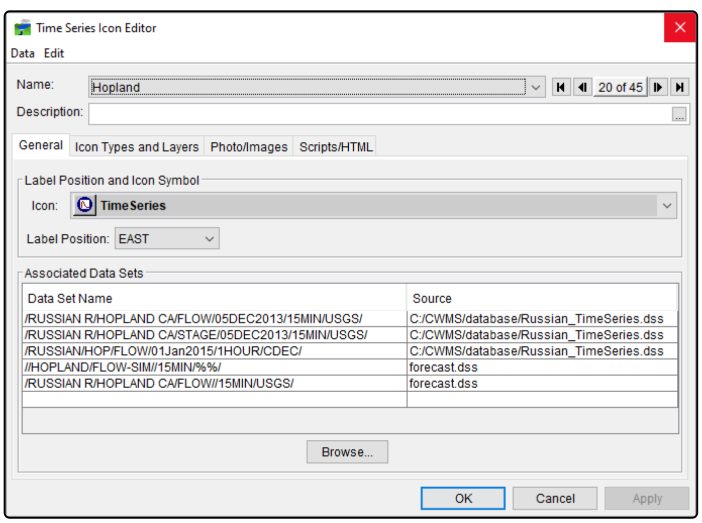
2. Find a location on the **Map Window** where you want to place a time series icon (Figure 2), hold down the **CTRL** key, and click the left mouse button. A time series icon with a default name (*Location 61*) will display on the **Map**

Window (Figure 2).



18.2 Time Series Icon Editor

After you have defined a new time series icon, you need to configure/edit the properties of the time series icon. From the **Time Series Icon Editor** dialog (Figure 1), properties include the name of the time series icon (view only), description, type of time series icon, position of the time series icon label, the location and the data associated with the time series icon, and the appearance of the time series icon in the different HEC-RTS modules. In addition, you can associate images, websites, webcams, documents, and scripting information with a time series icon.



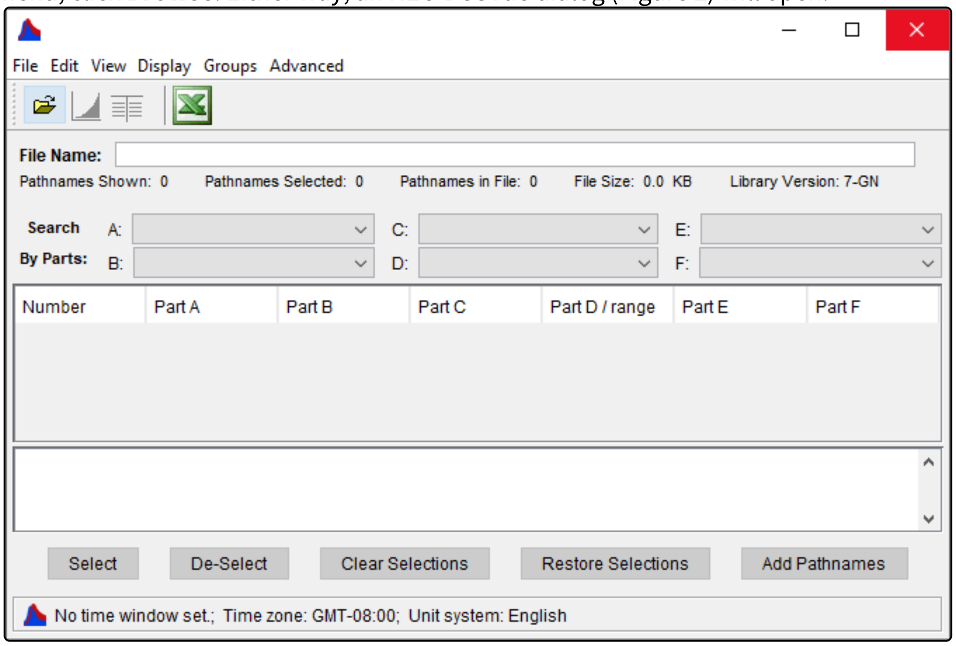
90 Figure 1 Time Series Icon Editor

18.2.1 Associate Data with a Time Series Icon

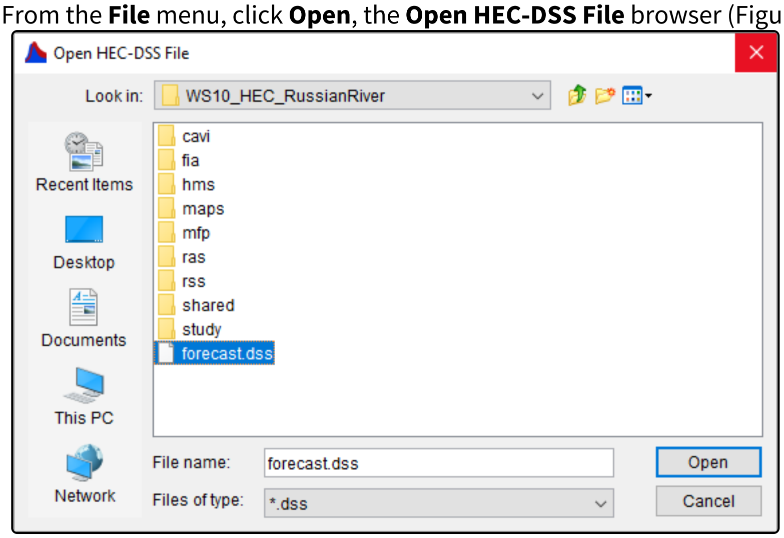
Within HEC-RTS, time series data can be accessed from DSS files. You can either enter the pathnames (time series identifiers) associated with the data or select the time series identifiers from a DSS file. Before selecting time series identifiers from a DSS file, you need to make sure that you have a DSS file available. A recommended choice would be the DSS file that is created when an HEC-RTS forecast is computed (*forecast.dss*).

To select time series identifiers from a DSS file:

1. From the **Time Series Icon Editor** (Figure 1), be sure the **General** tab is selected, click **Browse**, or from the **Data** menu, click **Browse**. Either way, an HEC-DSSVue dialog (Figure 2) will open.

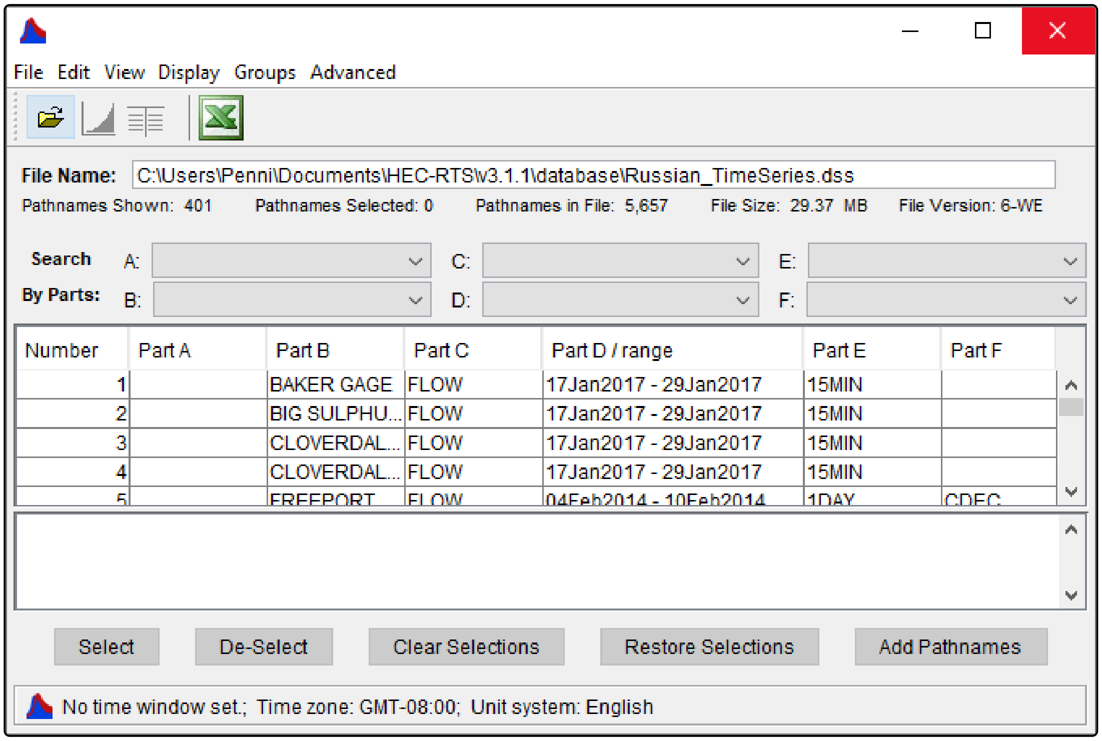


2. From the **File** menu, click **Open**, the **Open HEC-DSS File** browser (Figure 3) will open.



3. Browse to the location of the appropriate DSS file, click on the DSS file, click **Open**, the **Open HEC-DSS File** browser (Figure 3) will close. The table on the HEC-DSSVue dialog (Figure 4) will now contain the time series

identifiers that are available from the selected DSS file.

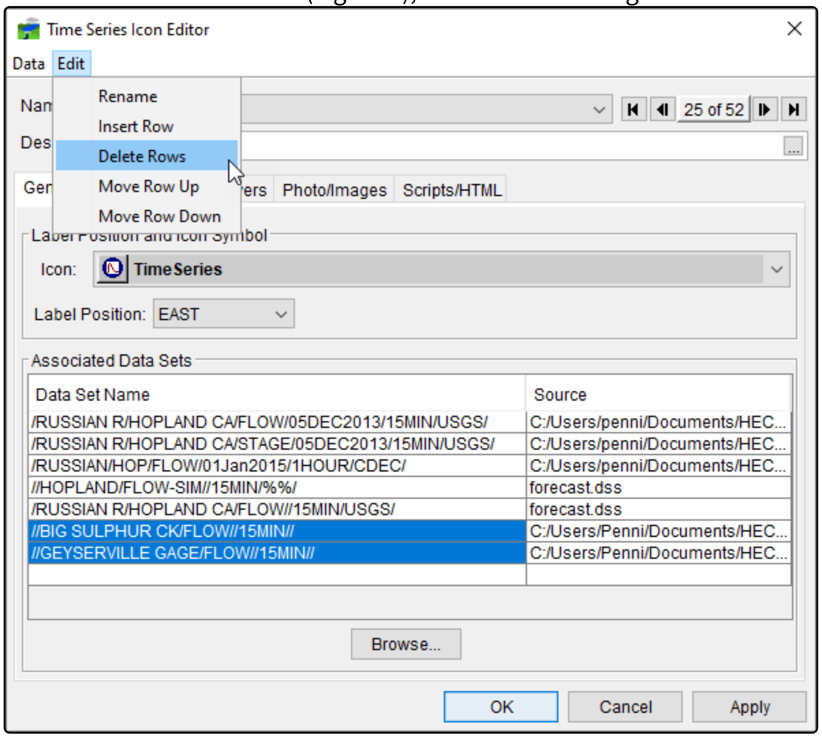


4. Click on a time series identifier in the table that contains data that you want to associate with the time series icon. Click **Select** (Figure 4), the time series identifier will display in the selection list at the bottom of the HEC-DSSVue dialog. Continue adding time series identifiers by selecting them in the table and clicking **Select**. You can also double-click on a time series identifier to add it from the selection list. Another way is to hold down the **CTRL** key and select multiple time series identifiers from the table.
5. You might want to use the **Search By Parts** (Figure 4) lists to aide in searching for specific time series identifiers. There are six lists that are available, that will aid you in the filtering of time series identifiers. Refer to DSS Pathnames for further details on this feature of HEC-DSSVue.
6. Once you have chosen your time series identifiers, click **Add Pathnames** (Figure 4), the **Time Series Icon Editor** dialog (Figure 1) will display the time series identifiers you have selected in the **Associated Data Sets** box (Figure 1). The **Source** column of the table (Figure 1) will display information about the selected DSS file. When you are finished selecting time series identifiers, from the HEC-DSSVue dialog (Figure 4), from the **File** menu, click **Close,** the HEC-DSSVue dialog will close.
7. The **Time Series Icon Editor** (Figure 1) now displays what data has been associated with the selected time series icon.

18.2.2 Remove Data from a Time Series Icon

To remove data from a time series icon:

1. From the **Time Series Icon Editor** dialog (Figure 5), be sure the **General** tab is selected. Select a row(s) in the **Associated Data Sets** table (Figure 5), that should no longer be associated with the selected time series icon.



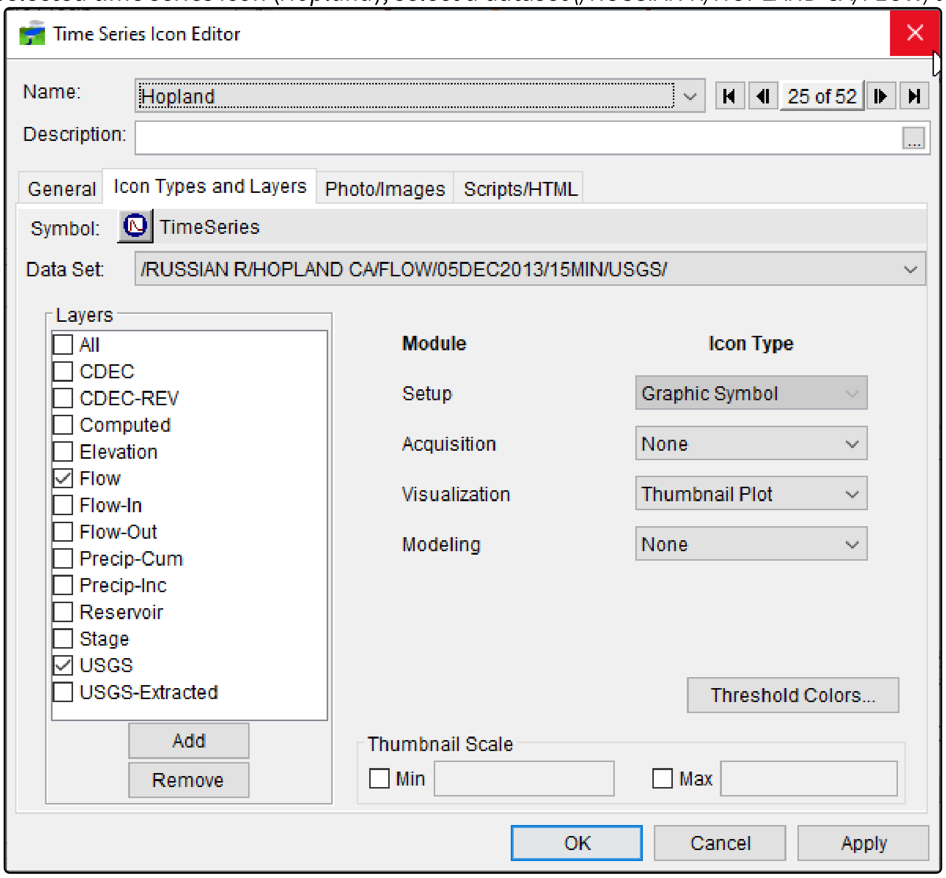
2. From the **Edit** menu (Figure 5), click **Delete Rows**. The selected rows will be deleted from the **Associated Data Sets** table, and the data will no longer be associated with the time series icon.

18.2.3 Icon Types and Layers Tab

Now that you have created time series icons, HEC-RTS provides you with two ways to organize time series icon, for display purposes. The first is the concept of time series icon layers. By default, HEC-RTS creates an "**All**" time series icon layer, and by default, time series icons are assigned to the "**All**" time series icon layer when created. Time series icon layers make refining the display of data faster by allowing selection of the displayed layers by their data types. From HEC-RTS, you can create your own time series icon layers. For example, a gage may report stage, precipitation, and compute flow from stage. If time series layers are created for each one, "Stage", "Flow", and "Precipitation", then the appropriate datasets for that location can be placed in a corresponding time series icon layer. The procedure to create time series icon layers is described in Time Series Icons Layers.

The following steps describe how to assign datasets to time series icon layers:

1. From the **Time Series Icon Editor**, click the **Icon Types and Layers** tab (Figure 6). From the **Data Set** list, for the selected time series icon (*Hopland*), select a dataset *(/RUSSIAN R/HOPLAND CA/FLOW/05DEC2013/15MIN/USGS/*).



2. From the **Layers** box (Figure 6), is a list of time series icon layers. Select and de-select times series icon layers for the selected dataset *(/RUSSIAN R/HOPLAND CA/FLOW/05DEC2013/15MIN/USGS/*) by clicking on the check boxes. Repeat this step for all datasets, that you want assigned to time series icon layers.

Now that you are finished with assigning time series icon layers, you now need to setup how the time series icon will be display in the individual HEC-RTS modules (Acquisition, Visualization, Modeling). There are seven available icon types:

 **Graphic Symbol** - provides an at-a-glance reference to the type of location represented and allow direct access to data associated with a location (such as model results). This is the only icon type for the **Setup** module.

 **Dot** - allows you to mark gages and time series locations without cluttering the map window. Selecting this option displays a small red dot along with the icon label ( ) in the map window.

 **Thumbnail Plot** – is a miniature version of the full-size plots available in HEC-RTS, showing data for the duration of the time window. As the time window updates, HEC-RTS refreshes the data and updates the plot. Thumbnail plots offer a quick overview of current data for the selected time window.

 Quality Color Bar - are linear histogram that offers a quick view of the quality of data within a set time window. Quality color bars reflect the quality of data by displaying bands of color that display quality information for datasets that are configured for this icon type. Refer to Reviewing Data Quality for additional information regarding quality color bar configuration.

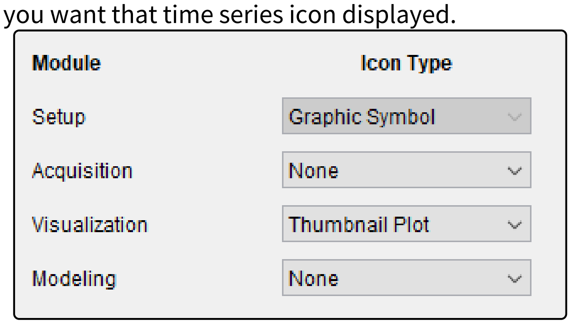
Threshold Color Bar - are linear histograms that compare data against threshold values set for a location. For example, at a "Flow" location, green might be assigned to flow values less than 5,000, which represent normal flow conditions. Yellow may represent values greater than 5,000, but less than 7,000 to indicate flows in a flood warning level, and red may represent values greater than 7,000 to indicate flows above flood stage. Because threshold color bars are location dependent, you must set the values for each time series icon individually. See Visualizing Data on how to configure your threshold color bars.

Cumulative Threshold Color Bar - are color bars that display accumulated values. For example, display precipitation over time or the sum of volumes of inflows to a lake are accumulated values.

None - by selecting **None**, you can configure the time series icon not to display in a specific HEC-RTS module (Acquisition, Visualization, Modeling). This is useful when you only use a time series icon in a specific module, as it keeps the map window uncluttered with unnecessary time series icons.

To assign a dataset to an HEC-RTS module (Acquisition, Visualization, Modeling) and select an icon type:

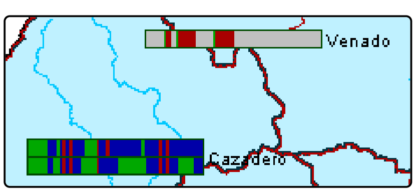
1. From the **Time Series Icon Editor**, click the **Icon Types and Layers** tab (Figure 6). From the **Data Set** list, for the selected time series icon (*Hopland*), select a dataset *(/RUSSIAN R/HOPLAND CA/FLOW/05DEC2013/15MIN/USGS/*).
2. From the module and icon type assignment section (Figure 7), for the selected dataset you need to determine if you wish to view the time series icon in that module. In addition, if you want the time series icon displayed how you want that time series icon displayed.



3. For example, in Figure 7, for the selected time series icon and dataset, for the **Acquisition** module, the time series icon will not be displayed, as you have selected an icon type of **None**. For the **Visualization** module, the time series icon will be display as a **Thumbnail** plot. Finally, for the **Modeling** module, the time series icon will not be displayed, as you have selected an icon type of **None**.

18.2.3.1 Configuring a Threshold Color Bar for a Dataset

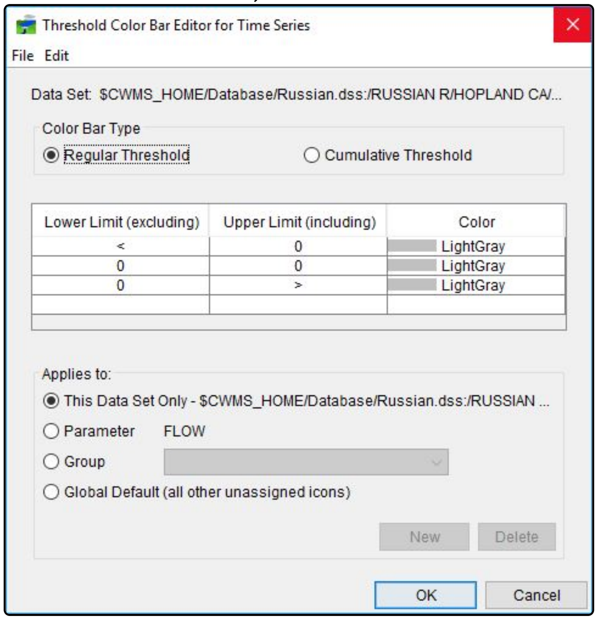
Threshold color bars (Figure 8) are linear histograms that compare incoming data against threshold values set for a location. Because a threshold color bar dataset is dependent, you typically set the color values for the dataset of a time series icon. The length of the threshold color bar represents a timeline equal to the duration of the time window. Segments along the color bar reflect time intervals, with colors corresponding to the data quality legend you have defined. The threshold color bars are updated dynamically at the time interval that has been set for the time window.



91 Figure 8 Threshold Color Bars

To set the thresholds and colors a threshold color bar associated with a dataset:

1. From the **Time Series Icon Editor**, click the **Icon Types and Layers** tab (Figure 6). From the **Data Set** list, for the selected time series icon (*Hopland*), select a dataset *(/RUSSIAN R/HOPLAND CA/FLOW/05DEC2013/15MIN/USGS/)*.
2. Click **Threshold Colors**, the **Threshold Color Bar Editor for Time Series** will open (Figure 9).



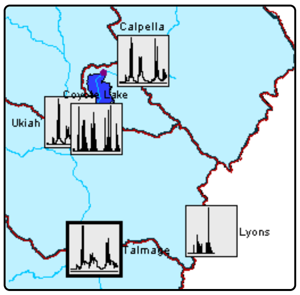
3. The dataset's name is displayed on the **Threshold Color Bar Editor for Time Series**. The default threshold color type is **Regular Threshold** (Figure 9), for **Cumulative Threshold** color bars you would use the **Threshold Color Bar Editor for Time Series** to create your threshold color bar.
4. HEC-RTS creates a default threshold for you. For example, the selected dataset is a flow location, you need to set **DarkGreen** (Figure 9) to flow values less than 5000 (normal flow conditions). From the table, for the **DarkGreen** row, under the **Upper Limit** column, double-click on the cell, enter *5000* (do not enter a comma).The entered value is displayed for the **DarkGreen** row, and for the **DarkBlue** row 5000 is now displayed in the **LowerLimit** column of the table.
5. Now you need to set your next threshold limit for the dataset. For the **DarkBlue** (Figure 9) row, we want the color to be yellow. Click on **DarkBlue**, from the list select **Yellow**. For the **Yellow** row, in the **UpperLimit** cell, enter *7000*. Values greater than 5000 but less than 7000 indicate flows that are in a flood warning level.
6. For the **DarkRow** (Figure 10), you have a threshold of flow values greater than 7000 indicating that flows for this dataset are above flood stage.

Lower Limit (excluding)	Upper Limit (including)	Color
<	5000	DarkGreen
5000	7000	Yellow
7000	>	DarkRed

7. By default, HEC-RTS will use this as the threshold color scheme (Figure 10) for datasets with the parameter of flow (**Parameter**). You can apply this threshold color scheme to the selected dataset only, to a group, or to all other unassigned time series icon (**Global Default**).
8. Click **OK**, the **Threshold Color Bar Editor for Time Series** will close (Figure 9).

18.2.3.2 Configuring Thumbnail Plots – Thumbnail Scale

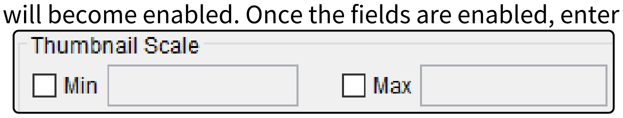
A thumbnail plot (Figure 11) is a miniature version of the full-size plots available in HEC-RTS, showing data for the duration of the time window. You can adjust the size of the thumbnail plot.



92 Figure 11 Thumbnail Plots - Visualization Module

To set the scale of a thumbnail plot:

1. From the **Time Series Icon Editor**, click the **Icon Types and Layers** tab (Figure 6). From the **Data Set** list, for the selected time series icon (*Hopland*), select a dataset *(/RUSSIAN R/HOPLAND CA/FLOW/05DEC2013/15MIN/USGS/)*.
2. The **Thumbnail Scale** box (Figure 12) represents the minimum and maximum values displayed with in a thumbnail plot. To set these values, first check the checkbox next to **Min** or **Max**, and the fields next to the labels will become enabled. Once the fields are enabled, enter the numeric values for the items you selected.



3. Click **OK**, the **Time Series Icon Editor** dialog (Figure 1) will close. To save the changes to the watershed, from the HEC-RTS main window, from the **File** menu, click **Save Watershed**.

18.2.4 Photo/Images Tab

Photos, webcam images, web pages, or documents can be assigned to time series icons. By adding them to a time series icon in the **Setup** module, you can access them in the **Visualization, Acquisition**, and **Modeling** modules through the icon shortcut menu. Images and webcams can be viewed in their own separate dialog or as icons on the map window. For more information on how to set these features up see Time Series Icons.

18.2.5 Scripts/HTML Tab

Scripts can be assigned to individual time series icons. The scripts can be used to update the data at given intervals, compute other information from the data, or complete other tasks. Scripts are assigned to the icons in the **Setup** module, through the **Time Series Icon Editor**. For more detailed information on assigning and using scripts, see Scripting.

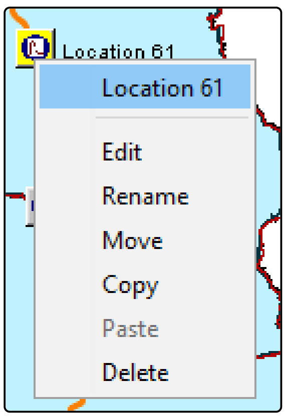
18.3 Time Series Icons Shortcut Menu

You can move, delete, rename, and add datasets to time series icons in the Setup module. You can also organize time series icons for display purposes. All these tasks are accomplished through the time series shortcut menu (Figure 1).

From the **Setup** module, from a **Map Window**, select the **Time Series Icon Tool** ( ) or the **Select Tool** ( ), right-click on a **Time Series Icon**, a shortcut menu will display (Figure 1).

18.3.1 Move a Time Series Icon

To move a time series icon:



93 Figure 1 Time Series Icon Shortcut Menu

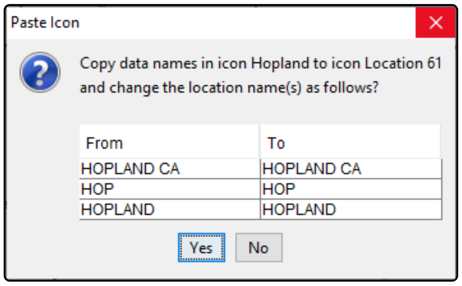
1. From the **Setup** module, from a **Map Window**, select the **Time Series Icon Tool** ( ) or the **Select Tool** ( ), from the shortcut menu (Figure 1), click **Move**.
2. The time series icon now has four little black dots (handles) at each corner. Click and drag the icon to its new position.

18.3.2 Copy/Paste Data from Icon to Icon

You can copy all the information associated with a time series icon to another time series icon. This is useful for setting up similar time series icons where only the location name is different.

1. From the **Setup** module, from a **Map Window**, select the **Time Series Icon Tool** ( ) or the **Select Tool** ( ), right-click on a **Time Series Icon**, from the shortcut menu (Figure 1), click **Copy**.
2. Right-click on the **Time Series Icon** that you want to copy the information to, from the shortcut menu, click **Paste**.
3. The **Paste Icon** dialog (Figure 2) will open. In the **To** column change the location name for the new **Time Series Icon**, click **Yes**. The **Paste Icon** dialog (Figure 2) will close. The time series icon will display on the **Map Window** with the new name. This copies the contents of the original icon into the new one, changing the location name at

the same time.



18.3.3 Delete a Time Series Icon

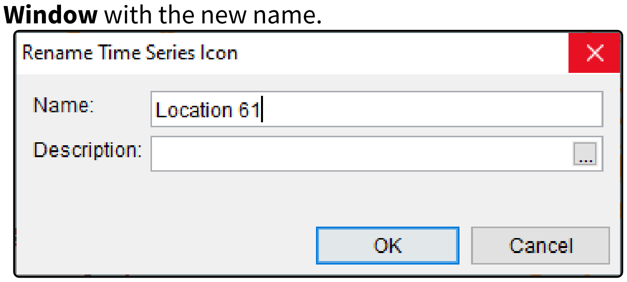
To delete a time series icon:

1. From the **Setup** module, from a **Map Window**, select the **Time Series Icon Tool** ( ) or the **Select Tool** ( ), from the shortcut menu (Figure 1), click **Delete**.
2. A warning message will appear asking if you really want to delete the time series icon, click **Yes**. The selected **Time Series Icon** will no longer display on the **Map Window**.

18.3.4 Rename a Time Series Icon

To rename a time series icon:

1. From the **Setup** module, from a **Map Window**, select the **Time Series Icon Tool** ( ) or the **Select Tool** ( ), from the shortcut menu (Figure 1), click **Rename**.
2. The **Rename Time Series Icon** dialog will open (Figure 3). In the **Name** box, enter a name for the selected time series icon. Click **OK**, the **Rename Time Series Icon** dialog will close. The time series icon will display on the **Map Window** with the new name.

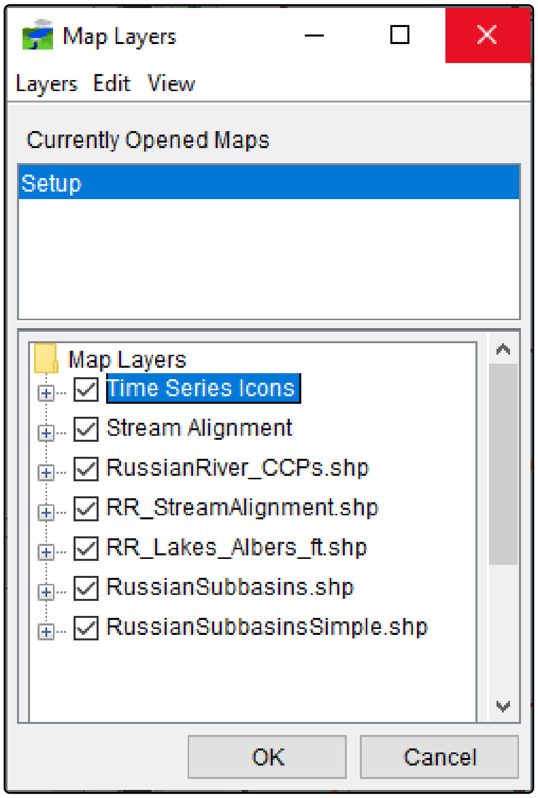


18.3.5 Edit a Time Series Icon

To edit a time series icon, from the From the **Setup** module, from a **Map Window**, select the **Time Series Icon Tool** ( ) or the **Select Tool** ( ), from the shortcut menu (Figure 1), click **Edit**. The **Time Series Icon Editor** will open, Time Series Icon Editor provides more details.

19 Time Series Icons Layers

This chapter describes the creation, configuration, and management of time series icons layers. When you create a new watershed, a default layer called **Time Series Icons** (Figure 1) is created. This layer includes all time series icons that you define for your watershed.



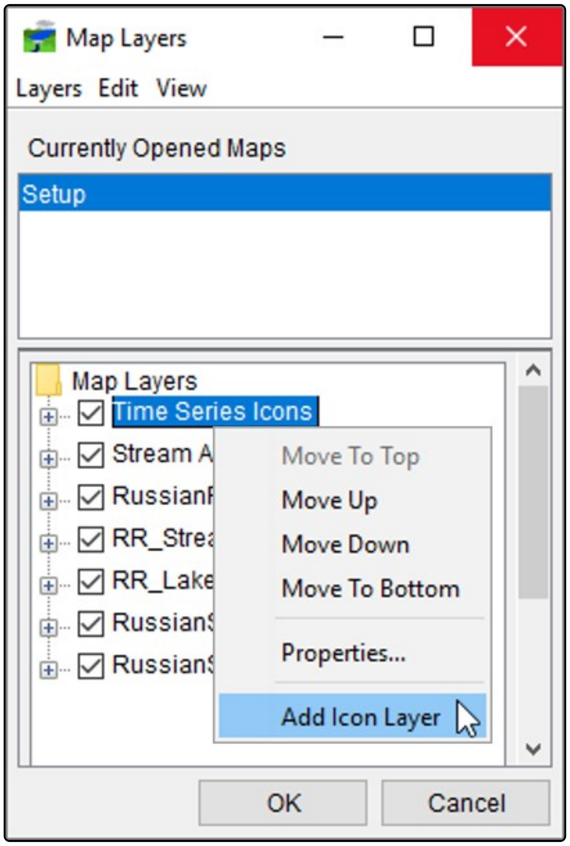
94 Figure 1 Map Layers Dialog

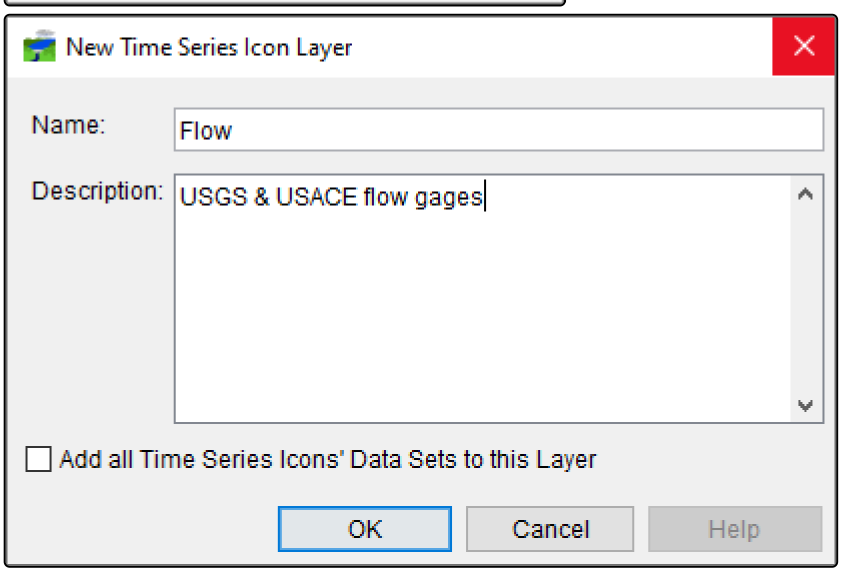
You can create additional time series icons layers to separate time series icons by data type. For example, a gage may report stage and precipitation (and compute flow from stage). You can create layers for each of these data types and name them, for example, **Stage**, **Flow**, and **Precip**. Creating additional layers allows you to focus on the specific data type you wish to see.

19.1 Creating Time Series Icons Layers

To create time series icons layers:

1. From the HEC-RTS main window, click on the **Setup** tab, from the **Maps** menu, click **Map Layers**. The **Map Layers** dialog will open. From the **Map Layers** tree, right-click on the **Time Series Icons** layer. From the shortcut menu (Figure 1), click **Add Icon Layer**, the **New Time Series Icon Layer** dialog opens (Figure 2).

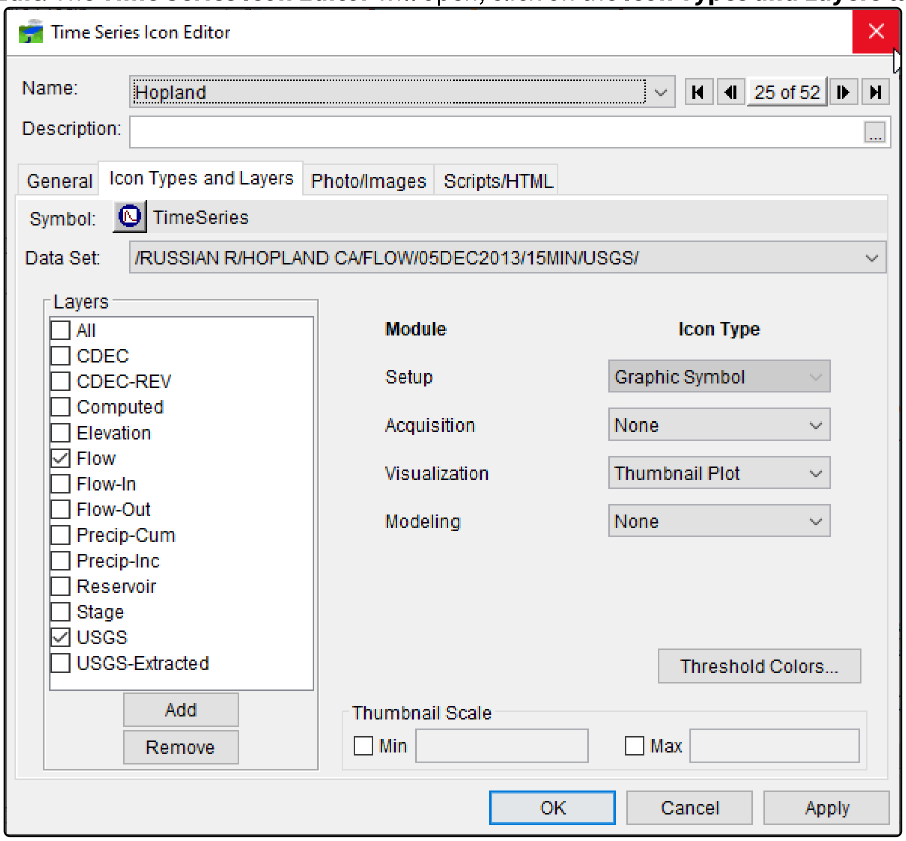




2. In the **Name** field (Figure 2), type a name for the time series icon layer. For example - *Flow, Stage,* or *Precip*.
3. In the **Description** field (Figure 2), type an optional description for the timer series icon layer.
4. You can add all of the time series icons that you defined for the watershed to the newly created time series icon layer, by selecting **Add all Time Series Icons' Data Sets to this Layer** (Figure 2). You should only do this if the time series icons are of the type that the time series icon layer represents. By default, all icons are part of the

"**All**" time series icon layer. For more detail on how to add individual time series icons to timer series icon layers, refer to Time Series Icons.

5. Another way to create time series icon layers, from the **Setup** module, from the **Map Window**, select the **Time Series Icon Tool** () or the **Select Tool** (), right-click on a time series icon, from the shortcut menu, click **Edit**. The **Time Series Icon Editor** will open, click on the **Icon Types and Layers** tab (Figure 3).



6. From the **Layers** box (Figure 3), click **Add**. The **New Time Series Icon Layer** dialog will open (Figure 2). Repeat Steps 2 through 4 to create a time series icons layer.
7. Click **OK**, and the **New Time Series Icon Layer** dialog will close (Figure 2).

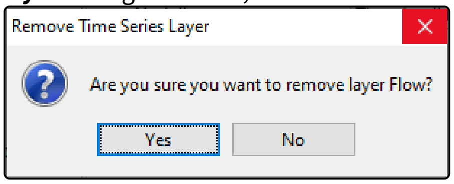
When you are done creating time series icons layers, you can assign datasets from a selected time series icon, to individual time series icons layers (for more details on how to configure time series icons see Time Series Icons).

19.2 Removing a Time Series Icons Layer

To remove a time series icons layer:

1. From the **Setup** module, from the **Map Window**, select the **Time Series Icon Tool** () or the **Select Tool** (), right-click on a time series icon, from the shortcut menu, click **Edit**. The **Time Series Icon Editor** will open. Click on the **Icon Types and Layers** tab.
2. From the **Layers** box, select a time series icon layer, click **Remove**. A **Remove Time Series Layer** dialog will open (Figure 1), asking you if you really want to remove the selected layer. Click **Yes**, the **Remove Time Series**

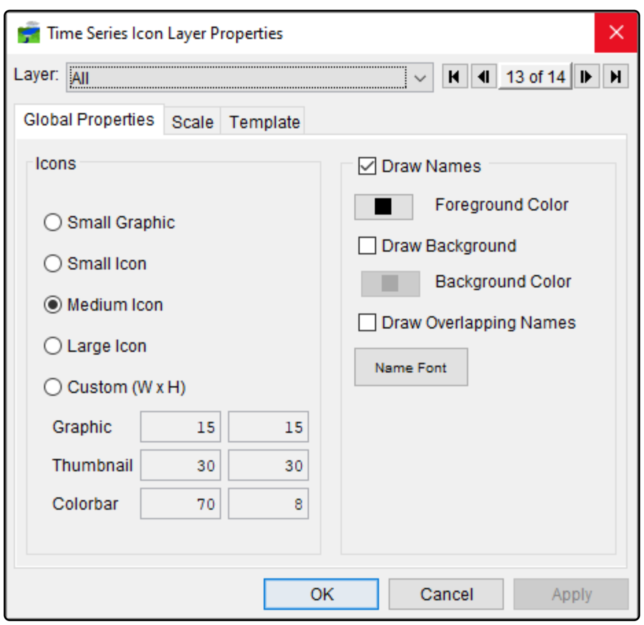
Layer dialog will close, and the time series icon layer will no longer display in the **Layers** box.



19.3 Time Series Icons Layer Properties

You can configure the appearance of a time series icons layer's time series icons and their labels (**Global Properties**), set the visualization scale (**Scale**), and associate a template (**Template**) from the **Time Series Icon Layer Properties** dialog (Figure 1). From CWMS, there are two ways to access the **Time Series Icon Layer Properties** dialog. From the CWMS CAVI main window, click on the **Setup** tab, from the **Maps** menu, click **Map Layers**. The **Map Layers** dialog will open. From the **Map Layers** tree, right-click on the **Time Series Icons** layer. From the shortcut menu, click **Properties**, the **Time Series Icon Layer Properties** dialog will open (Figure 1).

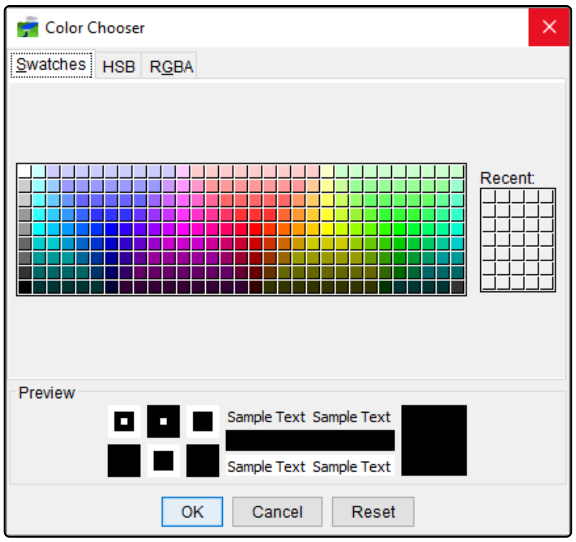
Another way to access the **Time Series Icon Layer Properties** dialog, is from the **Time Series Icon Editor**, click on the **Icon Types and Layers** tab. From the **Layers** box, right-click on a time series icons layer, from the shortcut menu, click **Properties**, the **Time Series Icon Layer Properties** dialog will open (Figure 1).



95 Figure 1 Time Series Icon Layer Properties Dialog - Global Properties Tab

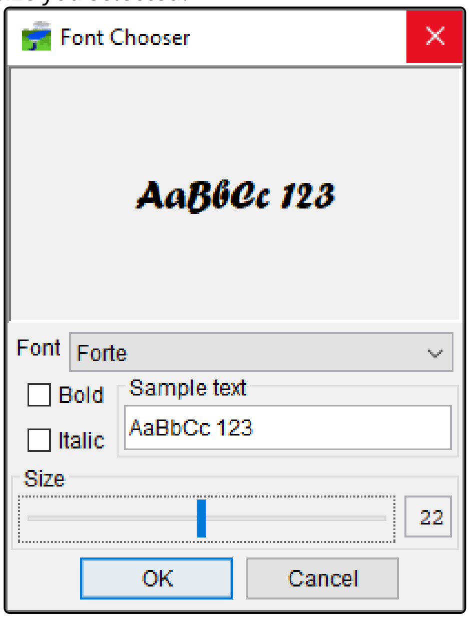
19.3.1 Global Properties

The **Global Properties Tab** (Figure 1) allows you to change the appearance of time series icons and their labels for the time series icons layer that has been selected in the **Layer** list (Figure 1) for only the Acquisition and Visualization modules. Time series icons that have been selected for the Modeling and Setup modules are not affected by changes to **Global Properties**. There are several ways to control the appearance of time series icon labels:

1. Select a time series icons layer, by default, the time series icons display using the **Medium Icon** (Figure 1) format. From the **Icons** box, you have five options for displaying time series icons – **Small Graphic**, **Small Icon**, **Medium Icon**, **Large Icon**, and **Custom**.
2. By default, the labels for time series icons display, if you do not want the labels for a specific layer to display, clear **Draw Names** (Figure 1). Click **Apply**, for the selected time series icons layer, the time series icon labels will no longer display.
3. To change the color of a time series icon's labels, click **Foreground Color**. A **Color Chooser** opens (Figure 2), from this chooser you can select the color you want the labels to display in. Click **OK**, and the **Color Chooser** closes. Click **Apply**, for the selected time series icons layer, the time series icon labels will display in the color. For more information see Color Chooser.


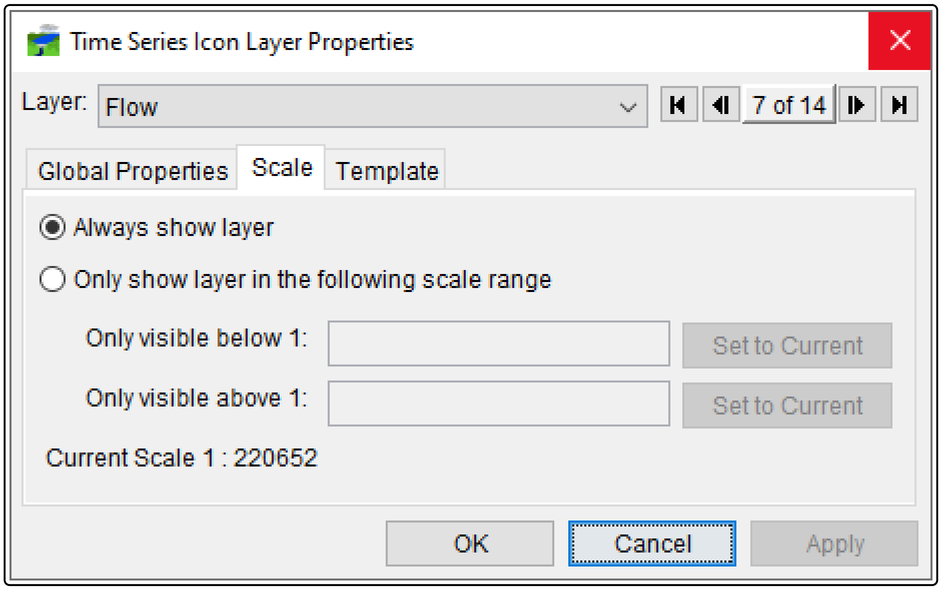
4. By default, time series icon labels do not have a background color, if you want a time series icon label to have a background color, click **Draw Background** (Figure 1) and click **Apply**. For the selected time series icons layer, the time series icon labels will now display with a background color (light gray). You can change the background color by clicking **Background Color** (Figure 1). The **Color Chooser** will open (Figure 2), select the color you want the background of the time series icon labels to display in, and click **OK**. The **Color Chooser** will close, click **Apply**. For the selected time series icons layer, the time series icon labels will now display with a background color in the color you choose.
5. When a module's **Map Window** displays the whole watershed, some time series icon labels will not display when the time series icons overlap other elements. To display a time series icon layer's labels, click **Draw Overlapping Names** (Figure 1), and click **Apply**. For the selected time series icons layer, the time series icon labels will now display when viewing the whole watershed.
6. To change the font of a time series icons layer's time series icon labels, click **Name Font** (Figure 1). A **Font Chooser** will open (Figure 3), where you can select the font type from the **Font** list (Figure 3), and you can select a size using the **Size** slider (Figure 3). In addition, you can select whether to have the font display in **Bold** and/or in **Italic**. Click **OK**, and the **Font Chooser** closes. From the **Time Series Icon Layer Properties** dialog (Figure 1), click **Apply**. For the selected time series icons layer, the time series icon labels will now display in the font and

size you selected.



19.3.2 Scale

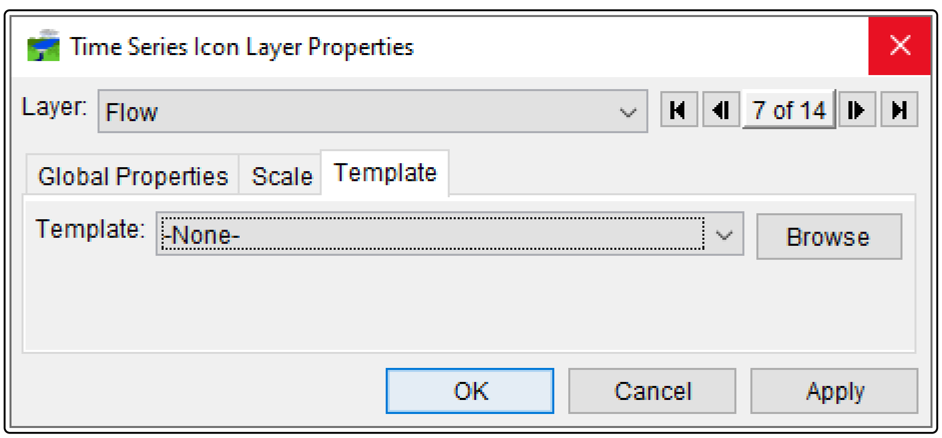
The **Scale Tab** (Figure 4) of the **Time Series Icon Layer Properties** dialog allows you to specify the map scale, or zoom level, at which a time series icon sub-layer becomes visible. By default, **Always show layer** is selected, which means the selected time series icons layer will always be displayed in the **Map Window**, regardless of the zoom level. However, you can select **Only show layer in the following scale range**, if you want to display the selected time series icons layer only when the **Map Window** scale is zoomed to the specified scale. You can specify scales below, above, or both, for when the selected time series icons layer will display time series ions. You can either enter the scale directly into the **Only visible below 1** and/or **Only visible above 1** or click **Set to Current** to set the scale equal for the current display.



96 Figure 4 Time Series Icon Layer Properties Dialog - Scale Tab

19.3.3 Template

Templates are created to save customized plot settings that can be used by other plots and the time series icons layers. From the **Template Tab** (Figure 5) of the **Time Series Icon Layer Properties** dialog, you can associate a template with the **Time Series Icons Layer** by selecting a template from the **Template** list. In HEC-RTS, templates are created from plot windows.



97 Figure 5 Time Series Icon Layer Properties Dialog - Template Tab

20 Model Alternatives and Forecast Runs

A **model alternative** is a single model configuration with a specific set of input and parameters. A second model alternative would be defined by a different configuration or different input or parameters. A **forecast run** is a single forecast scenario defined by a specific set of data, information, and model alternatives. A second forecast run would be defined by different data, information, or model alternatives. Model alternatives run sequentially during a forecast. Model alternatives and forecast runs are described in this chapter. The model alternatives and forecast runs are manipulated primarily in the **Setup** module of HEC-RTS. Setup Module provides a detailed description of the Setup module and its interface.

20.1 Model Alternatives

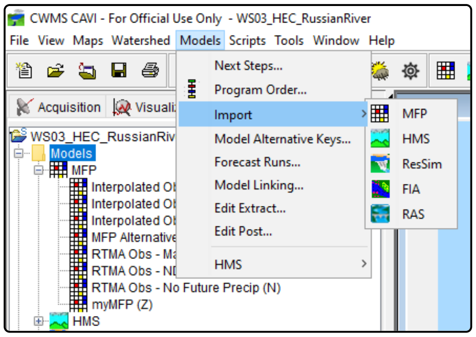
Model alternatives consist of input data sets and configurations for each model. Model alternatives configurations do not include time-specific data. Therefore, model alternatives can be used in conjunction with any time window. In general, models and their alternatives are created using the software of the native model, for example, HEC-HMS, HEC-ResSim, HEC-RAS, and HEC-FIA. Once models and the alternatives created and configured initially, the model alternatives are imported into the watershed using the import commands in the **File** menu of the Setup module. The model alternative import process is detailed in Importing Watersheds and Model Alternatives. Once imported, model alternatives can be configured and edited in the HEC-RTS interface.

HEC-RTS uses plug-ins that allow interfacing with the software programs of the native model. This means that all aspects of model alternatives are configurable using the native model interface of HEC-RTS. For example, you can edit HEC-HMS base model parameters directly using the HEC-HMS software.

MFP deals with observed meteorological data and future precipitation. You can configure MFP model alternatives for use in conjunction with HEC-HMS to compute a variety of observed or future precipitation scenarios. An HEC-HMS model alternative simulates watershed response to precipitation. You can use HEC-HMS Model alternatives to forecast future flows. An HEC-ResSim model alternative assists with reservoir operations, consisting of the physical data representing a river-reservoir system and operating rules for all controllable elements in the system. HEC-RAS model alternatives describe water surface profile and stages at gage locations, geometry information for a river, and defined flow data sets. An HEC-FIA model alternative allows you to analyze flood consequences. HEC-FIA model forecast alternatives are a combination of impact areas and hydrograph time series sets. You can edit the parameters within each model alternative to reflect changing conditions.

20.2 Importing Watersheds and Model Alternatives

There are two ways to import models and their alternatives into an HEC-RTS study: (1) the **Import 2.1 Watershed** command found in the **Watershed** menu, or (2) choose commands from the **Import** menu option (Figure 1) found in the **Models** menu. To create an HEC-RTS study from a CWMS watershed, the CWMS watershed needs to have been configured in CWMS Version 2.1 or higher.

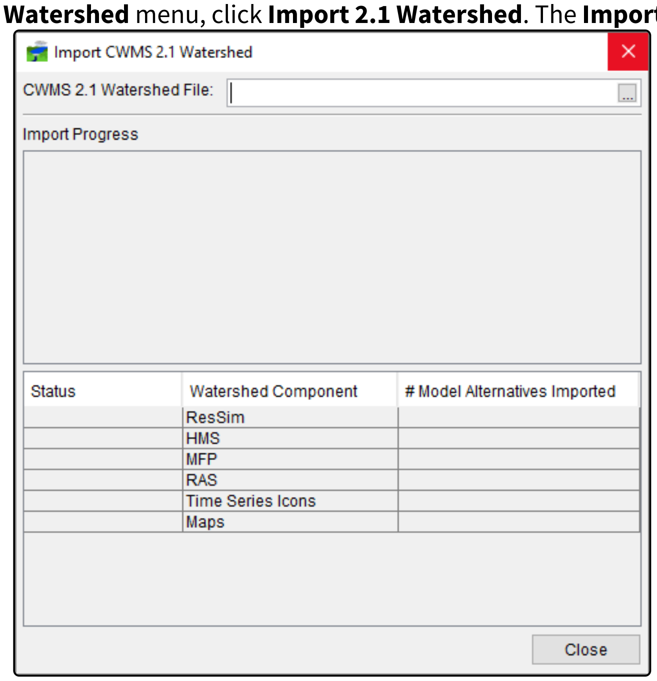


98 Figure 1 Setup Module - Models Menu - Import

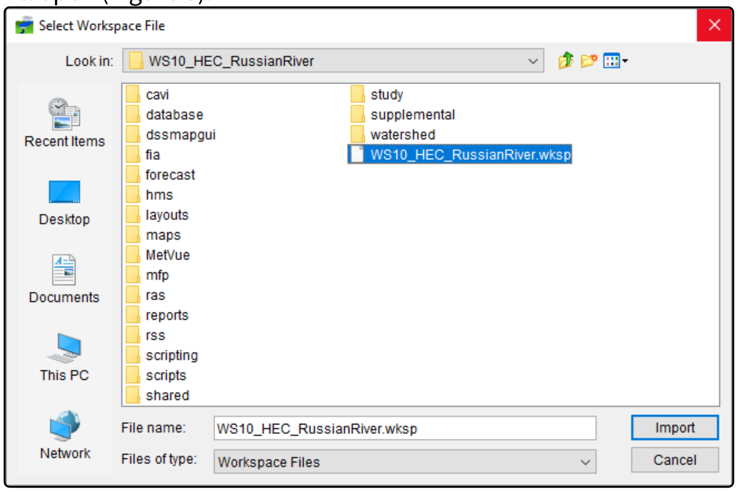
20.2.1 Importing an Entire CWMS Watershed

To create an HEC-RTS study from a CWMS watershed, perform the following steps:

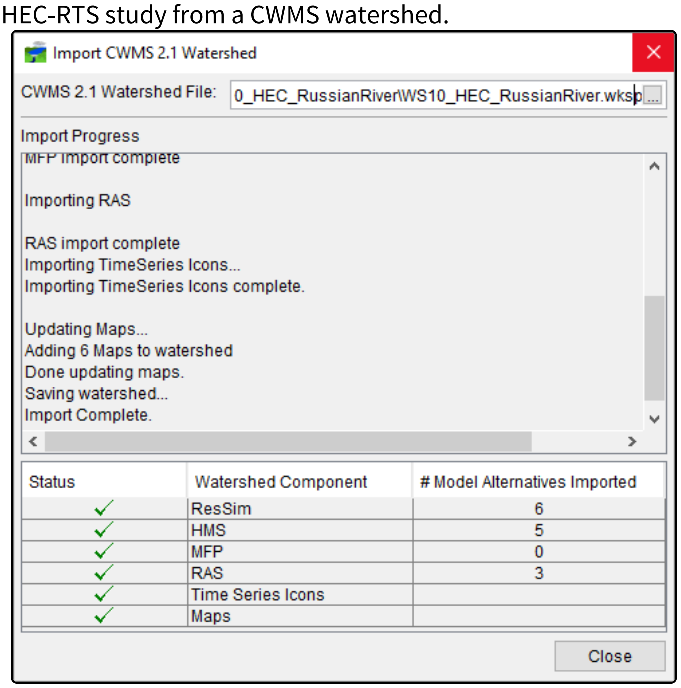
1. Create a **New Watershed** as detailed in Creating a Watershed. From the HEC-RTS main window, from the **Watershed** menu, click **Import 2.1 Watershed**. The **Import CWMS 2.1 Watershed** dialog (Figure 2) will open.



2. To the right of the **Import CWMS 2.1 Watershed** box (Figure 2), click  , the **Select Workspace File** browser will open (Figure 3).



3. From the **Select Workspace File** browser (Figure 3), navigate to and select the CWMS workspace file (.*wksp*) you want to import. Click **Import**, the **Select Workspace File** browser will close.
4. A **Continue Import** window will open, asking you if you want to continue with the import. Click **Yes**, the **Continue Import** window will close. The import process will start, and the results of the import are displayed in the **Import CWMS 2.1 Watershed** dialog (Figure 2).
5. When the import is finished from the **Import CWMS 2.1 Watershed** dialog (Figure 4) click **Close**, the **Import CWMS 2.1 Watershed** dialog will close. The **Next Steps** dialog will open, click **Close**. You now have created an HEC-RTS study from a CWMS watershed.



20.2.2 Importing Model Alternatives

Individual model alternatives can be imported into an HEC-RTS study. A model's alternative (plan, alternative, run) can be imported regardless of whether the model is part of an HEC-RTS watershed or exists as a stand-alone model.

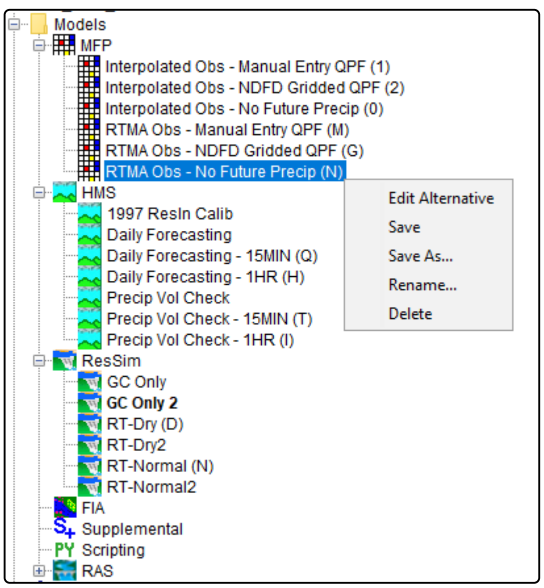
To import model alternatives into an HEC-RTS study, complete the following steps:

1. If an HEC-RTS study does not already exist, create a new watershed as detailed in Creating a Watershed.
2. Import an MFP alternative as detailed in Section MFP Model Alternative.
3. Import an HEC-HMS alternative (run) as detailed in Create an HEC-HMS Alternative.
4. Import a HEC-ResSim watershed as detailed in Create an HEC-ResSim Alternative.
5. Import a HEC-RAS plan as detailed in Create an HEC-RAS Alternative.
6. Import an HEC-FIA alternative as detailed in Create an HEC-FIA Alternative.

20.3 Interacting with Model Alternatives

20.3.1 Setup Module

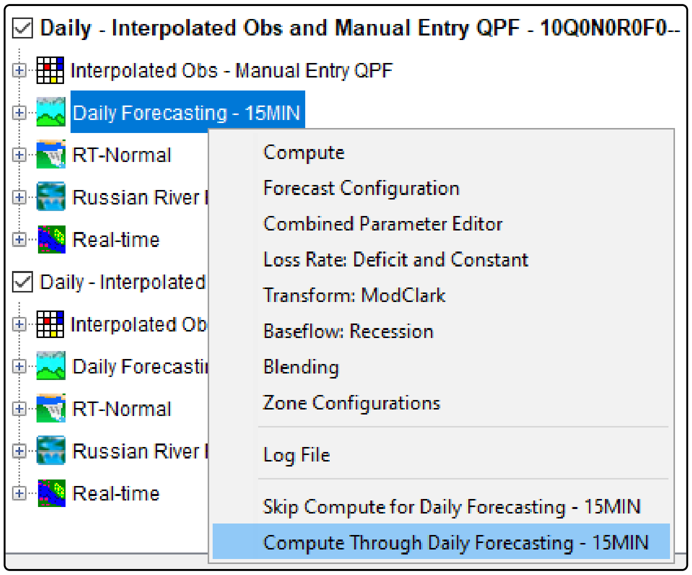
Model alternatives are manipulated primarily in the Setup module. You can access the individual model editors by right-clicking on a specific model alternative in the watershed tree and using the shortcut menu (Figure 1) that displays.



99 Figure 1 Watershed Tree - Models Folder - Model Alternatives Shortcut Menu

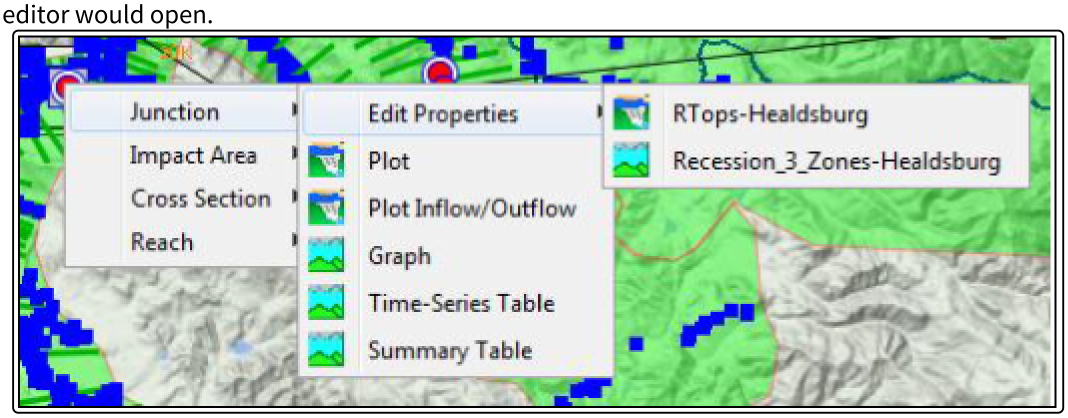
20.3.2 Modeling Module

You can also interact with specific forecast run's model alternatives in the Modeling module. With a forecast alternative active, from the forecast runs details section (Figure 2) of the Modeling module, you can also access the individual model editors (Figure 2). From the control panel in the Modeling module, right-click on the active model and a shortcut menu will display.



20.3.3 Schematic Elements

From the map window, you can access certain model editors from schematic elements on the display area. For example, if your active forecast run has a HEC-ResSim model, on the display area you will see HEC-ResSim model elements, such as junctions. Right click on a junction, from the shortcut menu (Figure 3) point to **Junction**, point to **Edit Properties,** and model alternatives available at a junction will be displayed (Figure 3). For example, if you want to edit an HEC-ResSim junction, from the shortcut menu you would click *RTops-Healdsburg*, and the HEC-ResSim **Junction** editor would open.

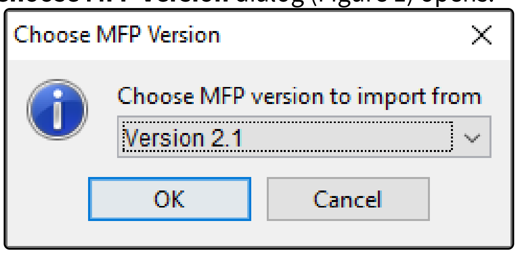


20.4 MFP Model Alternative

An MFP alternative consists of an HEC-HMS meteorological model, a zone configuration (based on an HEC-HMS basin model) and a set of precipitation forecast data. Your HEC-HMS project can contain more than one meteorological model with gridded precipitation from different sources, such as NEXRAD radar, interpolation from gage reports, or results from a grid-based snow model. You will need to create a separate MFP alternative for each source of lookback precipitation grids you intend to use in your forecast models.

20.4.1 Import an MFP Alternative

To import an **MFP** alternative:

1. From the HEC-RTS main window, click the **Setup** tab, from the **Models** menu, point to **Import**, click MFP, and the **Choose MFP Version** dialog (Figure 1) opens.


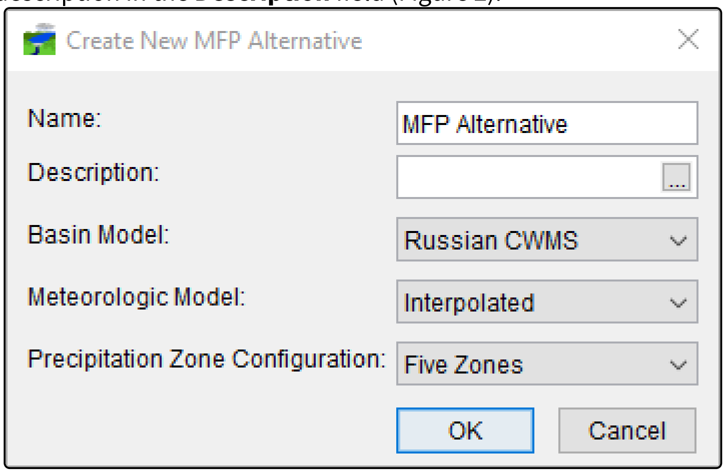
2. Select the MFP version to import from in the list (Figure 1). Available import version choices for MFP alternatives are Versions 2.1 and 3.x.
3. Click **OK**, the **Choose MFP Version** dialog (Figure 1) closes, and the **Select MFP project file to import from** browser will open.
4. For MFP Version 2.1, navigate to and select the **MFP** project file (.*conf*) you want to import, click **Open**.
5. For MFP Version 3.0 or higher, navigate to and select the folder containing the **MFP** project file (*.mfp*) **you want to import, click *Open**.
6. The **Select MFP project file to import from** browser will close. An **Importing** progress bar will display, once that closes, an MFP model alternative(s) is imported into the watershed. **Save** your watershed.

20.4.2 Create an MFP Alternative

To create an **MFP** alternative:

1. From the HEC-RTS main window, click the **Setup** tab, from the watershed tree, from the **Model** folder, right-click **MFP**, from the shortcut menu, click **New**.

2. The **Create New MFP Alternative** dialog (Figure 2) opens. Enter a name in the **Name** field and enter a description in the **Description** field (Figure 2).



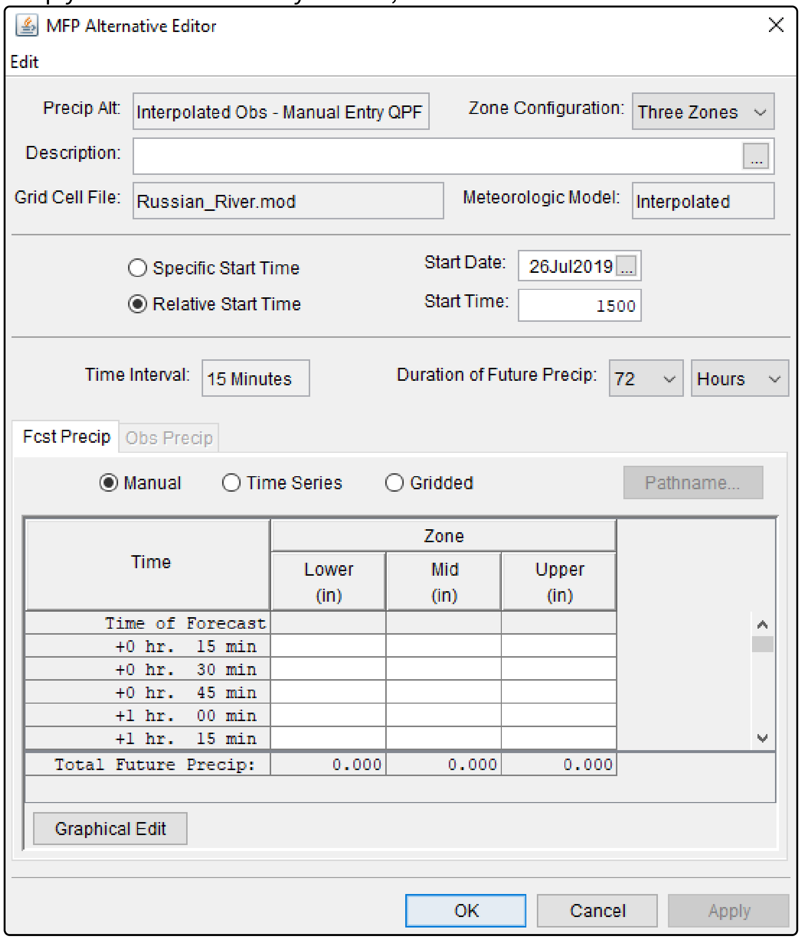
3. From the **Basin Model** list (Figure 2), select a basin model. The choice of basin model will determine which precipitation zone configurations will be available for the MFP alternative. Make sure to select a basin model that covers the same area covered by any basin models used in MFP alternatives that will combine with this MFP alternative.
4. From the **Meteorologic Model** list (Figure 2), select a meteorologic model. The meteorologic model determines the grids used in the lookback portion of the forecast.
5. From the **Precipitation Zone Configuration** list (Figure 2), select a precipitation zone configuration.
6. Click **OK**, the **Create New MFP Alternative** dialog closes. To edit the newly created MFP alternative, from the HEC-RTSI main window, click the **Setup** tab, from the watershed tree, from the **Model** folder, right-click on the newly created MFP alternative. From the shortcut menu, click **Edit Alternative**, the **MFP Alternative Editor** opens (Figure 3).

By default, the **MFP Alternative Editor** (Figure 3) is set for manual entry of forecast precipitation data by precipitation zone. If you have QPF data available, (DSS file) you can enter the data by choosing one of the QPF options.

To add precipitation data for the selected precipitation zone configuration:

1. From the **MFP Alternative Editor** (Figure 3), select the precipitation zone configuration from the **Zone Configuration** list.

2. Setup your time window. By default, **Relative Start Time** is selected.



3. From the **Duration of Future Precipitation** list (Figure 3), select the duration for future precipitation. This choice sets the number of rows in the table where you will enter values for each precipitation zone of the selected precipitation zone configuration.
4. Enter precipitation data in the table. Values are entered by time step, with each cell representing one-time step for the zone. A running total of the forecast precipitation is kept at the bottom of the table. You may want to prepare a set of forecast values with a spreadsheet program and then copy the values from the spreadsheet to the **MFP Alternative Editor**. Once you have entered your precipitation data, click **OK**, the **MFP Alternative** editor (Figure 3) closes.
5. From the **File** menu, click **Save Watershed**.

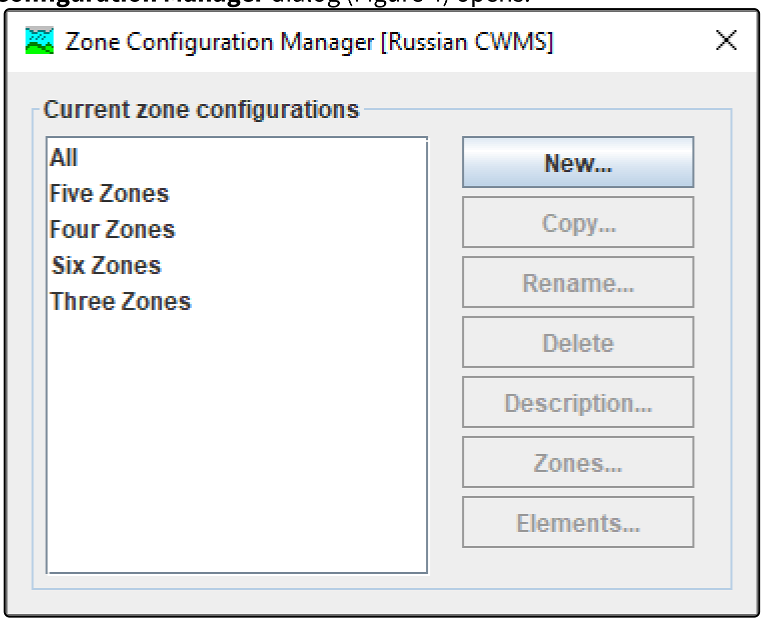
20.4.3 MFP Precipitation Zone Configurations

A precipitation zone configuration is required to create an **MFP** alternative. A zone configuration is a set of zones, a **zone** is a set of subbasins in the watershed. In MFP, precipitation forecasts are assigned by zone, with all subbasins in the zone receiving the same precipitation increments. You should group subbasins into zones based on similarity of expected precipitation.

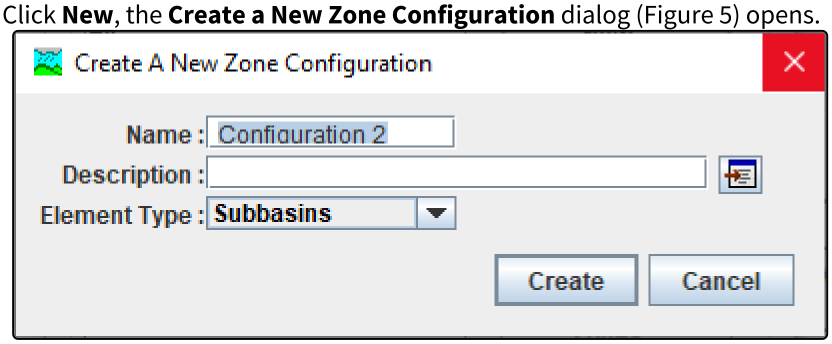
20.4.3.1 Create a Zone Configuration

To create a precipitation zone configuration:

1. From the **MFP Alternative Editor** (Figure 3), from the **Edit** menu, click **Zone Configuration**. The **Zone Configuration Manager** dialog (Figure 4) opens.



2. Click **New**, the **Create a New Zone Configuration** dialog (Figure 5) opens.



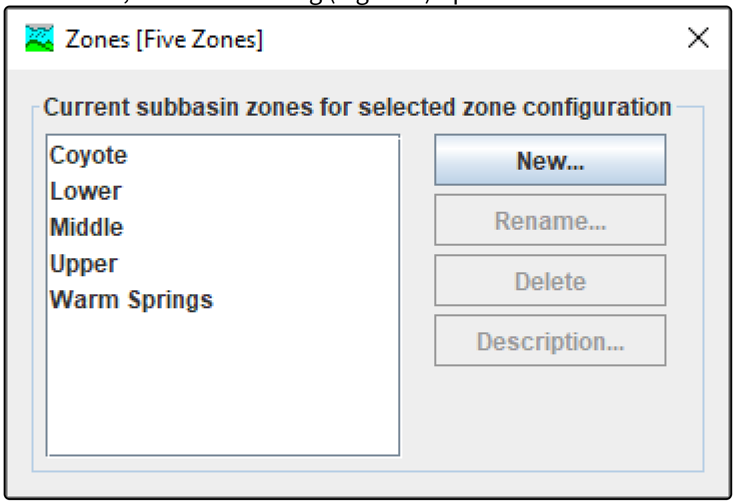
3. From the **Create a New Zone Configuration** dialog, enter a name in the **Name** field and a description into the **Description** field.
4. From the **Element Type** list (Figure 5), select either **Subbasins** or **Reaches**. Click **Create**, and the **Create A New Zone Configuration** dialog closes.

20.4.3.2 Create Zones

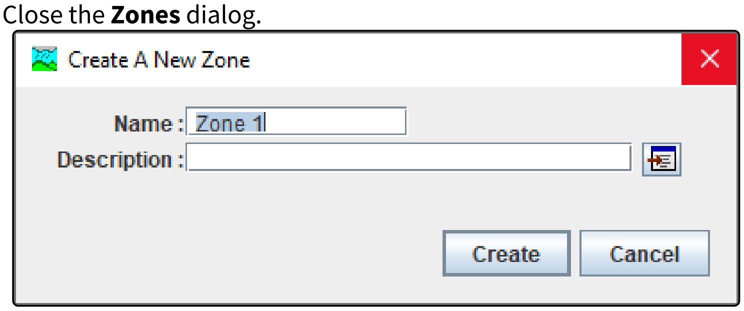
For each zone configuration, you will need to create at least one zone. To create a zone:

1. From the **Zone Configuration Manager** (Figure 4) select the newly created zone configuration.

2. Click **Zones**, the **Zones** dialog (Figure 6) opens.



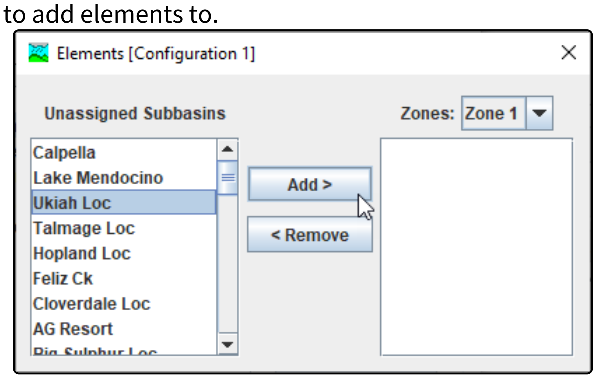
3. Click **New**, the **Create A New Zone** dialog opens (Figure 7). Enter a name in the **Name** field and description into the **Description** field. Click **Create**, the **Create A New Zone** dialog closes. The name of the zone you just created displays in the **Current subbasin zones for selected zone configuration** list on the **Zones** dialog (Figure 6). Close the **Zones** dialog.



20.4.3.3 Add Subbasin to a Zone

Once precipitation zones have been created, you can assign elements (subbasins or reaches) to the zones.

1. From the **Zone Configuration Manager** (Figure 4) select the zone configuration you want to configure.
2. Click **Elements**, the **Elements** dialog (Figure 8) opens. From the **Zones** list (Figure 4), select which zone you want to add elements to.



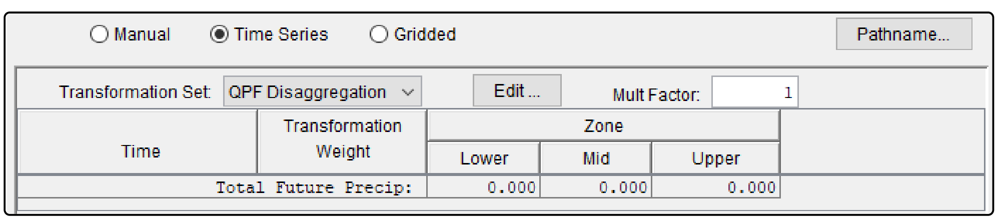
3. Now you need to define the elements (subbasins or reaches) that are part of the selected zone. For example, in Figure 8, from the **Unassigned Subbasins** list you will be selecting a subbasin(s) to add to the selected zone. You can select individual element(s); select ranges of subbasins by shift-clicking; select discontinuous groups of subbasins by control-clicking; or, double-click on element. Click **Add**, the selected subbasin names will now display in the box that is under the **Zone** list (Figure 8).
4. You can move elements out of a zone by selecting an element in the box under the **Zone** list, and then click **Remove** or by double-clicking on that element.
5. When you have finished assigning elements to zones, close the **Elements** dialog (Figure 8). Then close the **Zone Configuration Manager** (Figure 4).

20.4.4 Define a QPF Transform Set

In place of manually entered precipitation forecasts, MFP permits you to incorporate a sequence of QPF (quantitative precipitation forecast) values into an MFP alternative from an external source. There are two types of QPF data that are available from the National Weather Service or from other sources. One type is point QPFs, which consist of a time series of forecasted precipitation depths at a single location. The second type is gridded QPFs, which represent a spatially distributed forecast of precipitation depths.

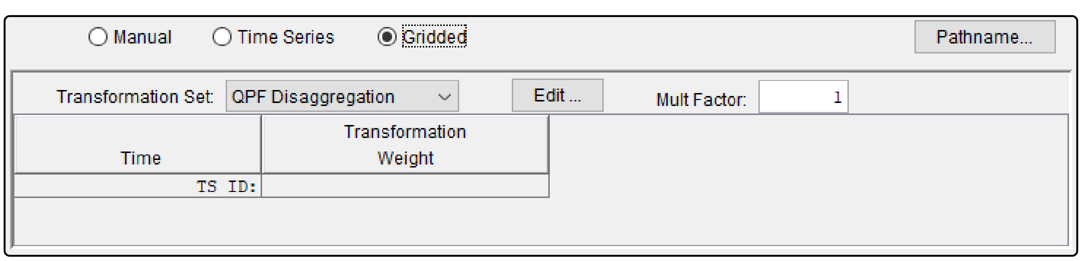
MFP can assign point QPF time series to precipitation forecast zones or assign a sequence of QPF grids to cover the entire watershed. In either case, QPFs are typically given in six-hour increments covering the next twenty-four hours, and a single 24-hour total following the four 6-hour increments. It is necessary to disaggregate these totals into increments that match your modeling time step.

To use QPFs in your forecast model, from the **MFP Alternative** editor (Figure 3), select either **Time Series** or **Gridded**. Each option (**Manual**, **Time Series**, or **Gridded**) presents a different form of the forecast precipitation table. If you select **Time Series**, the table appears as shown in Figure 9, and once you have defined your QPF data and defined a set of disaggregating weights, the table will be filled with precipitation increments for the zone.



100 Figure 9 Time Series Table - MFP Alternative Editor

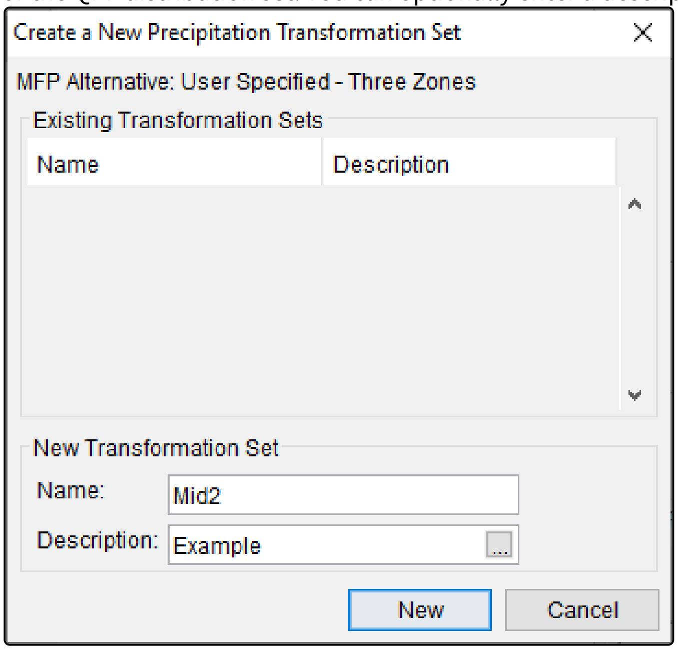
If you select **Gridded**, the table appears as shown in Figure 10 and once you have defined your QPF data, the table will be filled with the disaggregating weights (not the precipitation values themselves).



101 Figure 10 MFP Alternative Editor - QPF Table - Gridded

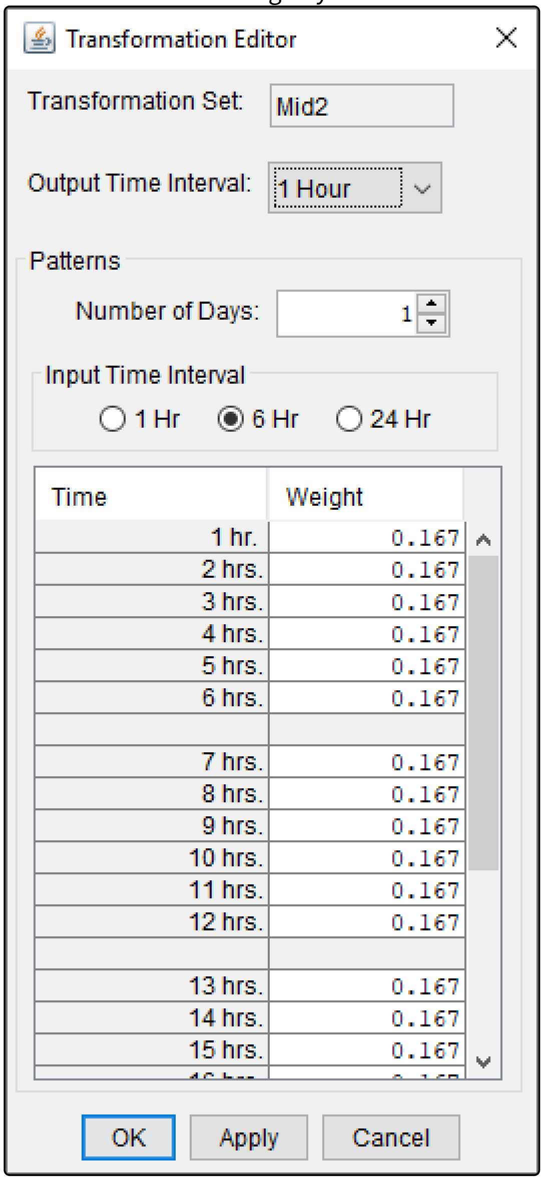
For either type of QPF data, to define QPF data:

1. From the **MFP Alternative Editor** (Figure 3), from the **Edit** menu, point to **Precip Transformation Set**, click **New**.
2. The **Create a New Precipitation Transformation Set** dialog opens (Figure 11). In the **Name** field, enter a name for the QPF distribution set. You can optionally enter a description in the **Description** field.



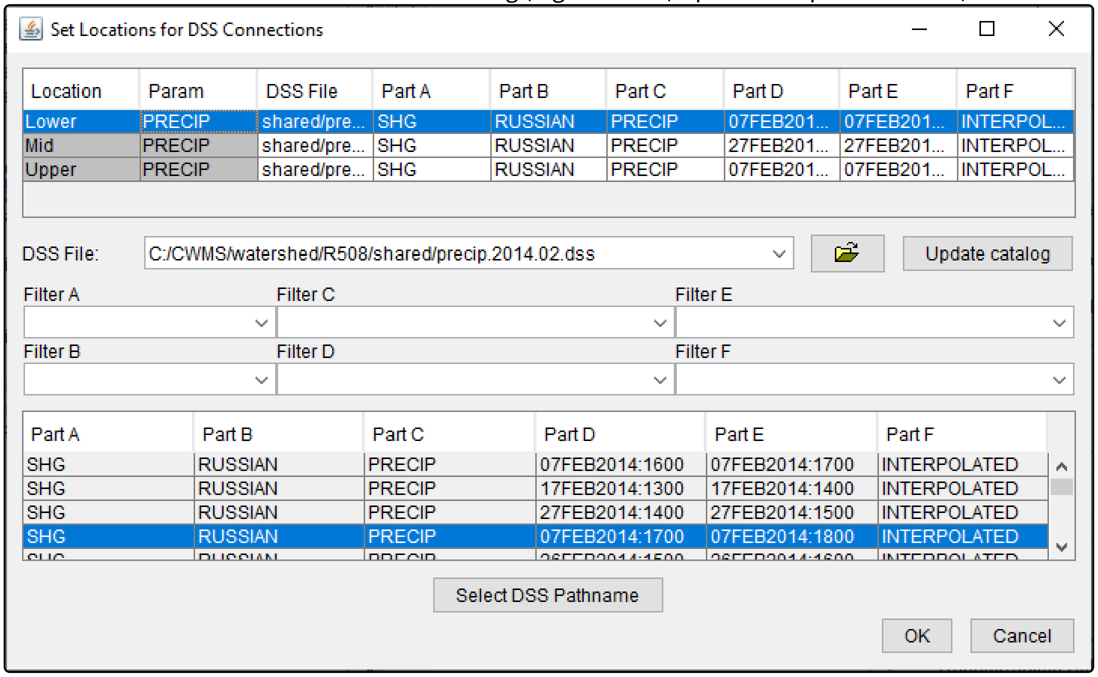
3. Click **New**, the **Transformation Editor** (Figure 12) opens. From the **Transformation Editor,** you can specify the **Output Time Interval** and the **Input Time Interval**. The **Transformation Editor** (Figure 12) allows you to specify weights for two adjacent blocks of one, six, or twenty-four hours each (based off the **Input Time Interval**). The weights will be normalized for each block, so the two six-hour blocks shown in the example will put half of the six-hour QPF in the third and fourth hours of the six-hour blocks. QPF are referenced to UTC so the

blocks will shift according to your watershed's time zone.



4. Once the disaggregation set is defined, you can associate a QPF time series record with a zone defined in the precipitation zone configurations associated with the MFP alternative. Click **OK**, the **Transformation Editor** closes (Figure 12).
5. To associate a QPF time series record with a zone, from the **MFP Alternative Editor** dialog (Figure 3), click **Time Series**. Select a row in the table and click **Pathname**.

6. The **Set Locations for DSS Connections** dialog (Figure 19.20) opens. To open a DSS file, click  .



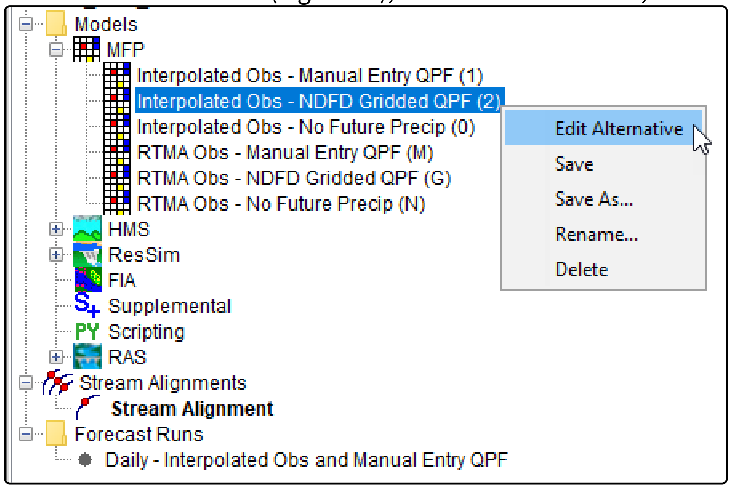
7. From the **Open File** browser, locate and open the appropriate DSS file. The lower table will be filled with the DSS catalog information. The top table has the list of precipitation zones for the selected MFP zone configuration. Select a precipitation zone in the top table by clicking on a row. Select a QPF HEC-DSS pathname by clicking on a row in the bottom table. Once a row from each table is highlighted, the HEC-DSS pathname will be assigned to the precipitation zone by clicking **Select HEC-DSS Pathname**.
8. Once you click **OK**, the **MFP Alternative** editor (Figure 3) will read the DSS file and try to find the QPF data.

20.4.5 Edit an MFP Alternative

To edit an MFP alternative:

1. From the watershed tree (Figure 14), under the **Models** folder, expand the **MFP** folder (Figure 14). Right-click on the MFP alternative you wish to edit.

2. From the shortcut menu (Figure 14), click **Edit Alternative**, the **MFP Alternative Editor** (Figure 3) opens.

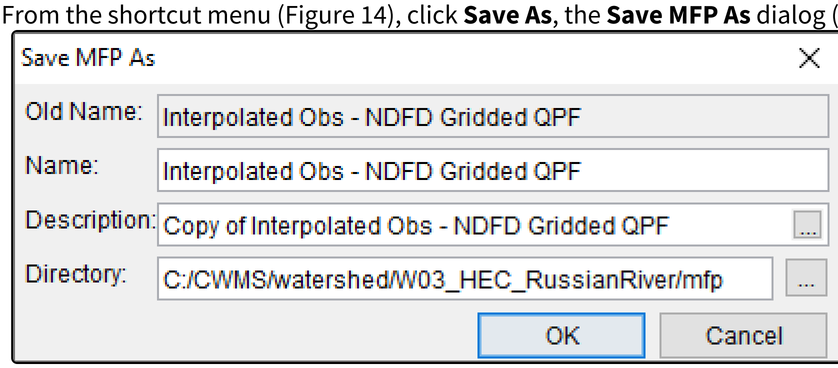


3. Use the **MFP Alternative Editor** to make changes to the MPF alternative you are editing.
4. Click the **OK,** and the **MFP Alternative Editor** closes.

20.4.6 Copy an MFP Alternative

To copy an MFP alternate:

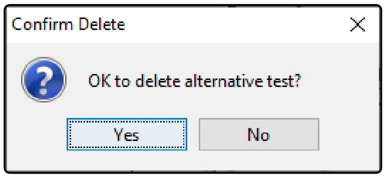
1. From the watershed tree (Figure 14), under the **Models** folder, expand the **MFP** folder (Figure 14). Right-click on the MFP alternative you wish to edit.
2. From the shortcut menu (Figure 14), click **Save As**, the **Save MFP As** dialog (Figure 15) opens.



3. Enter the new name in the **Name** field and fill out the **Description** field. (You can specify the directory where you save the watershed if you wish to save a copy of the alternative to a directory outside of the watershed.)
4. Click **OK**, and the **Save MFP As** dialog closes.

20.4.7 Delete an MFP Alternative

To delete an MFP alternative, from the watershed tree (Figure 14), under the **Models** folder, expand the **MFP** folder (Figure 14). Right-click on the MFP alternative you wish to edit. From the shortcut menu (Figure 14), click **Delete**, a **Confirm Delete** window (Figure 16) opens. Asking you are you sure you want to delete the selected MFP alternative. Click **Yes**, the **Confirm Delete** window (Figure 16) will close. The MFP alternative is no longer displayed under the MFP folder.



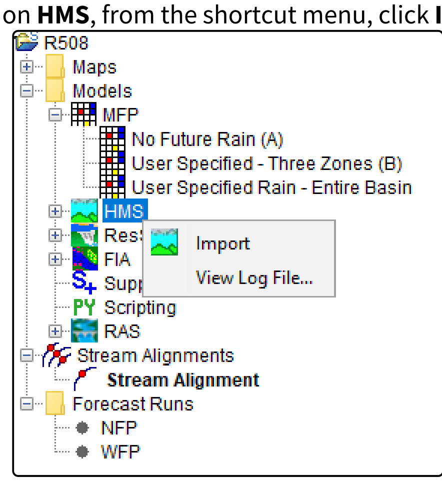
102 Figure 16 Confirm Delete MFP Alternative Dialog

20.5 Create an HEC-HMS Alternative

An HEC-HMS alternative consists of a basin model, loss parameter, transform parameter, baseflow zone configurations, and a set of parameter adjustments for each zone. Within an HEC-HMS alternative, you can adjust parameters in the HEC-HMS basin model to try to match observed flows at the time of forecast. You can also control blending of model results to observed flows where they are available. If the HEC-HMS project contains more than one basin model (for summer and winter conditions, for instance), you can build HEC-HMS alternatives around each of them.

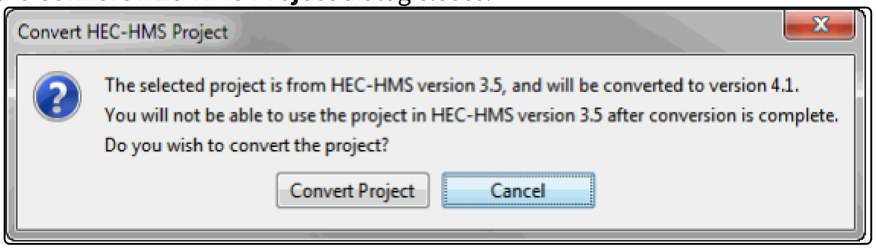
20.5.1 Import HEC-HMS Model Alternative

To import HEC-HMS model alternatives:

1. From the HEC-RTS main window, click the **Setup** tab, from the watershed tree, from the **Model** folder, right-click on **HMS**, from the shortcut menu, click **Import** (Figure 1), a **Select Project File** browser will open.


2. From the **Select Project File** browser, navigate to and select the HEC-HMS project file (***.hms*) you wish to import. Click *Select**, and the import process will begin.
3. If you are importing an HEC-HMS model alternative created in a previous version of HEC-HMS, then the **Convert HEC-HMS Project** dialog (Figure 2) displays prompting you to convert the project. Click **Convert Project**, and

the **Convert HEC-HMS Project** dialog closes.



4. Once the import is complete the **Message Pane** will display "HMS: NOTE 16161: Import finished". Be sure to save your watershed.

20.5.2 HMS Loss and Baseflow Zone Configurations

After you import the HEC-HMS model alternatives, you must create zone configurations for loss parameter, transform parameter, and baseflow parameter adjustments. A **zone configuration** is a set of zones, and a **zone** is a set of elements (subbasins or reaches) in the watershed. You should group elements into zones based on similarity of hydrologic properties and the availability of observed flows against which to calibrate zone parameters. For example, you might group the subbasins in a headwaters area above a reservoir together into a zone. At forecast time, the baseflow and loss parameters for the subbasins in that zone will be adjusted together so that the hydrograph calculated by HEC-HMS for the inflow to the reservoir fits the observed inflows to the reservoir as well as possible.

20.5.2.1 Create a Zone Configuration

To create a zone configuration:

1. From the HEC-RTS main window, click the **Setup** tab, from the watershed tree, from the **Model** folder, expand **HMS**. Right-click on an HEC-HMS model alternative. From the shortcut menu, click **Zone Configurations**. The **Zone Configuration Manager** opens.
2. To create a new zone configuration, from the **Zone Configuration Manager**, click **New**. The **Create a New Zone Configuration** dialog opens.
3. From the **Create A New Zone Configuration** dialog, enter a name into the **Name** box and description into the **Description** box.
4. From the **Element Type** list, select either **Subbasins** or **Reaches**. Click **Create**, the **Create A New Zone Configuration** dialog closes and the new zone configuration is added to the list of current zone configurations.
5. Close the **Zone Configuration Manager** and **Save** your watershed.

20.5.2.2 Create Zones

For each loss or baseflow zone configuration, you will need to create at least one loss/baseflow zone. You should base the zones on similarity of hydrologic properties and the availability of observed flows to calibrate zone parameters against.

To create a zone:

1. From the HEC-RTS main window, click the **Setup** tab, from the watershed tree, from the **Model** folder, expand **HMS**. Right-click on an HEC-HMS model alternative. From the shortcut menu, click **Zone Configurations**. The **Zone Configuration Manager** opens.
2. From the **Current zone configurations** box, select a zone configuration. Click **Zones**, the **Zones** dialog opens.

3. From the **Zones** dialog, click **New**, the **Create A New Zone** dialog opens. Enter a name into the **Name** field (required) and description into the **Description** field (optional).
4. Click **Create**, the **Create A New Zone** dialog will close. The name of zone you created displays in the **Current zones for selected zone configuration** list on the **Zones** dialog. Close the **Zones** dialog and **Save** your watershed.

20.5.2.3 Add Elements to a Loss Parameter

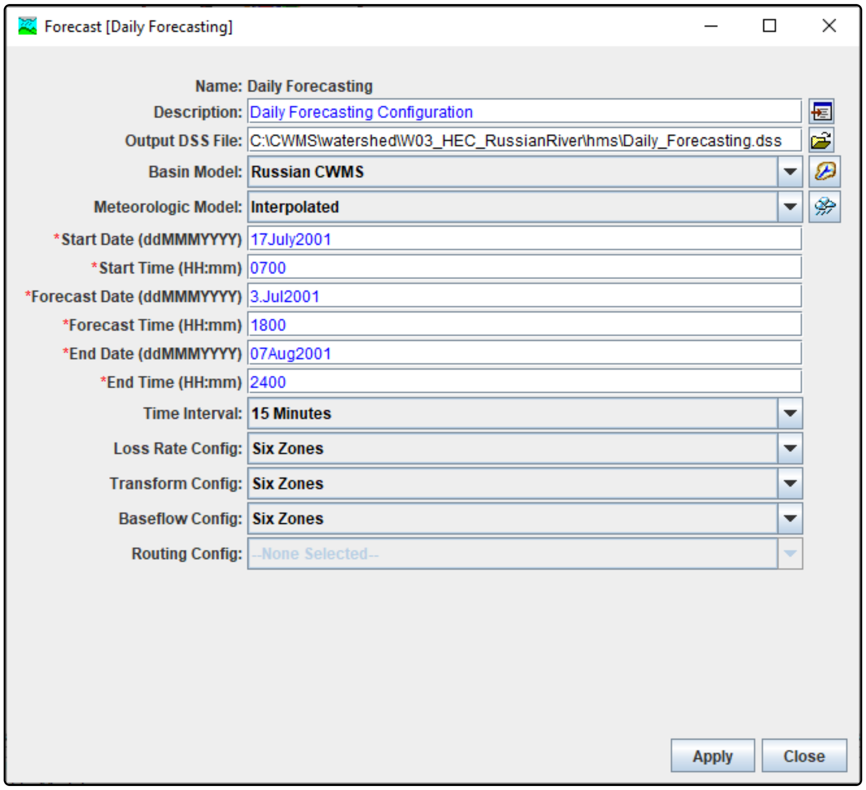
Once loss parameter zones have been created, you can assign elements to the zones.

To assign elements to zones:

1. From the HEC-RTS main window, click the **Setup** tab, from the watershed tree, from the **Model** folder, expand **HMS**. Right-click on an HEC-HMS model alternative. From the shortcut menu, click **Zone Configurations**. The **Zone Configuration Manager** opens.
2. From the **Current zone configurations** box, select a zone configuration. Click **Elements**, the **Elements** dialog opens. From the **Zones** list, select the zone that you want to add elements to.
3. Select the elements (subbasins or reaches) you want to add to the zone from the elements (subbasins or reaches) listed in the **Unassigned Subbasins** list. Select one subbasin by clicking on that subbasin; select ranges of subbasins by shift-clicking; select discontinuous groups of subbasins by control-clicking.
4. Click **Add** and the selected element names display in the box under the **Zones** list. You can also move a subbasin into a zone by double-clicking on the element in the **Unassigned Subbasins** list.
5. You can move elements out of a zone by selecting an element in the box under the **Zones** list and then clicking **Remove** or by double-clicking on that element.
6. Close the **Elements** dialog, close the **Zone Configuration Manager** and **Save** your watershed.

20.5.3 Forecast for an HEC-HMS Alternative

From the **Forecast Alternative** editor (Figure 3), you can select the zone configuration for the loss rate, transform, baseflow and routing configuration parameters.



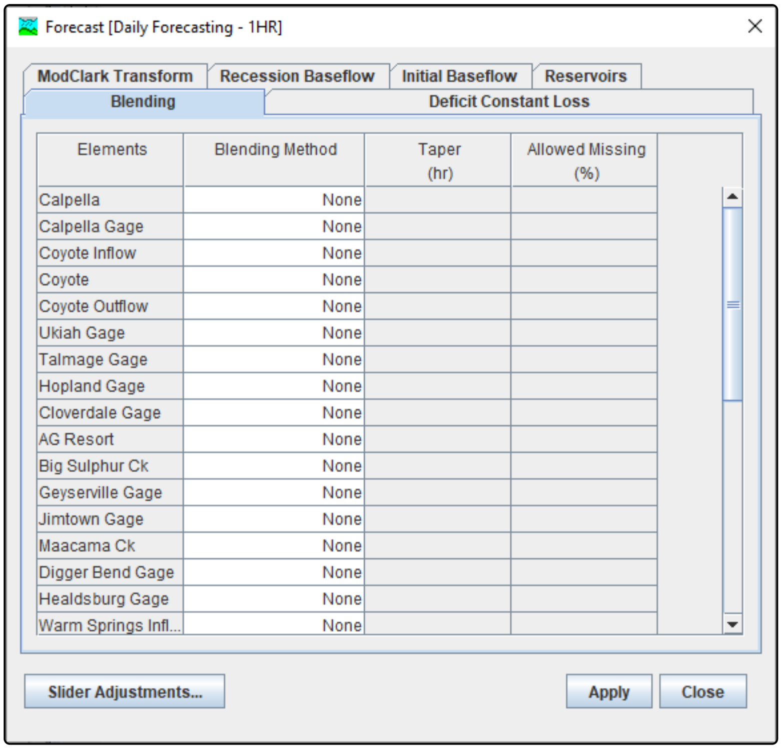
103 Figure 3 HEC-HMS - Forecast Alternative Editor

To set the forecast configuration and zone configuration parameters:

1. From the HEC-RTS main window, click the **Setup** tab, from the watershed tree, from the **Model** folder, expand **HMS**. Right-click on an HEC-HMS model alternative. From the shortcut menu, click **Forecast Configuration**. The **Forecast Alternative** editor (Figure 3) opens.
2. Enter a description in the **Description** field. Enter date and time values in for the **Start Date, Forecast Date**, and **End Date**. From the **Time Interval** list, select a time interval
3. From the **Loss Rate Config**, **Transform Config**, **Baseflow Config**, and **Routing Config** lists, select the appropriate zone configuration.
4. Click **OK**, and the **Forecast Alternative** editor closes (Figure 3). **Save** your watershed.

20.5.4 Adjust Parameters Using the Combined Parameter Editor

The **Combined Parameter Editor** (Figure 4) allows you to adjust loss rate, transform, and baseflow parameters to calibrate your HEC-HMS model to the forecasted event. It also controls **blending** (the substitution of observed flows for calculated flows in the model) for the HEC-HMS model. An HEC-HMS alternative consists of an HEC-HMS basin model, which sets initial values for parameters, and a set of adjustments to those initial parameter values. The methods for the parameters are based on what was specified in the HEC-HMS model.



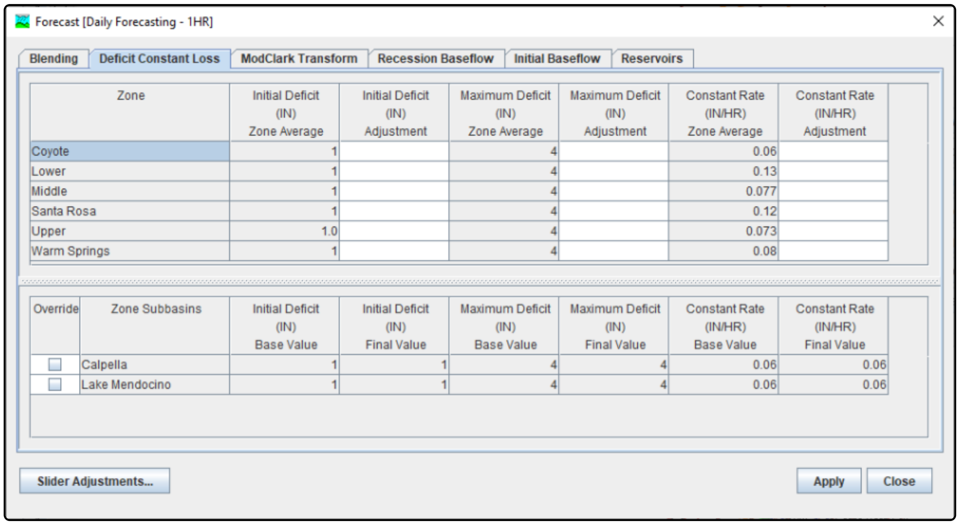
104 Figure 4 HEC-HMS - Combined Parameter Editor - Blending Tab

To adjust combined parameters:

1. From the HEC-RTS main window, click the **Setup** tab, from the watershed tree, from the **Model** folder, expand **HMS**. Right-click on an HEC-HMS model alternative. From the shortcut menu, click **Combined Parameter Editor**. The **Combined Parameter** editor (Figure 4) opens.
2. Click on the **Blending** tab, adjust the loss rate, transform, and baseflow parameters as needed. Click **Apply**, the **Combined Parameter** editor will close, save your HEC-RTS study.

20.5.4.1 Loss Parameters

To adjust loss parameters, from the **Combined Parameter** editor (Figure 5), click on the **Deficit Constant Loss** tab (name is dependent on the loss method selected). For example, in Figure 5, the loss method used was **Deficit** and **Constant**.



The upper panel shows a list of the loss zones in the watershed and the zone-average values for the initial soil-moisture deficit, the maximum deficit, and the constant loss rates in the HEC-HMS alternative. In the upper panel, for each parameter, there is a **Zone Average** column and an **Adjustment** column. The leftmost column displays the area-weighted average value of the parameter over the subbasins in the zone as read from the HEC-HMS basin model. To override a zone value, enter the new value in the Adjustment column.

In the lower panel of the editor are the parameters for the individual subbasins in the zone. For each parameter there are two columns, the **Base Value** and **Final Value**. To edit a loss parameter, select the override box next to the zone subbasin. The **Final Value** cell becomes editable and you can enter your new value. The list in the lower panel will contain the subbasins in that zone.

20.5.4.2 Baseflow Parameters

To edit the baseflow parameters, click on the **Baseflow** tab (Figure 4) of the **Combined Parameter** editor. For this example, the baseflow method used was **Bounded Recession**. The baseflow panel uses the same parameter adjustment method as the loss tab to adjust the baseflow at the time of forecast (this is not the initial baseflow, but the initial baseflow minus the recession that occurs between the lookback time and the time of forecast) and the recession ratio.

20.5.4.3 Transform Parameters Tab

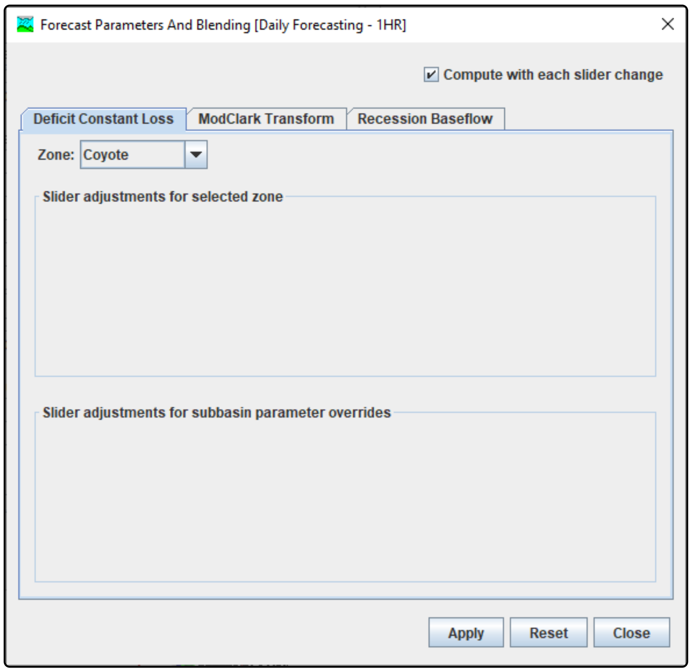
To edit the transform parameters, select the **Transform** tab (Figure 4) on the **Combined Parameter Editor**. For this example, the transform method used was **ModClark**. The transform panel uses the same parameter adjustment method as the loss tab to adjust the transform at the time of forecast.

20.5.4.4 Blending Tab

To edit blending parameters, select the **Blending** tab on the **Combined Parameter** editor (Figure 4). This contains a list of all locations in the HEC-HMS model where observed flows are available for blending. You can select blending methods (**Step** or **Taper**), set the taper interval, or turn blending off at each location.

20.5.4.5 Slider Adjustments

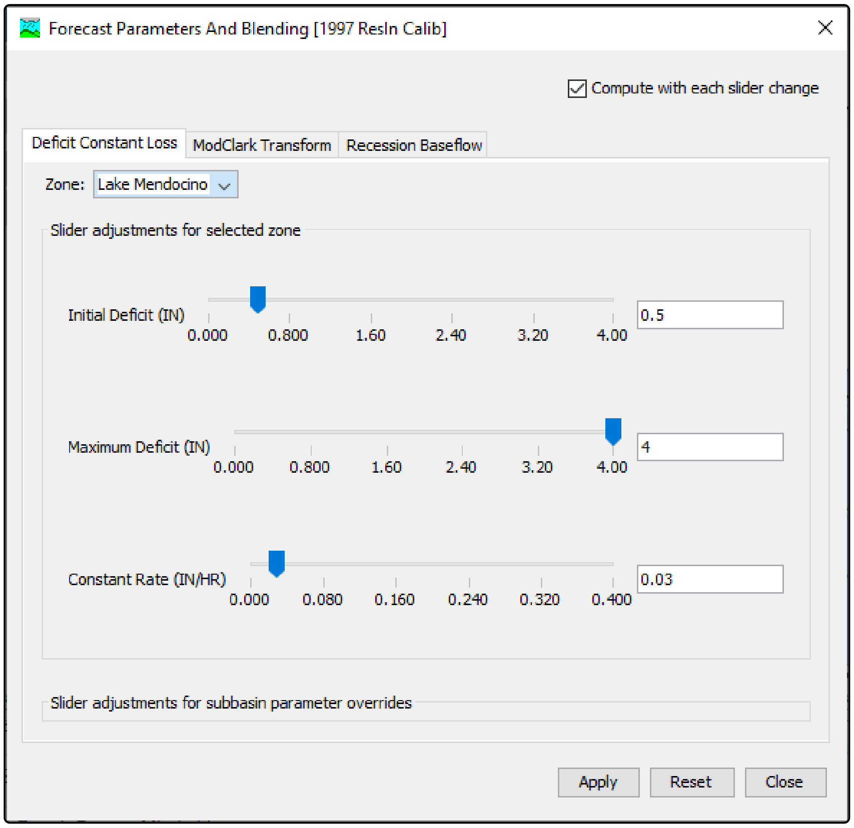
From the **Combined Parameter** editor (Figure 4), click **Slider Adjustments**, the **Forecast Parameters And Blending** dialog (Figure 6) will open. This allows you to adjust zonal, subbasin, and reach element parameters for calibration.



105 Figure 6 HEC-HMS - Forecast Parameters And Blending Dialog

20.5.5 Adjust Parameters Using Slider Adjustments

Sliders in HEC-HMS (Figure 7) allow you to adjust loss rate, transform, baseflow, and routing parameters easily. You adjust these parameters so that the simulated watershed response matches observed values (for example stream gages and reservoir stages) more closely during the lookback period of the event. An HEC-HMS alternative HEC-RTS consists of an HEC-HMS basin model, which sets initial values for parameters, a set of adjustments to those initial parameter values, and a metrological model that defines the precipitation input. The methods associated with the basin model parameters are based on what was specified in the HEC-HMS model alternative. The following sections describe setting up and using slider adjustments. Sliders depict a range of values, increments, or factors on a line and a selector that you can drag across the line to select a value, increment, or factor to adjust the parameter by. Sliders allow for quick, small adjustments of parameters and allow HEC-HMS to recompute runoff for a subset of basin elements instead of the entire watershed. This allows for faster computation when adjustments are made.



106 Figure 7 HEC-HMS - A Slider Configured to Adjust Loss Parameters

20.5.5.1 Configuring Slider Adjustments

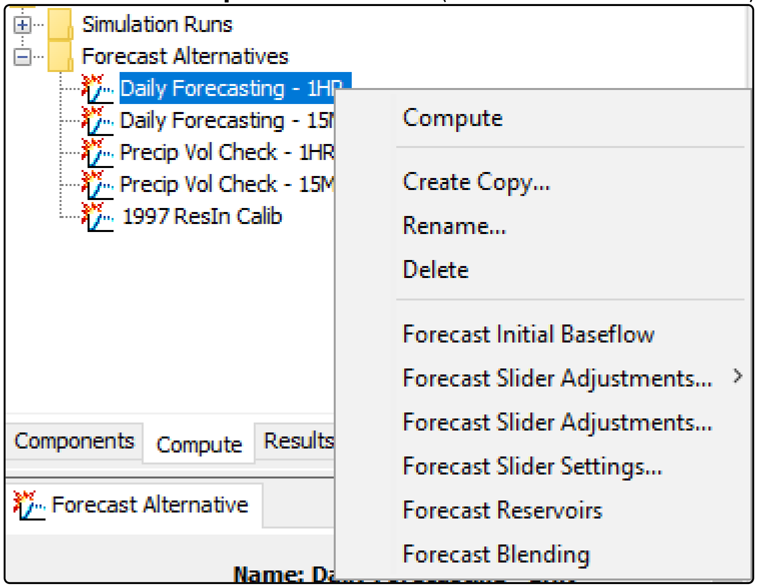
Before you can use slider adjustments, you must first configure slider adjustments. This is a two-step process: First, you must specify by zone and/or subbasin/reach which parameters you would like to adjust. Second, you must set the adjustment type (how the sliders will adjust the parameters) and limits (the maximum and minimum adjustments) of each slider.

20.5.5.2 Slider Adjustment Specification

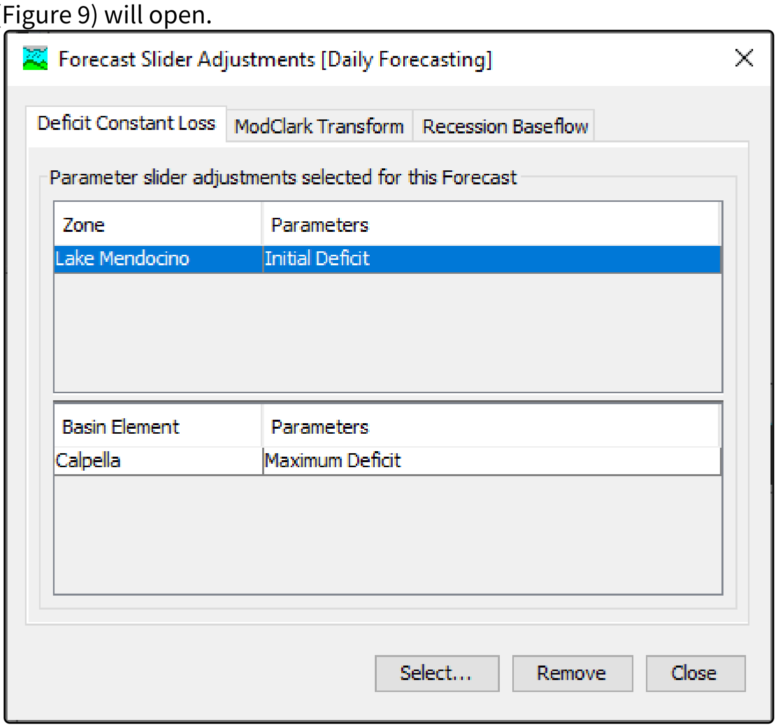
To specify the parameters, you would like to adjust:

1. From the HEC-RTS main window, from the main toolbar, click  . The HEC-HMS software application will display.

2. In the **Watershed Explorer** in HEC-HMS (set of tabs on the lower left), click the **Compute** tab (Figure 8).



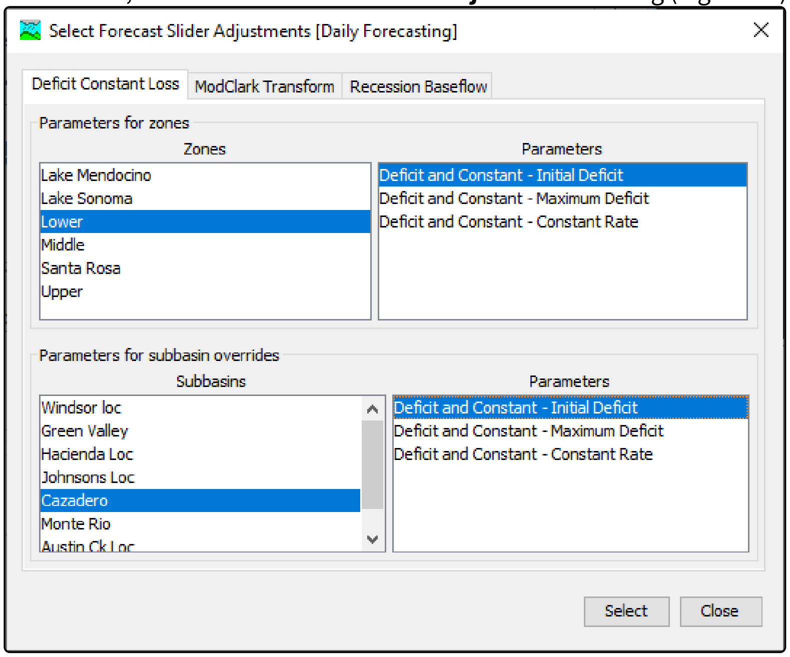
3. From the **Compute** tab, expand the **Forecast Alternatives** folder (Figure 8), and right-click on a forecast alternative for which you wish to configure slider adjustments.
4. From the shortcut menu (Figure 8), click **Forecast Slider Adjustments**, the **Forecast Slider Adjustments** dialog (Figure 9) will open.



5. On each tab of the **Forecast Slider Adjustments** dialog (Figure 9), the zone or subbasin/reach and the associated parameters are listed. The tabs in this dialog are named with the loss rate, transform, baseflow, and

routing methods used in the basin. (To be able to edit each type of parameter, the **Loss Rate Config**, **Transform Config**, **Baseflow Config**, and **Routing Config** zones must be defined in the HEC-HMS forecast alternative.)

6. Click **Select**, the **Select Forecast Slider Adjustments** dialog (Figure 10) will open.

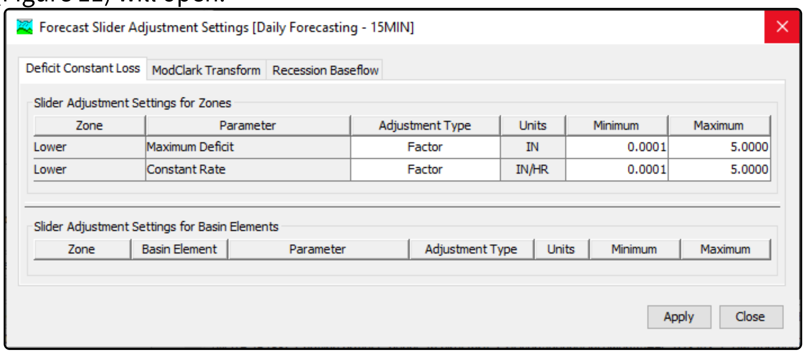


7. Select the tab with the parameters you wish to adjust. For example, in Figure 10, click the loss tab (**Deficit Constant Loss**) to select loss rate parameters for adjustment.
8. To select zonal adjustment sliders, in the **Parameters for Zones** panel (Figure 10), select a zone in the **Zones** box. This will populate the available parameters in **Parameters** box that can be assigned to sliders.
9. In the **Parameters** box (Figure 10), select one or more items you wish to adjust using sliders (you can choose multiple parameters by using the **CTRL** or **SHIFT** keys), click **Select**. The items selected in the **Parameters** box on the **Select Forecast Slider Adjustments** dialog (Figure 10) no longer display. The items now display on the **Forecast Slider Adjustments** dialog (Figure 9).
10. Repeat Steps 8 and 9 to select all desired zonal adjustment sliders.
11. Next, to select subbasin or reach adjustment sliders (for adjusting parameters for individual elements instead of zones), from the **Parameters for subbasin overrides** panel (Figure 10), select a subbasin or reach in the **Subbasins** box. This will populate the available **Parameters** box that can be assigned to sliders.
12. In the **Parameters** box (Figure 10), select one or more items you wish to adjust using sliders (you can choose multiple parameters using the **CTRL** or **SHIFT** keys), click **Select**. The items selected in the **Parameters** box on the **Select Forecast Slider Adjustments** dialog (Figure 10) no longer display. The items now display on the **Forecast Slider Adjustments** dialog (Figure 9).
13. Repeat Steps 11 and 12 to select all desired subbasin/reach adjustment sliders.
14. Repeat Steps 6 through 12 for each of the parameters you wish to adjust (loss rate, transform, baseflow, and routing methods). When you have finished selecting slider adjustments, from the **Select Forecast Slider Adjustments** dialog (Figure 10), click **Close**. The **Select Forecast Slider Adjustments** dialog will close.
15. Review your selections in the **Forecast Slider Adjustments** dialog (Figure 9). If you have selected one in error, you can remove it by selecting it in the **Parameters** column and clicking **Remove**. When you are satisfied with your selections, from the **Forecast Slider Adjustments** dialog, click **Close**. The **Forecast Slider Adjustments** dialog will close.

20.5.5.3 Slider Settings

In HEC-HMS parameters have minimum and maximum allowable values. These values are used to initialize the limits of the sliders. To setup the type of adjustment and limits of each slider:

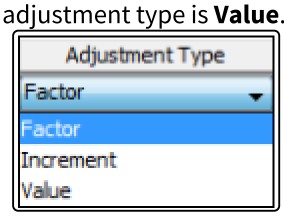
1. From the HEC-RTS main window, from the main toolbar, click . The HEC-HMS software application will display.
2. In the **Watershed Explorer** in HEC-HMS (set of tabs on the lower left), click the **Compute** tab (Figure 8).
3. From the **Compute** tab, expand the **Forecast Alternatives** folder (Figure 8), and right-click on a forecast alternative for which you wish to configure slider adjustments.
4. From the shortcut menu (Figure 8), click **Forecast Slider Settings**, the **Forecast Slider Adjustments** dialog (Figure 11) will open.



5. Each tab has two tables. The upper table shows zonal parameters, and the lower table shows parameters for individual elements. The **Zone** and **Subbasin/Reach** columns list the zone and subbasin/reach name. The **Parameter** column lists the parameter name of the slider. The **Adjustment Type** column lists the adjustment type. The **Units** column lists the units for the adjustment. The last two columns list the **Minimum** and **Maximum** values for the parameter adjustment.

To be able to configure loss rate, transform, baseflow, and routing slider settings, they must first be configured as detailed in the following section.

1. Specify an adjustment type using the dropdown in the **Adjustment Type** column (Figure 12). An **Adjustment Type** can be a new **Value**, an **Increment** to be added or subtracted from the parameter value, or a **Factor** to be applied to the parameter value. By default, the zonal adjustment type is **Factor**, and the subbasin/reach adjustment type is **Value**. If the adjustment type is **Factor**, the unit's entry will be blank.



2. Specify the minimum and maximum values for each parameter. These columns are populated by default. However, for practical use of the slider adjustments, the limits should be set to a smaller range.
3. Review your selections in the **Forecast Slider Adjustments Settings** dialog (Figure 11). When you are satisfied, click **Apply**, and then click **Close**. The **Forecast Slider Adjustments Settings** dialog will close, save the HEC-HMS project.

20.5.5.4 Adjusting Parameters Using the Sliders

From the HEC-RTS main window, click the **Setup** tab, from the watershed tree, from the **Model** folder, expand **HMS**. Right-click on an HEC-HMS model alternative. From the shortcut menu, click **Combined Parameter Editor**. The **Combined Parameter** editor (Figure 4 will open, click **Slider Adjustments**. The **Forecast Parameters And Blending** dialog will open with slider adjustments displayed (Figure 7).

The dialog (Figure 7) consists of multiple tabs, each containing sliders by zone. (Again, the tabs in this dialog are named with the loss rate, transform, baseflow, and routing methods used in the basin.) The zone is selected using the dropdown menu located just under the tabs. The upper part of the dialog shows parameter adjustments for the selected zone. The lower part of the dialog shows parameter adjustments for the individual subbasin or reach within the selected zone (if configured).

Parameter values can be adjusted by moving a slider or by entering a value in the text box to the right of the slider. If a value is entered in the text box, it should be consistent with the adjustment type. A factor must begin with an asterisk (*), an increment must begin with a plus (+) or a minus (-), and a value adjustment needs no initial symbol. The slider will move to the value entered in the text box. If the value in the text box is outside the slider's range, the slider is disabled.

In the upper right of the dialog (Figure 7) there is a check box labeled **Compute with each slider change**. If this box is checked, any change in a slider will cause the parameter adjustment to be applied, and the forecast alternative will be computed.

If the box is not checked, the parameter adjustment will not be applied until **Apply** is clicked, and the results will not be updated until the forecast alternative is computed. If you prefer, you can configure HEC-HMS to compute automatically after clicking **Apply**. In the HEC-HMS software, select **Program Settings** from the **Tools** menu. In the **General** tab, check the box next to **Compute after apply in editors**, and click **OK**. This setting applies to both the slider adjustments and combined parameter editor.

Clicking **Reset** will reset the parameter adjustments to their value when the dialog was opened. **Close** closes the dialog.

Right-clicking a slider (in HEC-RTS) brings up a menu with two options. The first option **Reset to Original Value** will reset the parameter adjustment to the original value when the dialog was opened. The second option **Modify Slider Limits** opens the **Forecast Slider Adjustments Settings** dialog, so the **Adjustment Type** and slider **Minimum** and **Maximum** limits can be changed.

20.5.6 Edit an HEC-HMS Alternative

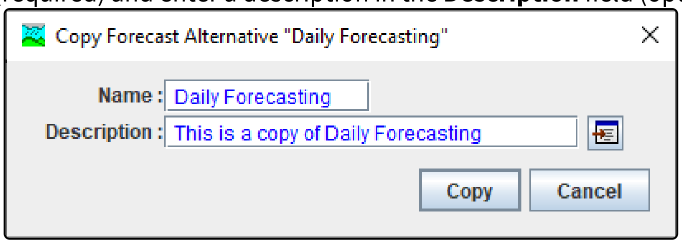
To edit an HEC-HMS model alternative:

1. From the HEC-RTS main window, click on the **Setup** tab, from the watershed tree, from the **Models** folder, expand **HMS**.
2. Right-click on an HEC-HMS model alternative, from the shortcut menu, click **Forecast Configuration**, the **Forecast Configuration** editor will open (Figure 3).
3. Edit the HEC-HMS model alternative as desired, click **Apply** button to save your changes. Click **Close**, the **Forecast Configuration** editor will close, save your HEC-RTS study.

20.5.7 Copy an HEC-HMS Alternative

To copy an HEC-HMS model alternative:

1. From the HEC-RTS main window, click on the **Setup** module, from the **Models** menu, point to **HMS**, click **Forecast Alternative Manager**, the **Forecast Alternative Manage**r dialog (Figure 3) opens.
2. From the **Current forecast alternatives** box, select the forecast alternative you wish to copy, click **Copy**.
3. The **Copy Forecast Alternative** dialog (Figure 13) opens. Enter a new alternative name in the **Name** field (required) and enter a description in the **Description** field (optional).



4. Click **Copy**, the **Copy Forecast Alternative** dialog will close. The new HMS alternative will now be display in the **Current forecast alternatives** box. Close the **Forecast Alternative Manager** dialog (Figure 3).

20.5.8 Delete an HEC-HMS Alternative

To delete an HEC-HMS alternative:

1. From the HEC-RTS main window, click on the **Setup** module, from the **Models** menu, point to **HMS,** click **Forecast Alternative Manager**, the **Forecast Alternative Manager** dialog (Figure 3) opens.
2. From the **Current forecast alternatives** box, select the forecast alternative you wish to delete, click **Delete**.
3. The **Delete Confirmation** window (Figure 14) opens, asking you if you wish to delete the selected HMS forecast alternative.

4. Click **OK**, the **Delete Confirmation** window will close. The HMS forecast alternative no longer displays in the **Current Forecast Alternatives** box. Close the **Forecast Alternative Manager** dialog (Figure 3).

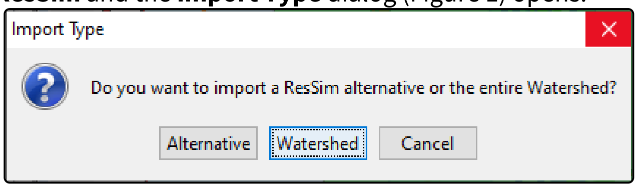
20.6 Create a HEC-ResSim Alternative

A HEC-ResSim model alternative consists of a watershed schematic, stream and reservoir network, and a set of operations for each reservoir. Within a HEC-ResSim alternative, you can override simulated forecasted releases to mimic desired operation during the forecast period. If the HEC-ResSim project contains more than one network alternative (for summer and winter conditions, for instance), you can build HEC-ResSim model alternatives around each of them.

20.6.1 Import an HEC-ResSim Watershed

To import an HEC-ResSim watershed:

1. From the HEC-RTS main window, click on the **Setup** module, from the **Models** menu, point to **Import**, click **ResSim** and the **Import Type** dialog (Figure 1) opens.

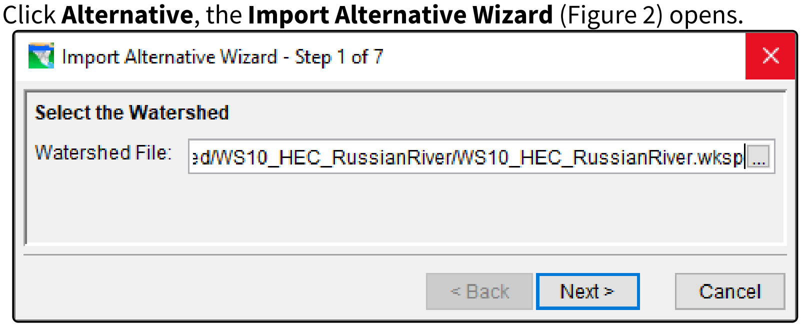


2. To import an HEC-ResSim watershed (stream alignment, network, configurations, and alternatives), click **Watershed**. The **Select Watershed File to Import From** will open.
3. Browse to the location of the HEC-ResSim watershed you want to import. Select an HEC-ResSim watershed file (***.wksp*), click *Open**. The **Select Watershed File to Import From** browser will close and the import will automatically begin. An **Import Progress** dialog opens indicating that the import is occurring.
4. The **Import Progress** dialog closes automatically once the import is complete and a **Message** window will display letting you know that the import is complete, save your HEC-RTS study.

20.6.2 Import an HEC-ResSim Alternative

To import an HEC-ResSim alternative:

1. From the HEC-RTS main window, click on the **Setup** module, from the **Models** menu, point to **Import**, click **ResSim**, the **Import Type** dialog (Figure 1) opens.
2. Click **Alternative**, the **Import Alternative Wizard** (Figure 2) opens.



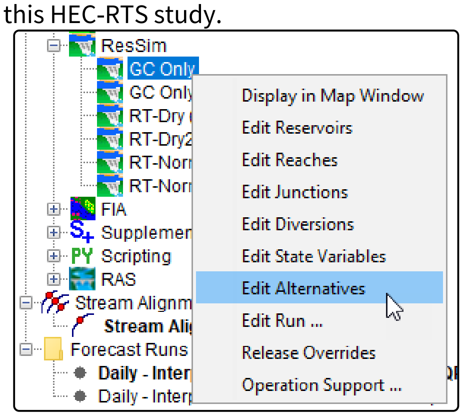
3. To select an HEC-ResSim watershed, to the right of the **Watershed File** box (Figure 2), click . An **Open File** browser will open, browse to the location of the HEC-ResSim watershed you want to import. Select a HEC-ResSim watershed file (***.wksp*), click *OK**. The **Open File** browser will close and the location of the selected HEC-ReSim watershed is displayed in the **Watershed File** box.
4. Click **Next,** select an HEC-ResSim alternative from the **Available Alternatives** table.
5. Click **Next**, if you want you can change the name of the selected HEC-ResSim alternative by entering a name in the **Name** field.
6. Click **Next**, if you want you can change the name of the selected HEC-ResSim reservoir network by entering a name in the **Name** field.
7. Click **Next**, the elements associated with the HEC-ResSim reservoir network are displayed along with stream assignments. Review this table, if you find any "To Stream" assignments that are incorrect, from the **To Stream** column in the table, select the correct stream assignment from the list.

8. Click **Next**, the elements associated with the HEC-ResSim reservoir network are displayed along with computation point assignments. Review this table, if you find any "Watershed Computation Point" assignments that are incorrect, from the **Watershed Computation Point** column in the table, select the correct computation point assignment from the list.\
9. Click **Next**, review the **Import Summary**, to make sure that everything is correct. If needed, you can copy or print the summary.
10. Click **Finish**, a **Continue with Import** window will display asking you if you wish to continue with the import operation. Click **Yes**, the **Continue with Import** window will close, and the HEC-ResSim import of an alternative will proceed. Once the import is finished, save your HEC-RTS watershed.

20.6.3 Edit a HEC-ResSim Alternative

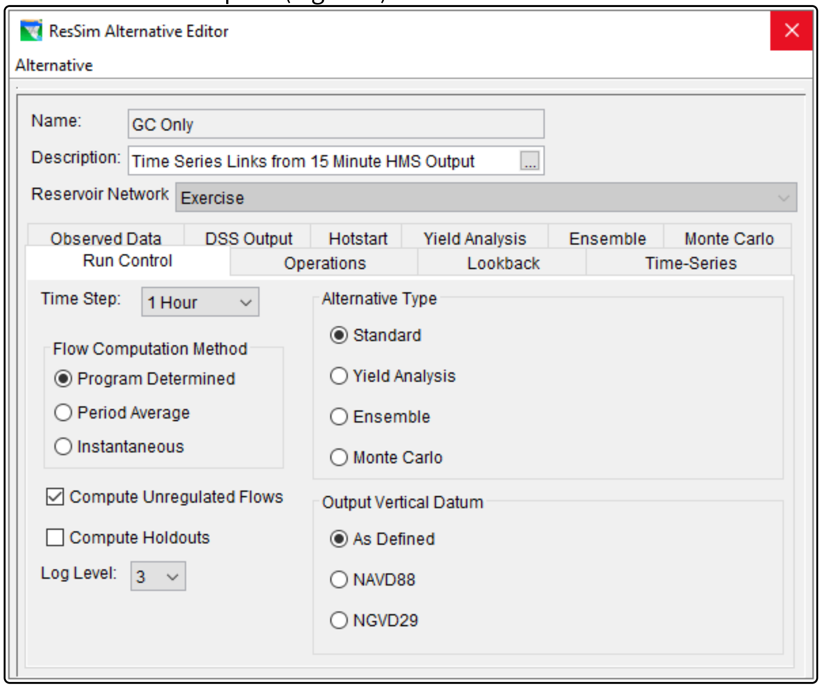
To edit a HEC-ResSim alternative:

1. From the HEC-RTS main window, click on the **Setup** module, from the **Watershed Tree**, from the **Models** folder, expand the **ResSim** node (Figure 3). This provides you with a list of all the available HEC-ResSim alternatives for this HEC-RTS study.



107 Figure 3 HEC-ResSim Model Alternative Shortcut Menu

2. Right-click on a HEC-ResSim alternative, from the shortcut menu (Figure 3), click **Edit Alternatives**, the **ResSim Alternative Editor** opens (Figure 4).



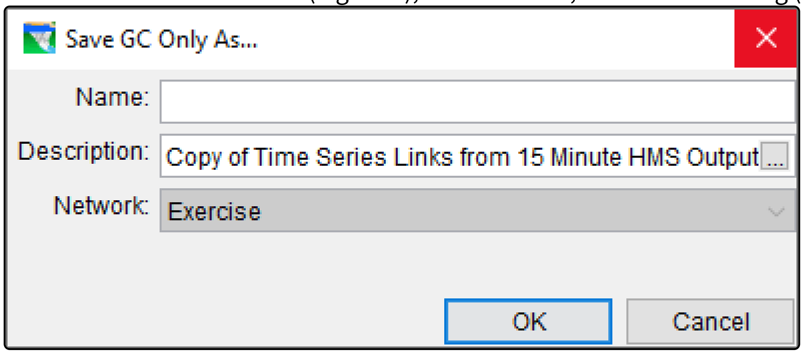
3. Edit the HEC-ResSim alternative as desired. For further information on how to modify an HEC-ResSim alternatives, refer to the HEC-ResSim User's Manual.
4. To save your changes, from the **Alternative** menu (Figure 4), click **Save**.
5. To close the **ResSim Alternative Editor**. From the **Alternative** menu, click **Close**, the **ResSim Alternative Editor** will close.

20.6.4 Copy a HEC-ResSim Alternative

To copy a HEC-ResSim alternative:

1. From the HEC-RTS main window, click on the **Setup** module, from the **Watershed Tree**, from the **Models** folder, expand the **ResSim** node (Figure 3). This provides you with a list of all the available HEC-ResSim alternatives for this watershed.
2. Right-click on an HEC-ResSim alternative, from the shortcut menu (Figure 3), click **Edit Alternatives,** the **ResSim Alternative Editor** opens (Figure 4).

3. From the **Alternative** menu (Figure 4), click **Save As**, the **Save As** dialog (Figure 5) opens.



4. Enter an alternative name in the **Name** field (required) and a description in **Description** field (optional).
5. Click **OK**, the **Save As** dialog closes. The new alternative is displayed in the **HEC-ResSim Alternative Editor**.
6. To close the **ResSim Alternative Editor**. From the **Alternative** menu, click **Close**, the **ResSim Alternative Editor** will close.

20.6.5 Delete an HEC-ResSim Alternative

To delete an HEC-ResSim alternative:

1. From the HEC-RTS main window, click on the **Setup** module, from the **Watershed Tree**, from the **Models** folder, expand the **ResSim** node (Figure 3). This provides you with a list of all the available HEC-ResSim alternatives for this watershed.
2. Right-click on an HEC-ResSim alternative the you want to delete, from the shortcut menu (Figure 19.40), click **Edit Alternatives,** the **ResSim Alternative Editor** opens (Figure 4).
3. From the **Alternative** menu (Figure 4), click **Delete**, a **Warning** window will display asking you if you really want to delete the selected HEC-ResSim alternative. Click **Yes**, the HEC-ResSim Alternative will no longer display in the **ResSim Alternative Editor**.
4. To close the **ResSim Alternative Editor**. From the **Alternative** menu, click **Close**, the **ResSim Alternative Editor** will close.

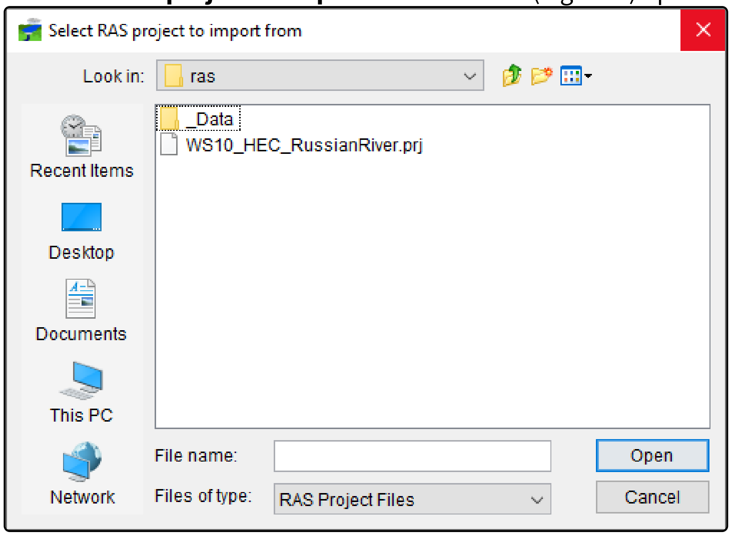
20.7 Create an HEC-RAS Alternative

A HEC-RAS alternative consists of model geometry, set of boundary conditions, and a plan file linking the two. If the HEC-RAS project contains more than one plan (for steady flow and unsteady flow conditions, for instance), you can build HEC-RAS model alternatives around each of them.

20.7.1 Import a HEC-RAS Alternative

To import a HEC-RAS model alternative:

1. From the HEC-RTS main window, click on the **Setup** module, from the **Models** menu, point to **Import**, click **RAS,** the **Select RAS project to import from** browser (Figure 1) opens.

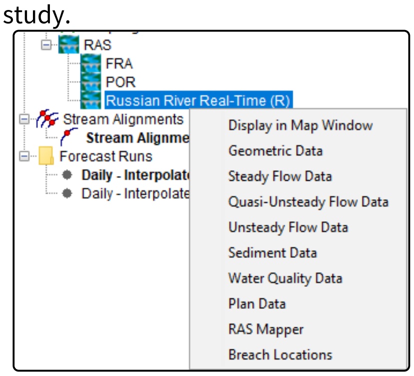


2. Browse to the location of an HEC-RAS project you wish to import. Select an HEC-RAS project file (***.prj*), click *****Open**, the **Select HEC-RAS project to import from** browser will close.
3. The import begins, an **Import Progress** window will open displaying the progress of the import. The **Import Progress** window will close when the import is complete, and a **Message** window will open letting you know the import is complete. Once the import is complete, save your HEC-RTS study.

20.7.2 Edit a HEC-RAS Alternative

To edit a HEC-RAS alternative:

1. From the HEC-RTS main window, click on the **Setup** module, from the **Watershed Tree**, from the **Models** folder, expand the **RAS** node (Figure 2). This provides you with a list of all the available HEC-RAS plans for the HEC-RTS study.

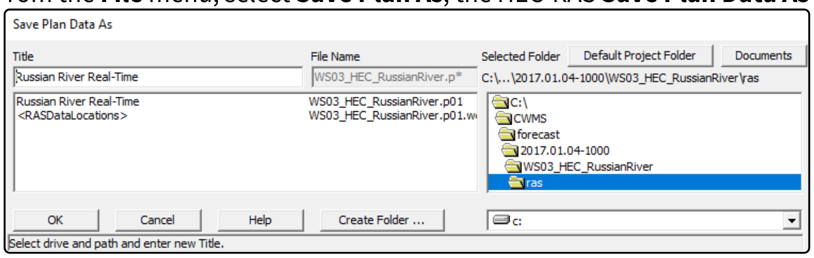


2. Right-click on an HEC-RAS alternative, a shortcut menu will display (Figure 2). From the shortcut menu, select the HEC-RAS component you wish to modify. Edit the HEC-RAS alternative as desired. For further information on how to modify HEC-RAS alternatives, refer to the HEC-RAS User's Manual.

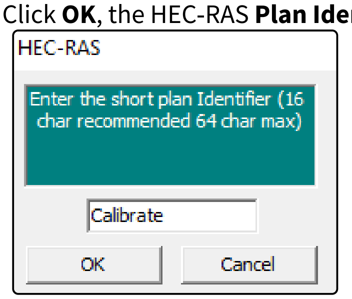
20.7.3 Copy a HEC-RAS Alternative

To copy a HEC-RAS alternative:

1. From the HEC-RTS main window, click on the **Setup** module, from the **Watershed Tree**, from the **Models** folder, expand the **RAS** node (Figure 2). This provides you with a list of all the available HEC-RAS plans for the HEC-RTS study.
2. Right-click on an HEC-RAS alternative, from the shortcut (Figure 2), click **Plan Data**. The HEC-RAS plan for the example watershed, is an unsteady flow plan, so the HEC-RAS **Unsteady Flow Analysis** dialog will open.
3. From the **File** menu, select **Save Plan As**, the HEC-RAS **Save Plan Data As** dialog (Figure 3) opens.



4. From the HEC-RAS **Save Plan Data As** dialog, specify a new plan name in the **Title** box (Figure 3).
5. Click **OK**, the HEC-RAS **Plan Identifier** dialog (Figure 4) opens.



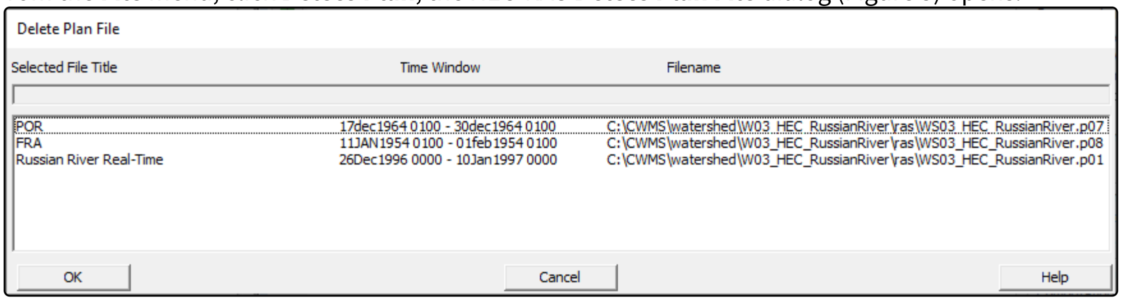
6. Enter an identifier in the HEC-RAS **Plan Identifier** dialog (Figure 4). Click **OK** to complete the copy process.

20.7.4 Delete a HEC-RAS Alternative

To delete a HEC-RAS alternative:

1. From the HEC-RTS main window, click on the **Setup** module, from the **Watershed Tree**, from the **Models** folder, expand the **RAS** node. This provides you with a list of all the available HEC-RAS plans for this watershed.
2. Right-click on an HEC-RAS alternative, from the shortcut (Figure 2), click **Plan Data**. The HEC-RAS plan for the example watershed, is an unsteady flow plan, so the HEC-RAS **Unsteady Flow Analysis** dialog will open.

3. From the **File** menu, click **Delete Plan**, the HEC-RAS **Delete Plan File** dialog (Figure 5) opens.



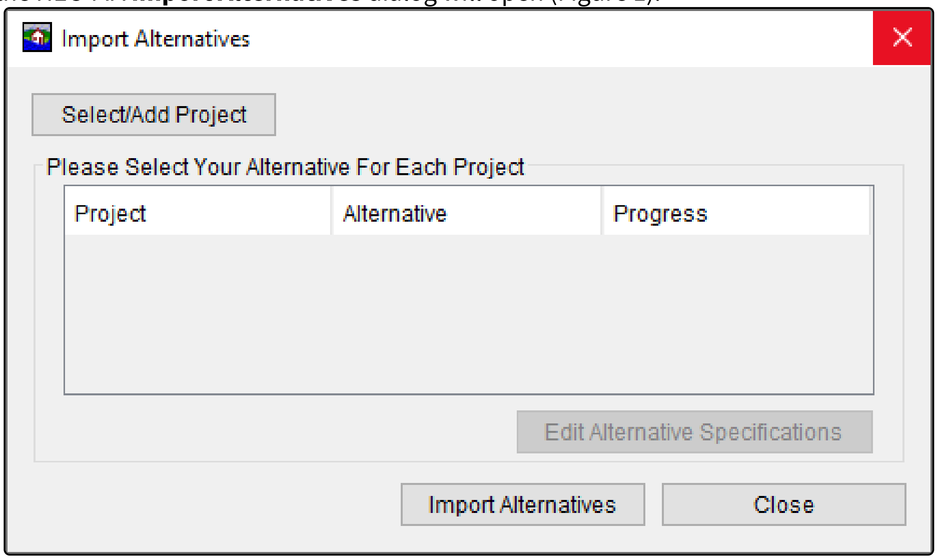
4. From the HEC-RAS **Delete Plan File** dialog, select the plan you want to delete, and click **OK**. The HEC-RAS **Delete Plan File** dialog will close.
5. A window will open asking you if you want to delete the selected plan. Click **OK** to complete the deletion process.

20.8 Create an HEC-FIA Alternative

An HEC-FIA alternative consists of information on consequences for a given hazard. An HEC-FIA alternative is a combination of an Inundation Configuration, Impact Areas, Structure Inventory, Critical Infrastructure, Impact Response Curves, Agricultural Data, and computation settings. If the HEC-FIA project contains more than one alternative (for summer and winter crop conditions, for instance), you can build HEC-FIA model alternatives around each of them.
For further information on how to develop the information required in defining an alternative in HEC-FIA, see the HEC-FIA User's Manual.

20.8.1 Import an HEC-FIA Alternative

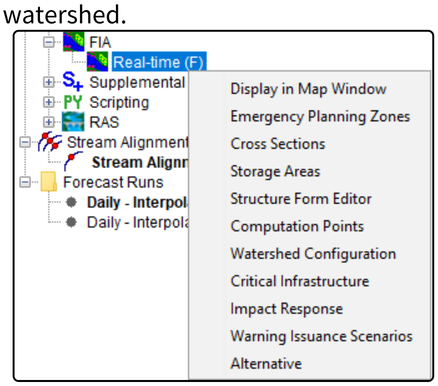
To import an HEC-FIA alternative:

1. From the HEC-RTS main window, click on the **Setup** module, from the **Models** menu, point to **Import,** click **FIA,** the HEC-FIA **Import Alternatives** dialog will open (Figure 1).


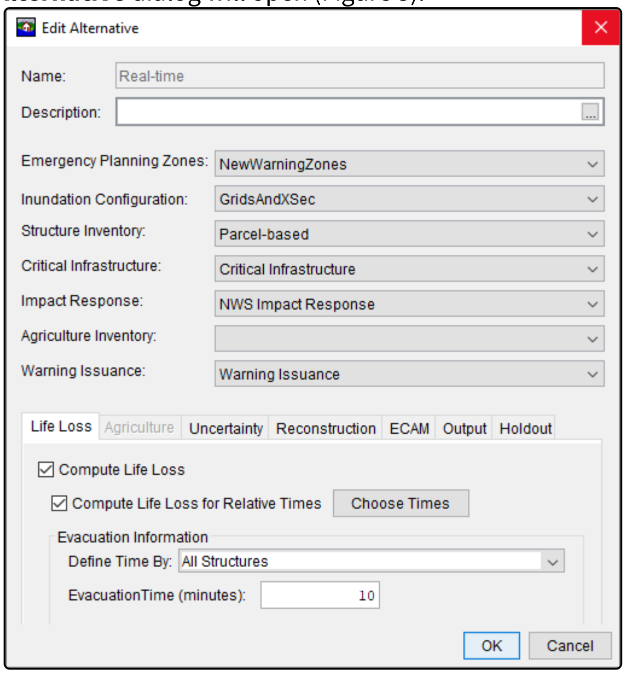
2. Click **Select/Add Project**, an **Open** browser will open. Browse to the location of an HEC-FIA watershed you wish to import. Select an HEC-FIA project file (***.prj*), click *Open**, the **Open** browser will close.
3. From the HEC-FIA **Import Alternatives** dialog (Figure 1), the table will display available HEC-FIA alternatives. Select the HEC-FIA alternative you wish to import. click **Import Alternatives**. Once the import is finished, confirmation dialog will open. Click **OK**, and the confirmation dialog will close.
4. From the **Import Alternatives** dialog (Figure 1), click **Close**. The **Import Alternatives** dialog will close, and you should save your HEC-RTS study.

20.8.2 Edit an HEC-FIA Alternative

To edit an HEC-FIA alternative:

1. From the HEC-RTS main window, click on the **Setup** module, from the **Watershed Tree**, from the **Models** folder, expand the **FIA** node (Figure 2). This provides you with a list of all the available HEC-FIA alternatives for this watershed.


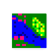
2. Right-click on an HEC-FIA alternative, from the shortcut menu (Figure 2), click **Alternative**. The HEC-FIA **Edit Alternative** dialog will open (Figure 3).

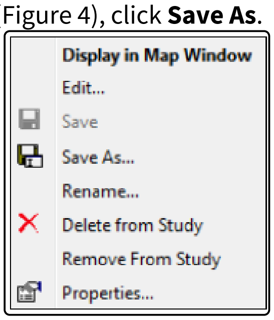


3. Edit the HEC-FIA alternative as desired. For further information on how to modify HEC-FIA alternatives, see the HEC-FIA User's Manual.
4. Click **OK**, the HEC-FIA **Edit Alternative** will close. Be sure to save your HEC-RTS study.

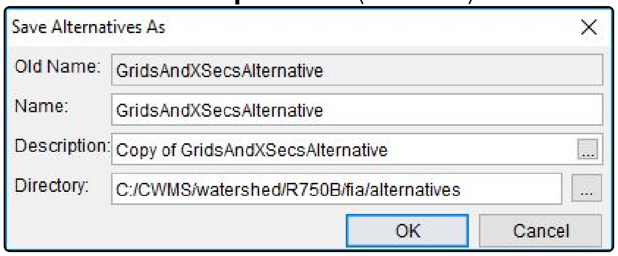
20.8.3 Copy an HEC-FIA Alternative

To copy an HEC-FIA alternative:

1. From the HEC-RTS main window, from the **Modeling Toolbar**, click  . The HEC-FIA software will be launched.
2. From the **HEC-FIA** main window, from the **Study Tree**, expand the **Alternatives** folder, right-click on the HEC-FIA alternative you wish to make a copy of. Right-click on the HEC-FIA alternative, from the shortcut menu (Figure 4), click **Save As**.



3. The HEC-FIA **Save Alternatives As** dialog will open (Figure 5). Specify a new alternative name in the **Name** field and fill out the **Description** field (if desired). Leave the **Directory** field alone.

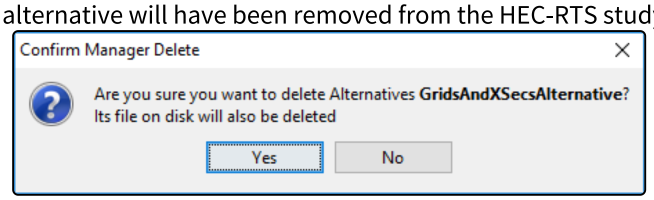


4. Click **OK**, to complete the copy process. The copied alternative will now be shown in the **HEC-FIA Alternative** editor.

20.8.4 Delete an HEC-FIA Alternative

To delete an HEC-FIA alternative:

1. From the HEC-RTS main window, from the **Modeling Toolbar**, click  . The HEC-FIA software will be launched.
2. From the **HEC-FIA** main window, from **Study Tree**, expand the **Alternatives** folder, right-click on the HEC-FIA alternative you wish to delete. Right-click on the HEC-FIA alternative, from the shortcut menu (Figure 4), click **Delete from Study**.
3. An HEC-FIA **Confirm Manager Delete** window will open (Figure 6), asking do you really want to delete the selected HEC-FIA alternative. Click **Yes**, the HEC-FIA **Confirm Manager Delete** window, and the HEC-FIA alternative will have been removed from the HEC-RTS study.



For further information on how to delete alternatives in HEC-FIA, see the HEC-FIA User's Manual.

20.9 Editing a Model Alternative

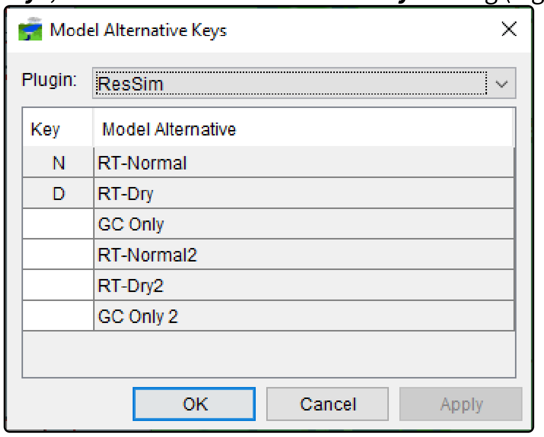
You can edit a model alternative by selecting the alternative you wish to edit in the watershed pane and launching the associated software from the HEC-RTS main toolbar. This will open the alternative in the native model and allow you to edit the model alternative. If you edit the model alternative in this manner and you have configured one or more forecast runs using this alternative you will need to make sure your changes propagate through to your forecast run configurations as detailed in HEC-RTS Forecasts.

20.10 Configuring Model Alternative Keys

To complete configuration of a model alternative, you must assign each model alternative a **Model Alternative Key**. The **Model Alternative Key** is an alpha-numeric identifier that will determine the names of your forecast runs.

To assign model alternative keys:

1. From the HEC-RTS main window, click on the **Setup** module, from the **Models** menu, click **Model Alternative Keys**, and the **Model Alternative Keys** dialog (Figure 1) opens.



2. From the **Plugin** list (Figure 1), select the model for which you wish to assign a key.
3. In the **Key** field (Figure 1), enter a single unique alpha-numeric character that will be used to identify the listed **Model Alternative**.
4. Repeat Steps 2 and 3 for all models and model alternatives. Click **OK,** and the **Model Alternative Keys** dialog (Figure 1) closes.

Model alternatives for the standard software applications in HEC-RTS might be named like the following:

MFP deals with future precipitation. You can configure model alternatives for use in conjunction with HEC-HMS to compute a variety of future precipitation scenarios. For example:

Example Model Alternatives for MFP	
ID	**Name**
N	No future precipitation
2	2-inches future precipitation
4	4-inches future precipitation
Q	QPF
R	One-half QPF
S	Twice QPF

HEC-HMS model alternatives simulate watershed response to precipitation. You will use HEC-HMS model alternatives to forecast future flows. For example:

Example Model Alternatives for HEC-HMS	
ID	**Name**
N	Normal conditions
W	Wet basin conditions
D	Dry basin conditions

HEC-ResSim model alternatives assist with reservoir operations, consisting of the physical data representing a river-reservoir system and operating rules for all controllable elements in the system. For example:

Example Model Alternatives for HEC-ResSim	
ID	**Name**
N	Normal operations
R	Restricted downstream channel capacity
W	White water release schedule
S	Fish spawning release schedule

HEC-RAS model alternatives describe water surface profile and stages at gage locations, geometry information for a river, and defined flow data sets. For example:

Example Model Alternatives for HEC-RAS	
ID	**Name**
N	Normal river conditions
H	High channel roughness
L	Low channel roughness
B	Bridge out

HEC-FIA model alternatives allow you to analyze flood consequences. Its model forecast alternatives are a combination of impact area and hydrograph time series sets. For example:

Example Model Alternatives for HEC-FIA	
ID	**Name**
N	Normal conditions
E	Evacuated with warning

You can edit the parameters within each HEC-RTS model alternative to reflect changing conditions. The beginning of this chapter describes how to edit model alternatives for the HEC-RTS software applications.

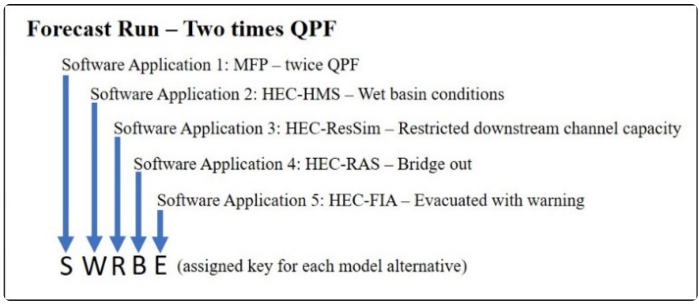
20.11 Forecast Run

A **forecast run** is the set of model alternatives that run sequentially during a forecast. As the with model alternatives, forecast runs exist independent of time. Therefore, you can use a forecast run for any forecast time window.
You create a forecast run by selecting a model alternative for each HEC-RTS software application. There are five standard software applications in HEC-RTS, a typical forecast run will include a combination of the five model alternatives. When you execute a forecast, HEC-RTS executes the software applications in a sequential order (program order).

The table shows three suggested forecast runs. The codes are the names HEC-RTS automatically assigns to each forecast run based on the one-character Model Alternative Keys you have assigned to the model alternatives. This coding is *program order dependent*.

Example Forecast Run	
Name	**Code**
No future precip	NNNNN
QPF	QNNNN
Two times QPF	SWRBE

In the following example, the forecast alternative named *Two times QPF* has the code *SWRBE*. Each character in this code is the ID of a specific model alternative for each of the five standard software applications in HEC-RTS. The code translates as follows:

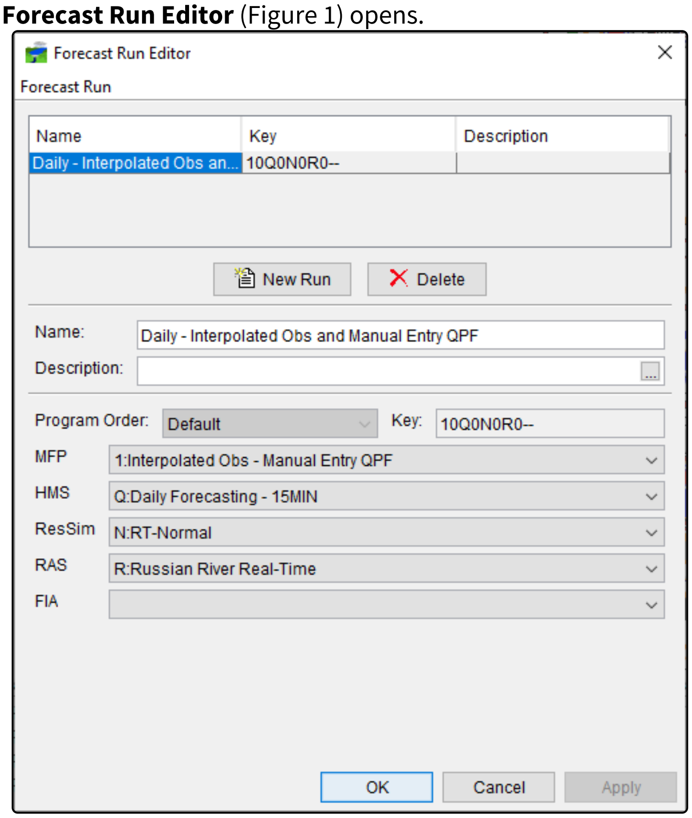


20.11.1 Creating Forecast Run

Before this step can be done, you must have created model alternatives.

Creating forecast runs:

1. From the HEC-RTS main window, click on the **Setup** tab, from the **Models** menu, click **Forecast Runs**, and the **Forecast Run Editor** (Figure 1) opens.



2. Click **New Run** (Figure 1), enter a new forecast run name in the **Name** field (required), and a description in the **Description** field (optional).
3. Select program order from the **Program Order** list (Figure 1). See Chapter 16 for information on program order.
4. From each program list (Figure 1), select the model alternatives for the forecast run.
5. From the **File** menu, click **Save**. The new forecast alternative displays in the table on the **Forecast Run Editor** (Figure 1).

6. Click **OK**, and the **Forecast Run Editor** closes. Be sure to save your HEC-RTS study.

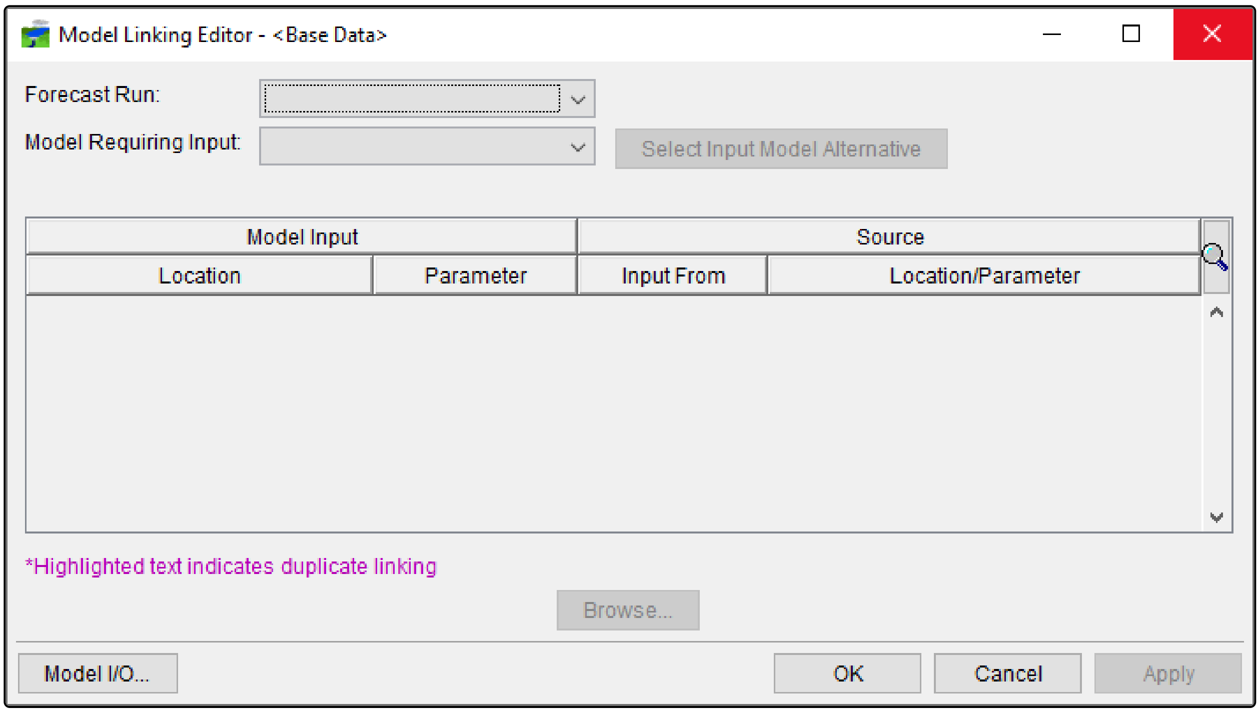
20.11.2 Delete a Forecast Run

Delete a forecast run:

1. From the HEC-RTS main window, click on the **Setup** tab, from the **Models** menu, click **Forecast Runs**, and the **Forecast Run Editor** (Figure 1) opens.
2. From the table, select the forecast run you want to delete. From the **Forecast Run** menu, click **Delete**. A **Delete Forecast Runs** window will open, asking do you really want to delete the selected forecast run.
3. Click **Yes**, the **Delete Forecast Runs** window will close, and the forecast run will be removed from the table on the **Forecast Run Editor**.
4. Click **OK** and the **Forecast Run Editor** closes. Be sure to save your HEC-RTS study.

21 Model Linking Editor

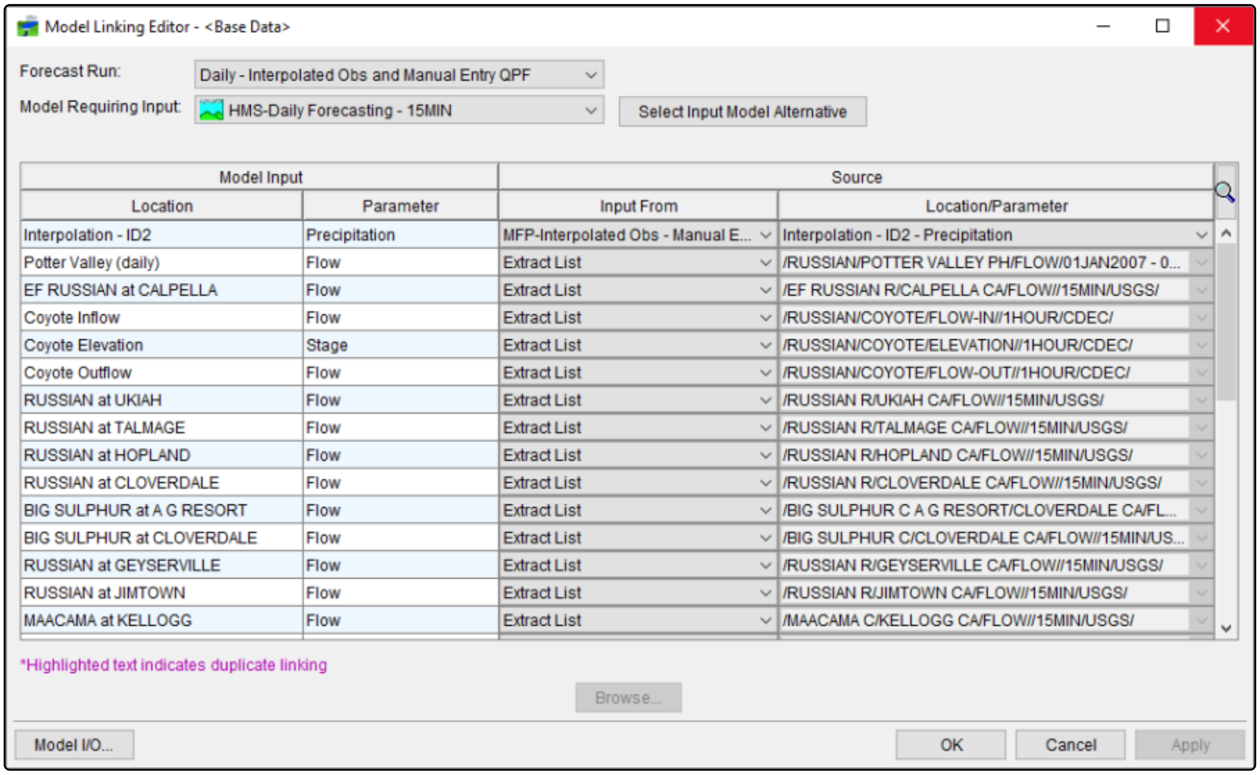
After forecast runs have been created, you need to link the model alternatives that are associated with a forecast run. For example, you might want to use output from MFP as input to HEC-HMS, and the output of HEC-HMS as the input to HEC-ResSim. This linking of the model alternatives is done using the **Model Linking Editor** (Figure 1). From the HEC-RTS main window, click the **Setup** tab, from the **Models** menu, click **Model Linking**, the **Model Linking Editor** will open (Figure 1).



108 Figure 1 Model Linking Editor

21.1 Model Linking Editor Overview

At the top of the **Model Linking Editor**, from the **Forecast Run** list (Model Linking Editor) you will select the forecast run that needs to be linked. The **Model Requiring Input** list (Model Linking Editor) displays the model alternatives configured for use in the selected forecast run. To complete the model linking process for a given forecast run, you will need to step through all the model alternatives listed in the **Model Requiring Input** list. The remaining portion of the **Model Linking Editor** is the table which lists the inputs required by the model alternative selected in the **Model Requiring Input** list and the source of that input. Figure 1 shows an example of the **Model Linking Editor** for the Russian River watershed with an HEC-HMS model alternative requiring input.



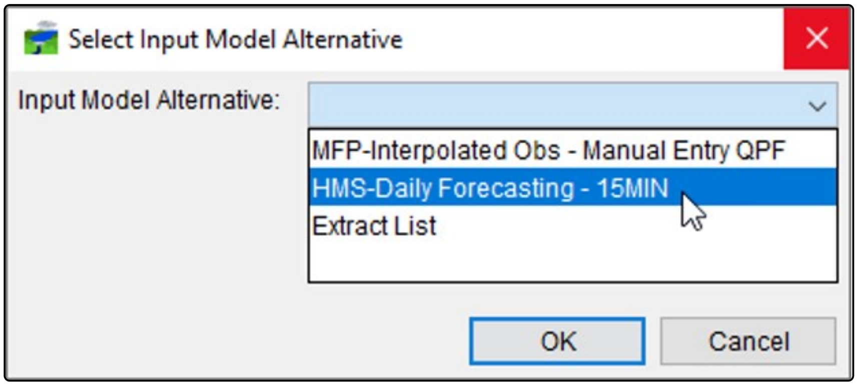
This table has four columns:

- **Location**. This column identifies the name of the required input. For example, this could be:
- The precipitation grid name or discharge gage name specified in the HEC-HMS model alternative.
- The local flow name specified in the HEC-ResSim model alternative.
- The river station and node name specified in the HEC-RAS model alternative.
- **Parameter**. This column identifies the type of data or information required by the model alternative. For example, this could be flow, stage, or precipitation.
- **Input From**. This column consists of lists that identify the source of the input. These lists display each available model alternative from the selected forecast runs and **Extract List**. **Extract List** is used to bring data and information from external sources (i.e., DSS files), into the watershed. Refer to Data Extract and Post Editors for additional information on the **Extract Editor**.
- **Location/Parameter**. If the **Input From** column displays the name of a model alternative, this column provides a list that contains all model alternative results for the parameter specified in **Parameter** column. For example, in Figure 1, the first row in the table indicates that the *Interpolation – ID2* precipitation input should come from the MFP model alternative named *MFP-Interpolated Obs – Manual Entry QPF*. If the **Input From** column displays *Extract List*, this column will display either a DSS pathname or a CWMS database pathname.

21.1.1 Select Input Model Alternative

The **Select Input Model Alternative** (Figure 2) provides a user with a way to set all the **Input From** values equal to a specific model alternative or *Extract List*. For example, you may want all the HEC-ResSim model alternative inputs to come from the HEC-HMS model alternative results. Click **Select Input Model Alternative**, the **Select Input Model**

Alternative dialog (Figure 2) opens. From the **Input Model Alternative** list (Figure 2), select a model alternative (or *Extract List*), click **OK**, the **Select Input Model Alternative** dialog (Figure 2) closes.

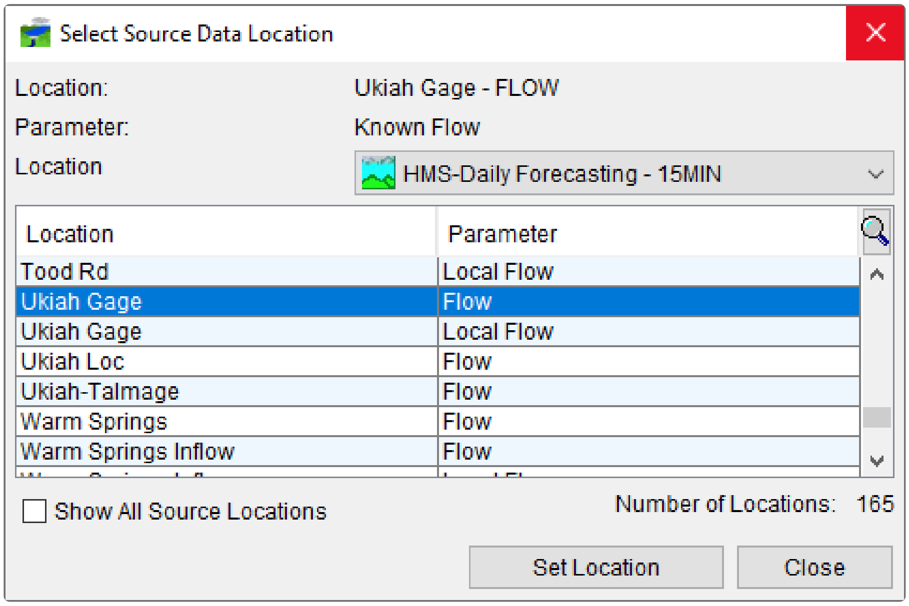


109 Figure 2 Select Input Model Alternative Dialog

A **Confirm Input Selection** window will open, asking do you want to link all unlinked rows to the selected **Input Model Alternative**. Click **Yes**, **the Confirm Input Selection** window will close. All rows in the table that were not linked, will now be linked to the selected **Input Model Alternative**.

21.1.2 Select Source Data Location

Are you having difficulty determining for a specific location what the correct parameter might be? From the **Model Linking Editor** (Figure 1), select a **Location** (i.e., *Ukiah Gage – FLOW*), click **Browse**, the **Select Source Data Location** dialog will open (Figure 3). In this dialog you can scroll through the entire list of available locations and parameters for the selected model alternative (*HMS-Daily Forecasting – 15 MIN*). Once you have found the correct location (i.e., *Ukiah Gage*), review the available parameters, select the row in the **Select Source Data Location** dialog (Figure 3) that contains the correct parameter (i.e., *Flow*). Click **Set Location** (Figure 3), for the selected **Location** in the **Model Linking Editor** (Figure 1), the **Location/Parameter** column will display the selected parameter. To close the **Select Source Data Location** dialog (Figure 3), click **Close.**



21.2 Model Linking Editor - Model Linking

To link the model alternatives, complete the following:

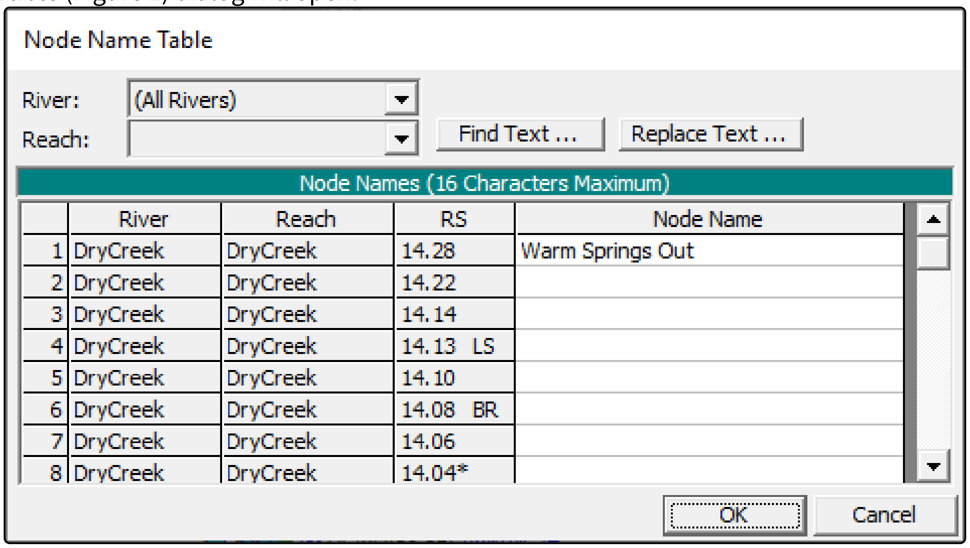
1. From the HEC-RTS main window, click the **Setup** tab, from the **Models** menu, click **Model Linking**, the Model Linking Editor will open.
2. Select the **Forecast Run** you wish to configure in the **Forecast Run** list (Example Linking).
3. Select the first model alternative in the **Model Requiring Input** list (Example Linking).
4. In the **Model Linking Editor** table (Example Linking), specify the **Input From** for each row as either a model alternative or *Extract List*.
5. For each row that has a model alternative selected in the **Input From** column, HEC-RTS has compiled a list of locations and parameters for the selected model alternative. To specify the **Location/Parameter** there are two ways, from the **Location/Parameters** cell, from the list, select the correct parameter. Another way to select the correct Location/Parameters, click **Browse**, the **Select Source Data Location** dialog will open (Model Linking Editor Overview).
6. Repeat steps 3 through 5 for each model alternative in the **Model Requiring Input** list (Example Linking).
7. Repeat all steps for each **Forecast Run** you wish to configure.
8. When you are finished with model linking, from the **Modeling Linking Editor** (Example Linking), click **OK**. The **Modeling Linking Editor** will close, and all changes will be saved.

21.3 Model Linking HEC-RAS Model Alternatives

The location listed in the **Model Linking Editor** for HEC-RAS inputs will be the river station and node name of the boundary condition location. Typically, your HEC-RAS model will not have node names configured and the location will display the river station. To configure HEC-RAS node names, complete the following:

1. From the CWMS CAVI main window, from the **Modeling Toolbar**, click  , the HEC-RAS software application will open.

2. From the HEC-RAS main window, from the **Edit** menu, click **Geometric Data**, the **Geometric Data Editor** will open.
3. From the **Geometric Data Editor**, from the **Tables** menu, point to **Names**, click **Node Names**. The **Node Name Table** (Figure 1) dialog will open.



4. Specify name for each river station with a boundary condition in the **Node Name** column. Hint: it may be useful to open the boundary condition editor to correlate river stations, node names, and boundary condition locations.
5. When you are finished, click **OK**, the **Node Name Table** dialog will close (Figure 1). From the **Geometric Data Editor**, from the **File** menu, click **Save Geometry Data**. To close the **Geometric Data Editor**, from the **File** menu, click **Exit Geometry Data Editor**, the **Geometric Data Editor** will close.
6. From the HEC-RAS main window, from the **File** menu, click **Save Project**. To close the HEC-RAS software application, from the **File** menu, click **Exit**.

22 Data Extract and Post Editors

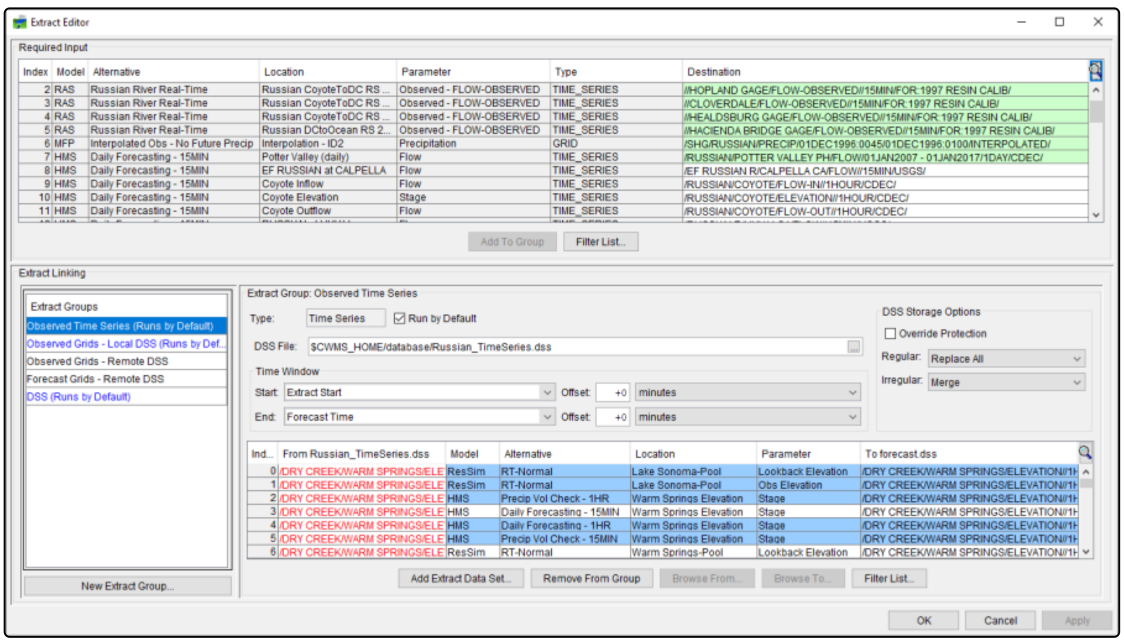
The extract and post processes denote the retrieval and dissemination of forecast data. The extract provides data to model alternatives that is required during a forecast compute. The post process disseminates forecast data after a forecast compute is complete. This chapter goes over the HEC-RTS tools used to set up and run extracts and posts.

22.1 Extracts

An **extract** refers to the process of obtaining data from DSS files or a WISKI database (KiWIS Write Extension) and mapping that data to a model alternative's required input record. The extract retrieves the source data and copies the data into records stored in a centralized DSS file called the *forecast.dss* file. The *forecast.dss* file is then used to provide input data during the forecast compute process. Extracts can contain both gridded data and time series data.

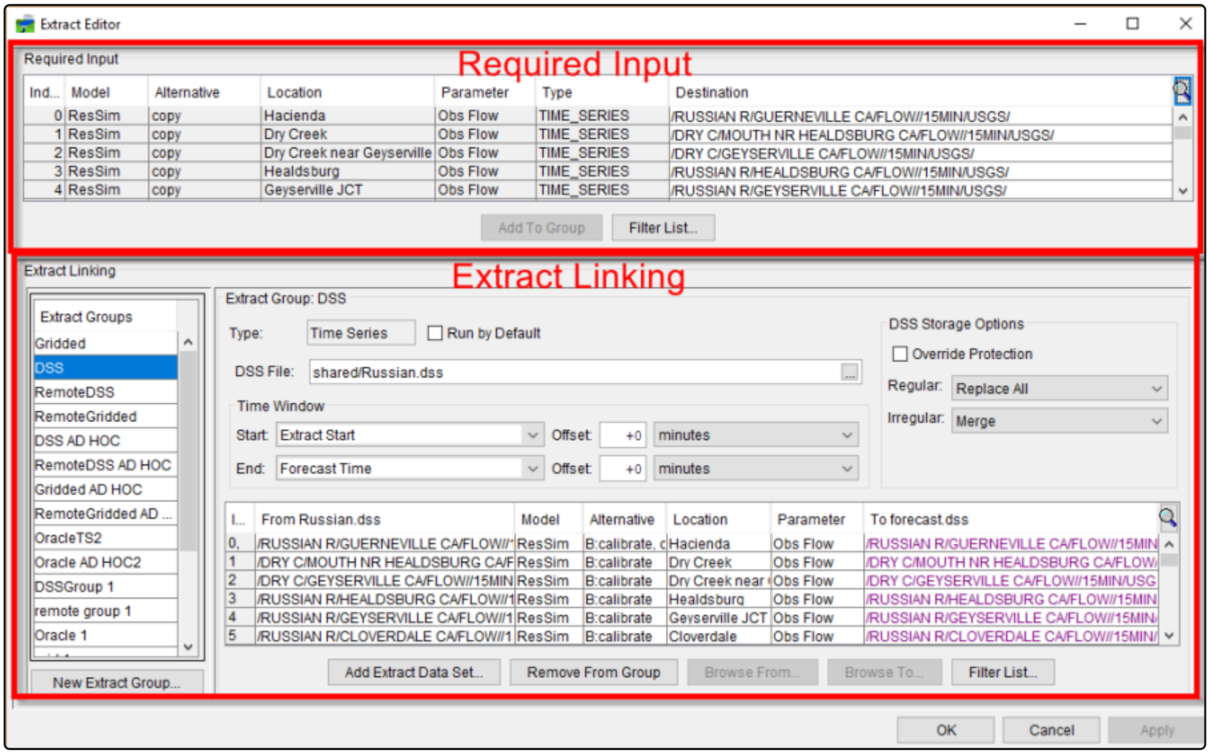
22.2 Extract Editor

After you have completed the model linking you will need to configure extracts that are required by model alternatives. Configuring an extract is completed using the **Extract Editor** (Figure 1).



The **Extract Editor** allows you to connect required model alternative input data from multiple sources to the *forecast.dss* file that provides input data to model alternatives during the compute process. The required input data is classified into extract groups that link the source data information to the required input records. To access the **Extract Editor**, from the HEC-RTS main window, click on the **Setup** tab, from the **Models** menu, click **Edit Extract**, the **Extract Editor** will open (Figure 1).

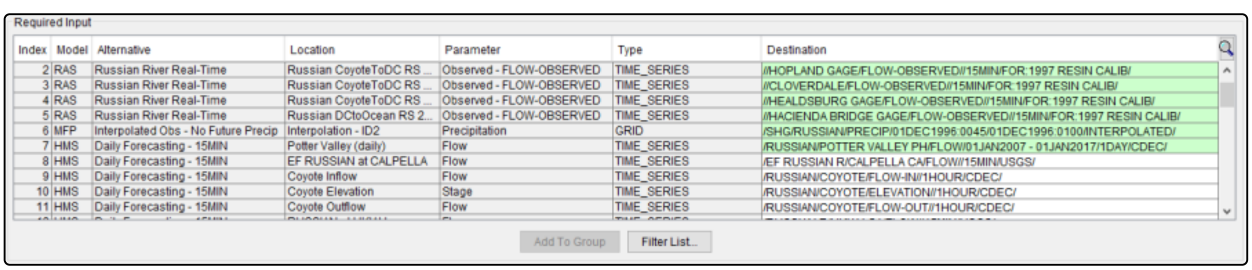
The **Extract Editor** is comprised of two sections: **Required Input** and **Extract Linking** (Figure 2). The **Required Input** section contains a table that lists the required input records for each model alternatives. The **Extract Linking** section contains information on defined extract groups, and contains a table, which lists the input records and source records assigned to an extract group.



110 Figure 2 Extract Editor - Sections

22.2.1 Required Input Section

The **Required Input** section (Figure 3) provides a list of the time series that needs to be mapped to a source record to provide input data for the forecast compute.



111 Figure 3 Extract Editor - Required Input Section

The **Required Input** section provides information on the following:

Model this column lists which software application is requesting the required input record.

Alternative this column lists the name of the model alternative that is requesting the input record.

Location this column displays the name of the model location that is associated with the input record.

Parameter this column displays the parameter associated with the input record.

Type this data column specifies if the record is times series data or gridded data. Types include: **GRID** or **TIME_SERIES**. **GRID** represents DSS gridded data. **TIME_SERIES** represents time series data.

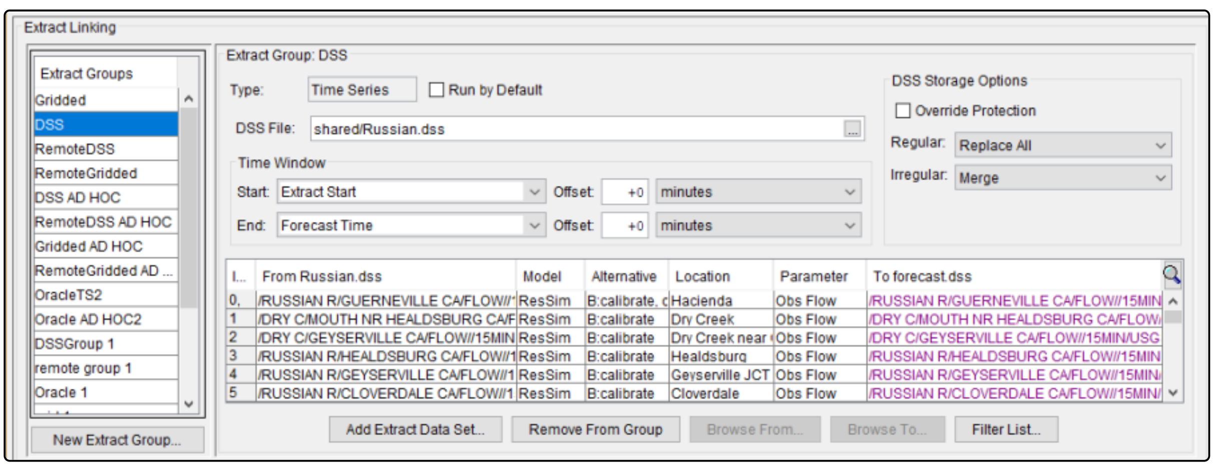
Destination this column displays the actual input record pathname that the model requires. This is the pathname that will be written to the *forecast.dss* file and used by the model during the compute.

Index Each required destination pathname is assigned an **Index** number when it is loaded into the **Extract Editor**. This number has no real usage except keeping the initial filtered list set. There can be multiple indices for a single DSS pathname.

This information does not display by default in the **Required Input** table, you will need to set up the table view, see Model Linking.

22.2.2 Extract Linking Section

The **Extract Linking** section (Figure 4), displays information on the extract groups. Extract groups link similar typed data (gridded or time series) to a single data source. The source can be a DSS file, a CWMS database, or a WISKI database (WISKI Extract and Post). The source records will be mapped to the required input records in the extract group. During the creation of a forecast, the source records will be copied into the required input pathnames when are created in the *forecast.dss* file.



112 Figure 4 Extract Editor - Extract Linking Section

The **Extract Linking** section provides information on the following:

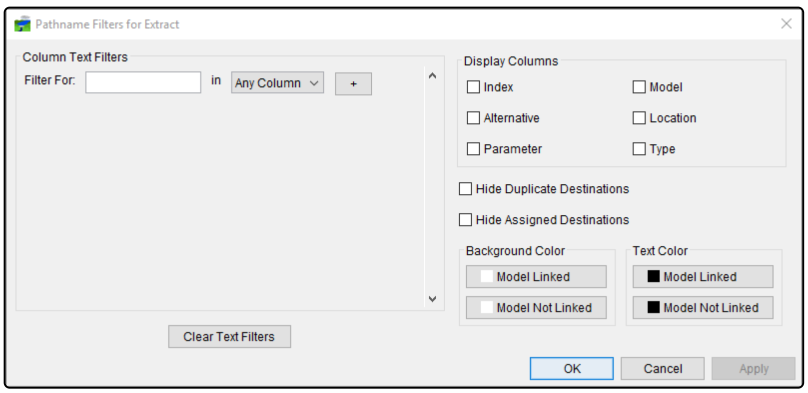
From <*data_source*>this column lists the source data pathnames that will fill the DSS record displayed in the **To forecast.dss** column (Figure 4).
Model this column lists which software application is requesting the required input record.
Alternativethis column lists the model alternative that is requesting the input record.
Location this column displays the location that is associated with the input record.
Parameter this column displays the parameter associated with the input record's data.
To forecast.dss this column displays the actual input record pathname that the model requires. This is the pathname that will be written to the *forecast.dss* file and used by the model during the compute.

Index each required destination pathname is assigned an index number (Figure 4) when it is loaded into the **Extract Editor**. This number has no real usage except keeping the initial filtered list set. There can be multiple indices for a single DSS pathname.

This information does not display by default in the **Extract Linking** table, you will need to set up the table view, see Model Linking.

22.2.3 Table View Options

You can change the amount of information displayed in the tables in both the **Required Input** and the **Extract Linking** sections. Clicking on the magnifying glass icon () at the top right-hand corner of the tables in each section. The **Pathname Filters for Extract** dialog (Figure 5) opens.



From the **Column Text Filters** box (Figure 5) you can filter the records to display only records that contain a certain phrase or string of text. Filtering can be done by **Column**, **Index**, **Model**, **Alternative**, **Location**, **Parameter**, **Type**, or **Destination**. You can filter for multiple requirements by clicking **+**. Below are examples of several different ways that you can filter:

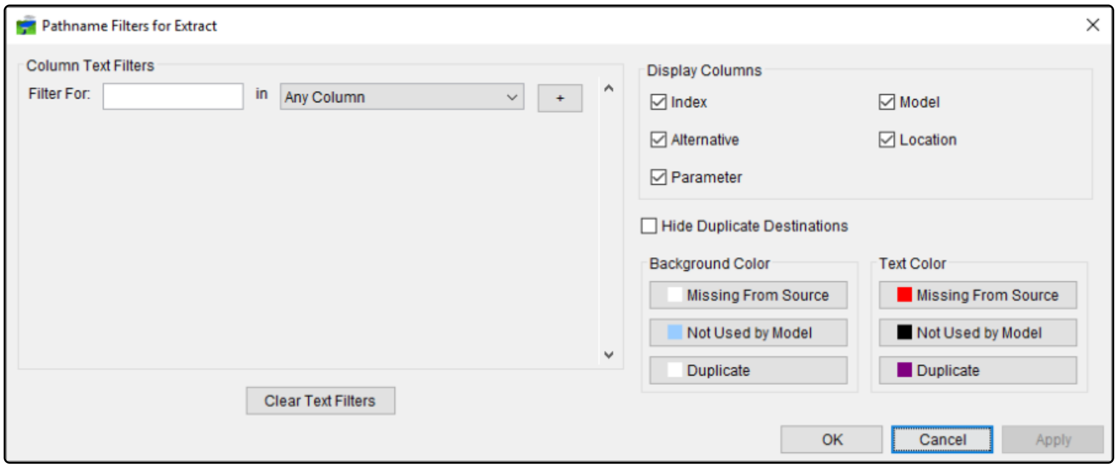
- Filters can be used to retrieve only records that are an exact match. You can do this by typing a string of text in the **Filter For:** box.
- Filters can be used to retrieve records that start, end, or contain a specific string. To do this, use an asterisk (*) after, before, or inserted into the string of text you want to filter for.

From the **Display Columns** box (Figure 5) you can select which columns in the table to view. For the examples in this chapter, **Index**, **Model**, **Alternative**, **Location**, **Parameters**, and **Type** have been selected for the tables in each section of the **Extract Editor**.

You can condense records that have the same destination to one row, selecting **Hide Duplicate Destinations** (Figure 5). To remove a record from a table that has already been assigned to an extract group, select **Hide Assigned Destinations** (Figure 5). The **Hide Assigned Destinations** option is not available for the **Extract Linking** section. For the examples in this chapter, neither **Hide Duplicate Destinations** or **Hide Assigned Destinations** has been selected.

You can also color code the destination records listed in the **Required Input** section (Figure 3) depending on if the destination record is linked to a source record. You can color code by background or text. For example, to add a background color to a record that is not linked, from the **Background Color** box (Figure 5), click **Model Not Linked**, a **Color Chooser** dialog will open (Using the Color Chooser). Select a color, click **OK**, the **Color Chooser** will close, and from the **Destination** column of the **Required Input** table, any record that has not been linked will be highlighted in the selected color (Figure 1).

You can also color code the records listed in the **Extract Linking** section (Figure 6) depending on if the destination record is missing from a source, not used by a model, or is a duplication. You can color code by background or text. For example, in Figure 6, to add a background color to a record that is not used by a model, from the **Background Color** box (Figure 6), click **Not Used by Model**, a **Color Chooser** dialog will open (Using the Color Chooser). Select a color, click **OK**, the **Color Chooser** will close, and from the **Extract Linking** table, any record that has not been used by a model will be highlighted in the selected color (Figure 1).

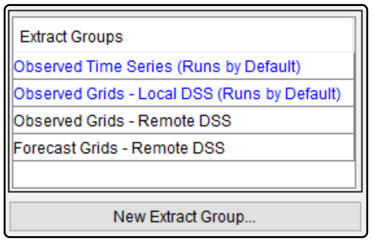


22.3 Extract Groups

Extract groups define the type of data and the source data that will be copied when an extract is running. Extract groups can be created for the following data sources:

- Local DSS time series file
- Remote DSS time series file
- Local DSS gridded data file
- Remote DSS gridded data file
- CWMS database
- WISKI database (KiWIS Write Extension)

From the **Extract Editor**, from the **Extract Linking** section, the **Extract Groups** box (Figure 1), lists the defined extract groups for a watershed. From the **Extract Groups** box, you can create an extract group; delete an extract group; rename an extract group; make a copy of an extract group; select which extract groups will run by default in a forecast compute; and, determine the order in which extract groups will run in a forecast compute. In addition, you can color code the extract groups that will run by default.

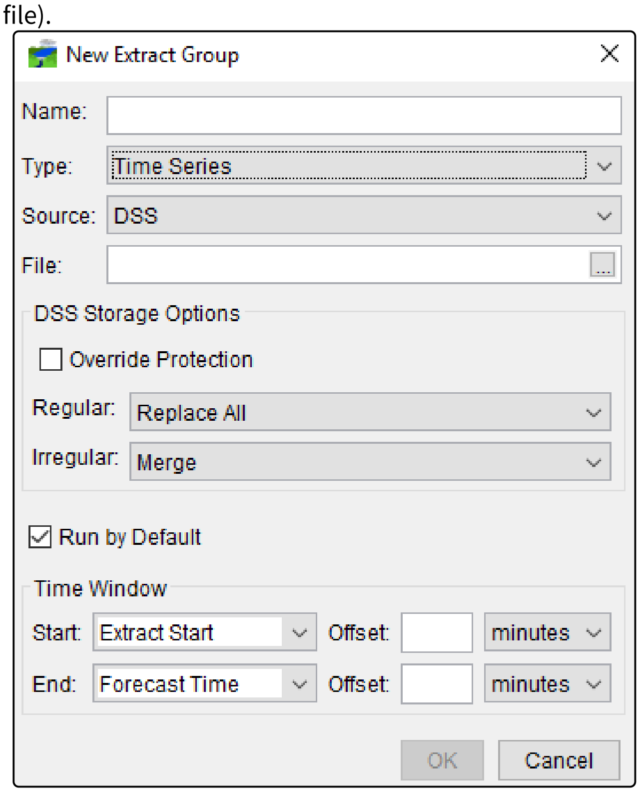


113 Figure 1 Extract Groups Box

22.3.1 Create an Extract Group

To create an extract group:

1. From the HEC-RTS main window, click on the **Setup** tab, from the **Models** menu, click **Edit Extract**, the **Extract Editor** will open.
2. From the **Extract Linking** section, from the **Extract Groups** box (Figure 1), click **New Extract Group**, the **New Extract Group** dialog will open (Figure 2).
3. In the **Name** box (Figure 2), enter a name for your extract group.
4. From the **Type** list (Figure 2) select the type of data (i.e., Time Series, Gridded Data) being extracted and mapped.
5. If you selected **Time Series** as your **Type**, from the **Source** list select either **DSS** if using a DSS file, or **KIWIS** if selecting data from a WISKI database. If you selected **Gridded Data** for the **Type**, **DSS** is the only option (DSS file).



The KiWIS option is detailed in Appendix K and will not be discussed in this chapter.

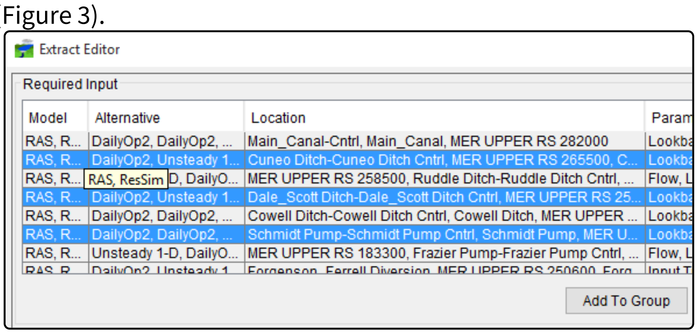
1. For **Time Series** or **Gridded Data** types, from the **File** box (Figure 2), click . An **Open** browser will open, browse to the location of the appropriate file, click on the filename, click **Open**. The **Open** browser will close, and you will be returned to the **New Extract Group** dialog (Figure 2).
2. If you want your extract group to run automatically when a forecast is made, select **Run by Default** (Figure 2).
3. Specify the **Time Window** to extract the data from the **Start** and **End** lists; available options are: **Extract Start, Start of Simulation, Forecast Time, End of Simulation**. Offsets for the start and end times can also be included, available options are **hours**, **minutes**, or **seconds**. In the **Offset** box, you will enter the offset increment. This time window is relative to the forecast time window that you defined when you created the forecast.
4. When you are done, click **OK,** the **New Extract Group** dialog will close (Figure 2). The new extract group will display at the bottom of the **Extract Groups** box, and the information about the extract group will display.

Note: Extract groups are run in the order they are displayed in the ***Extract Groups*** *box. If you run more than one extract group during the extract process and there are records that are mapped in more than one of the selected extract groups, the last extract group mapping will replace all previous mappings.*

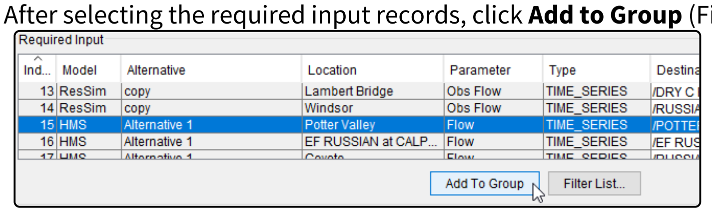
22.3.2 Adding Required Input Data to Extract Groups

Once an extract group is created, you can select what required input data you want to map to the selected source data. To include required input data with an extract group:

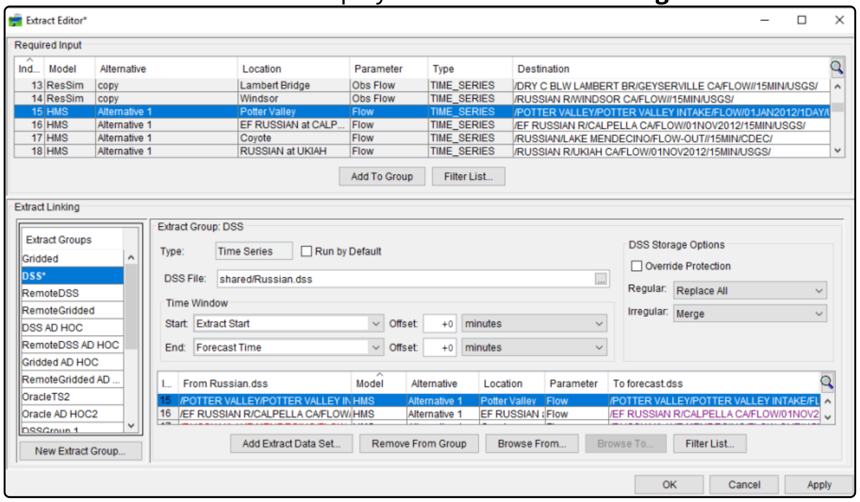
1. Select an extract group from the **Extract Groups** box (Figure 1).
2. In the **Required Input** section, select all the records you want to add to the extract group. You can select multiple records by selecting a record, then pressing the **CTRL** key, click any additional records you want to add (Figure 3).



3. After selecting the required input records, click **Add to Group** (Figure 4).



4. The records selected will be displayed in the **Extract Linking** section in the table (Figure 5).



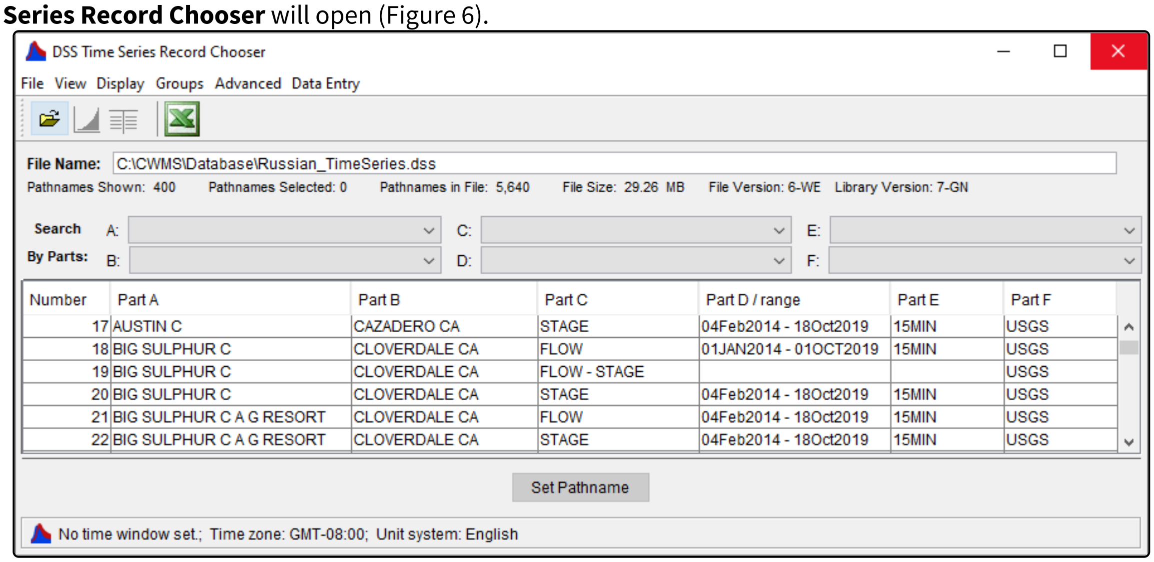
5. Select how much information you want to view in the **Extract Linking** table by clicking the magnifying glass icon ( ). Refer to Extract Editor for further details on formatting the **Extract Linking** table. *Note: If you have the* ***Hide Assigned Destinations*** *selected, the selected records will not display in the* ***Required Input*** *list.*

22.3.3 Selecting Source Records

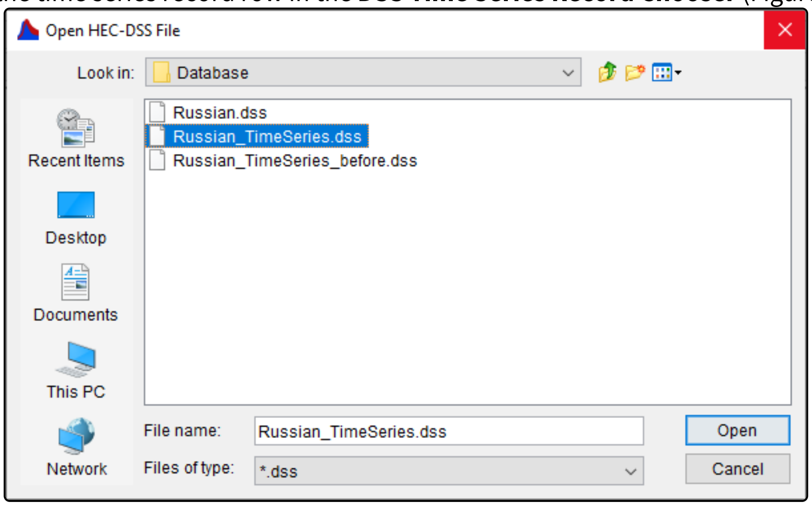
When records are added to an extract group, the records display the name of the record under the **From <*data_source*>** column in the **Extract Linking** table. HEC-RTS does its best to match the **From <*data_source*>** side record names to the **To forecast.dss** side record names, but sometimes HEC-RTS needs some help.

To select records for the **From <*data_source*>** record:

1. Double-click the row in the **From <*data_source*>** column you would like to set the time series record for.
2. Click , which appears in the right section of the cell with the **From <*data_source*>** name. The **DSS Time Series Record Chooser** will open (Figure 6).



3. From the **File** menu, click **Open**, an **Open HEC-DSS File** browser will open (Figure 7). Browse to the location of the DSS file that contains the records needed, click on filename, click **Open**. The **Open HEC-DSS File** browser will close, and the **DSS Time Series Record Chooser** will display its time series records.
4. From the **Search By Parts** area (Figure 6), use the various filter lists to locate the desired time series record.
5. To select a time series record to replace the one displayed in the selected row in the **Extract Linking** table, click the time series record row in the **DSS Time Series Record Chooser** (Figure 6), click **Set Pathnames**.



6. The time series record you selected will display back on the **Extract Linking** table in the row you original selected.
7. To set another time series record, click on the next row in the **Extract Linking** table where you would like to replace the record. From the **DSS Time Series Record Chooser** (Figure 6), click on the time series record you want, click **Set Pathname**.
8. When all the time series records have been set in the **Extract Linking** table, from the **DSS Time Series Record Chooser**, from the **File** menu, click **Close**.

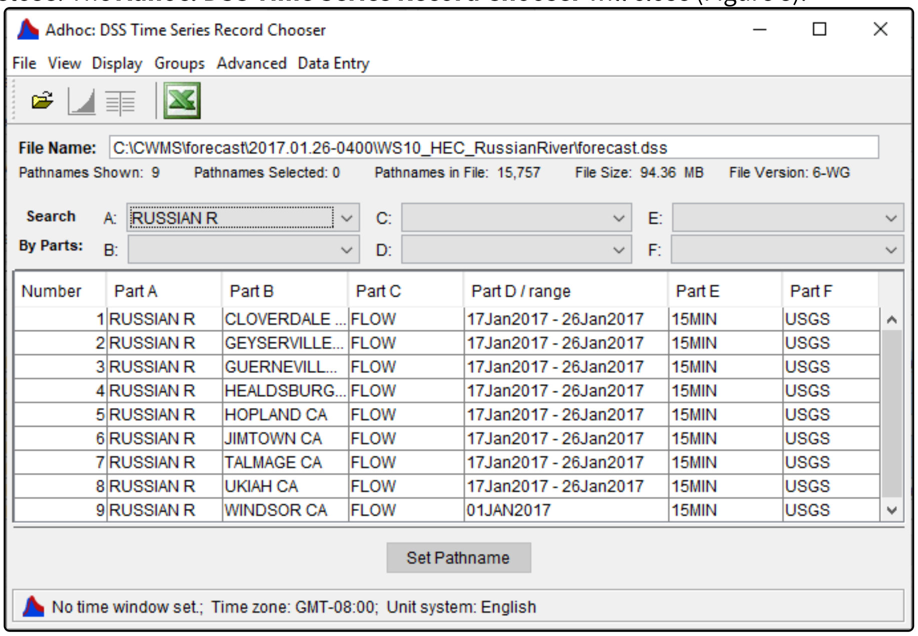
22.3.4 Ad-Hoc Records

Additional input data not listed in the **Required Extract** table may be required for a forecast compute. You can add additional time series records using the **Ad-Hoc Record** feature. An Ad-Hoc record allows you to add time series records to a DSS time series record in the *forecast.dss* file.

To add an Ad-Hoc record:

1. From the **Extract Editor**, click **Add Extract Data Set**, the **Adhoc: DSS Time Series Record Chooser** will open (Figure 8).
2. From the **Search By Parts** area (Figure 8), use the various filter lists to locate the desired time series record.
3. Select the pathname row for the record, click **Set Pathname**.

4. Once you have selected all the pathnames for all the necessary time series records, from the **File** menu, click **Close**. The **Adhoc: DSS Time Series Record Chooser** will close (Figure 8).



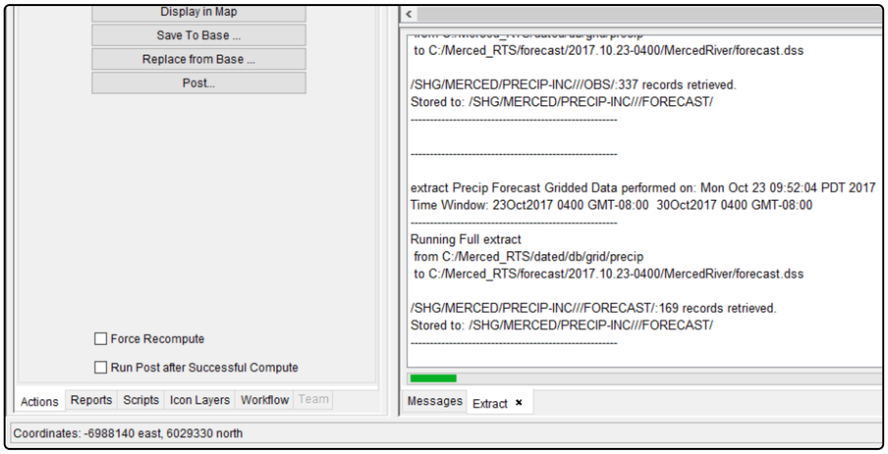
22.4 Running Extracts

Extracts can be run two ways, automatically when creating a forecast or when a forecast is open in the **Modeling** module.

22.4.1 Automatic Extracts

When a forecast is created, all extract groups that were set to **Run as Default** will automatically run. **Run as Default** is selected when you create an extract group; from the **Extract Groups** box, from the shortcut menu, you can toggle **Run as Default** on and off; and, from the **Extract Linking** section, you can also toggle **Run as Default** on and off.

During the creation of a forecast, as the extract process starts, an **Extract** tab (Figure 1) will display in the **Message Pane** of the HEC-RTS main window.

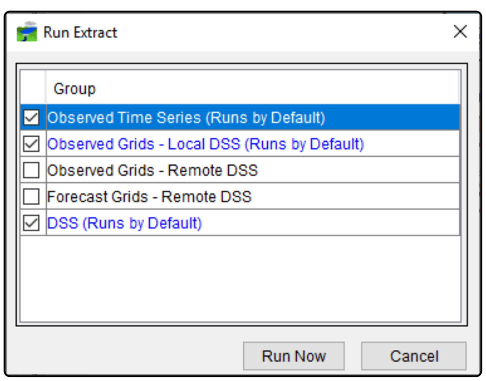


114 Figure 1 Extract Tab

The **Extract** tab displays messages on the extract process. These messages include how many values were retrieved for each time series, where they were retrieved from, and the pathname that the time series will be stored under in the *forecast.ds*s file.

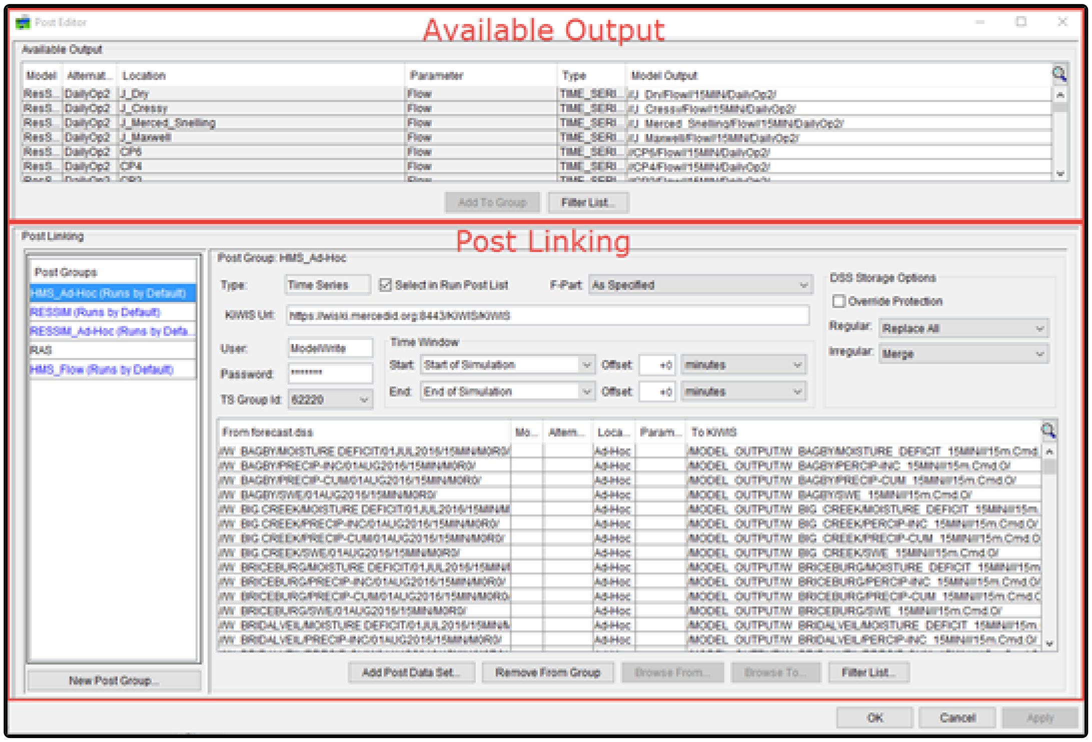
22.4.2 Manual Extracts

Extract groups can be run manually for a forecast. From the **Modeling** module, you must have a forecast open. From the **Forecast** menu, click **Extract**, the **Run Extract** dialog will open (Figure 2). From the **Group** list, select the extract groups you would like to manually run. Click **Run Now** (Figure 2), the **Run Extrac**t dialog will close, and the extract process will begin. The **Extract** tab (Figure 1) will display messages as the extract process proceeds.



22.5 Post

A **post** refers to the process of saving data from the *forecast.dss* file to a DSS file or a WISKI database (WISKI Extract and Post). A post selects an output time series from the *forecast.dss* file and disseminates that information to an external data location. Compute output can be stored as historical results to be viewed or used in future forecasts. HEC-RTS supports posting for DSS gridded data and time series data, and WISKI (WISKI Extract and Post) time series data. From the Setup module, you will use the **Post Editor** (Figure 1) to define post groups.



115 Figure 1 Post Editor

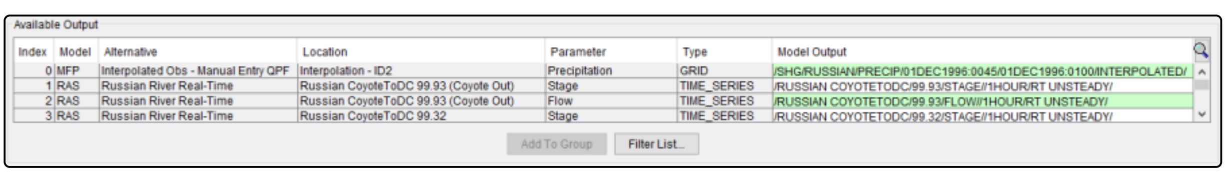
22.6 Post Editor

Now that you have successfully computed a forecast and reviewed results, you might want to save data from the *forecast.dss* file to other DSS files. A post is created and defined from the **Post Editor**. From the **Setup** module, from the **Models** menu, click **Edit Post**, the **Post Editor** will open.

The **Post Editor** is comprised of two sections: **Available Output** and **Post Linking**. The **Available Output** section contains a table that lists the output records from each model alternative. The **Post Linking** section contains information on the defined post groups, and contains a table, which lists the output records and sources records assigned to a post group.

22.6.1 Available Output Section

The **Available Output** section (Figure 1) provides a list of the model alternative output time series records that can be mapped to the CWMS database or other DSS files.



116 Figure 1 Post Editor - Available Output Section

The **Available Output Table** provides the following information:

Model this column lists which software application created the output record.

Alternative this column lists the model alternative name that created the output record.

Location this column displays the name of the model object that is associated with the output record.

Parameter this column displays the parameter associated with the output record.

Type this column specifies if the record is times series data or gridded data. Types include **GRID** or **TIME_SERIES**. **GRID** represents DSS gridded data. **TIME_SERIES** represents time series data.

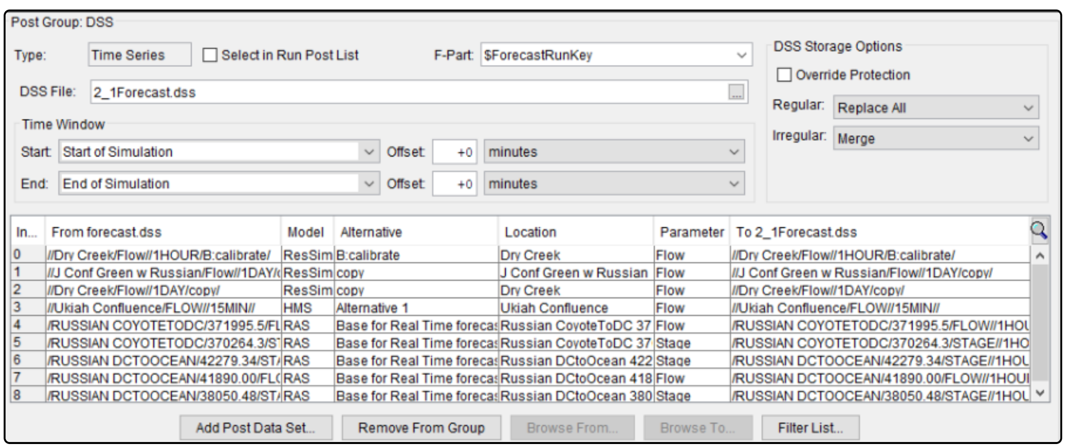
Model Output this column displays the actual output record pathname that a model alternative creates. This is the pathname that will be written to the *forecast.dss* file during the compute.

Index Each time series pathname is assigned an **Index** number when it is loaded into the **Post Editor**. This number has no real usage except keeping the initial filtered list set. There can be multiple indices for a single DSS pathname.

This information does not display by default in the **Available Output** table, you will need to set up the table view.

22.6.2 Post Linking Section

The **Post Linking** section (Figure 2) displays information on the post groups. Post groups link similar time series output data to a single data destination. The destination can be a DSS file or a WISKI database (WISKI Extract and Post). The destination records will be mapped to the required output records in a post group. During a forecast compute, the destination records will be copied into the required output pathnames when are created in the *forecast.dss* file.



117 Figure 2 Post Editor - Post Linking Section

The **Post Linking** section provides information on the following:

From forecast.dss this column lists the selected model alternative's output records for a post group. This data will fill the DSS record displayed in the **To *<data_source>*** column.

Model this column lists which software application generates the output data.

Alternative this column lists the model alternative that is created the output record.

Location this column displays the name of the model location object that is associated with the output record.

Parameter this column displays the parameter associated with the output record's data.

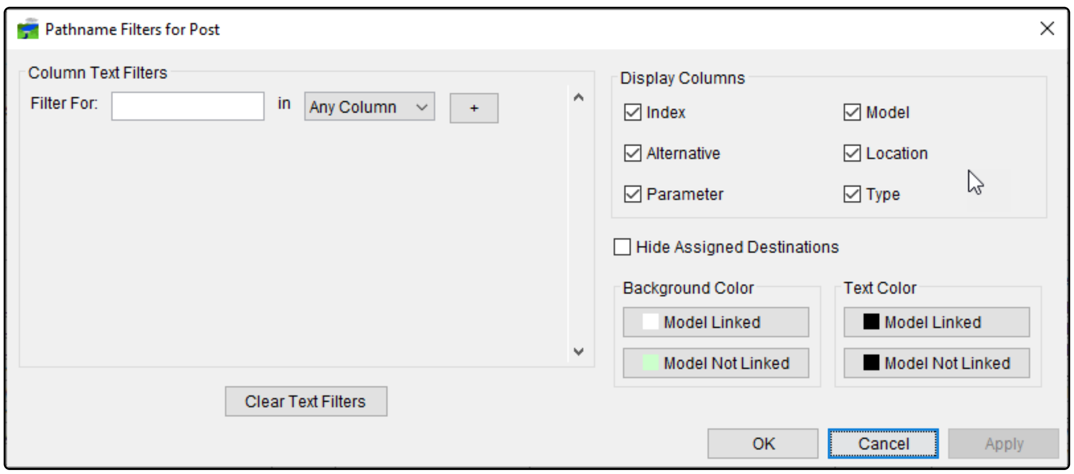
To <*data_source*> this column displays the output record pathname that will be written to the **To <*data_source*>** during the post process.

Index Each required destination pathname is assigned an **Index** number when it is loaded into the **Post Editor**. This number has no real usage except keeping the initial filtered list set. There can be multiple indices for a single DSS pathname.

This information does not display by default in the **Post Linking** table, you will need to set up the table view.

22.6.3 Table View Options

You can change the amount of information displayed in the tables in both the **Available Output** and the **Post Linking** sections. Clicking on the magnifying glass icon ( ) at the top right-hand corner of the tables in each section displays the **Pathname Filters for Post** dialog (Figure 3).



118 Figure 3 Pathname Filters for Post Editor - Available Output Table

From the **Column Text Filters** box (Figure 3) you can filter the records to display only records that contain a certain phrase or string of text. Filtering can be done by **Column**, **Index**, **Model**, **Alternative**, **Location**, **Parameter**, **Type**, or **Model Output**. You can filter for multiple requirements by clicking **+**. Below are examples of several different ways that you can filter:

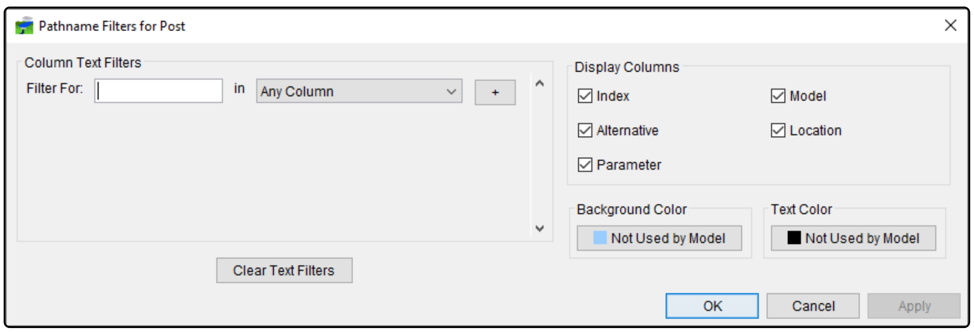
- Filters can be used to retrieve only records that are an exact match. You can do this by typing a string of text in the **Filter For:** box.
- Filters can be used to retrieve records that start, end, or contain a specific string. To do this, use an asterisk (*) after, before, or inserted into the string of text you want to filter for.

From the **Display Columns** box (Figure 3) you can select which columns in the table to view. For the examples in this chapter, **Index**, **Model**, **Alternative**, **Location**, **Parameters**, and **Type** have been selected for the tables in each section of the **Post Editor**.

You can remove a record from a table that has already been assigned to an extract group, select **Hide Assigned Destinations** (Figure 3). The **Hide Assigned Destinations** option is not available for the **Post Linking** section. For the examples in this chapter, **Hide Assigned Destinations** has not been selected.

You can also color code the output records listed in the **Available Output** section (Figure 3) depending on if the output record is linked to a destination record. You can color code by background or text. For example, to add a background color to a record that is not linked, from the **Background Color** box (Figure 3), click **Model Not Linked**, a **Color Chooser** dialog will open (Using the Color Chooser). Select a color, click **OK**, the **Color Chooser** will close, and from the **Model Output** column of the **Available Output** table, any record that has not been linked will be highlighted in the selected color (Figure 3).

You can also color code the records listed in the **Post Linking** section (Figure 4) when an output record is not used by a model. You can color code by background or text. For example, in Figure 4, to add a background color to a record that is not used by a model, from the **Background Color** box (Figure 4), click **Not Used by Model**, a **Color Chooser** dialog will open (Using the Color Chooser). Select a color, click **OK**, the **Color Chooser** will close, and from the **Post Linking** table, any record that has not been used by a model will be highlighted in the selected color.

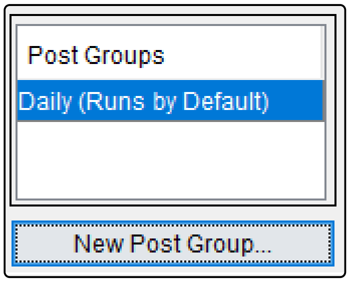


22.7 Post Groups

Post groups map the model alternative output time series data from the *forecast.dss* file to a destination data source. The following destinations are supported:

- DSS file
- Remote DSS file located on a CWMS server
- CWMS Database
- WISKI database (KiWIS Write Extension)

From the **Post Editor**, from the **Post Linking** section (Pathname Filters for Post Editor – Post Linking Table), the **Post Groups** box (Figure 1), lists the defined post groups for a watershed. From the **Post Groups** box, you can create a post group; delete a post group; rename a post group; make a copy of a post group; select which post groups will run by default during the post process; and, determine the order in which post groups will run in a post process. In addition, you can color code the post groups that will run by default.

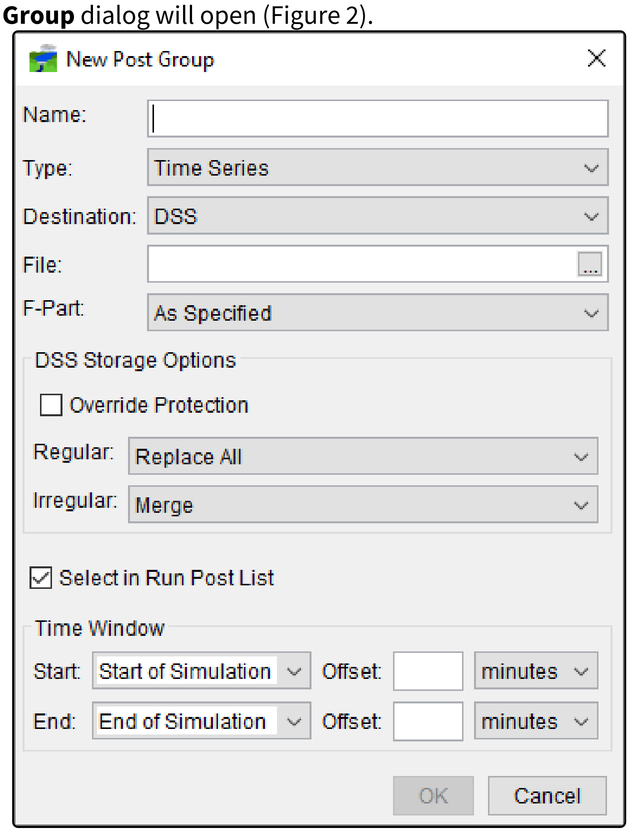


119 Figure 1 Post Groups Box

22.7.1 Create a Post Group

To create a post group:

1. From the HEC-RTS main window, click on the **Setup** tab, from the **Models** menu, click **Edit Post**, the **Post Editor** will open.
2. From the **Post Linking** section, from the **Post Groups** box (Figure 1), click **New Post Group**, the **New Post Group** dialog will open (Figure 2).



3. In the **Name** box (Figure 2), enter a name for your post group.
4. For posting data there is only one data type – **Time Series**.
5. From the **Destination** list (Figure 2) you can select **DSS** if using a DSS file or KIWIS when using a WISKI database.

The KiWIS option is detailed in WISKI Extract and Post and will not be discussed in this chapter.

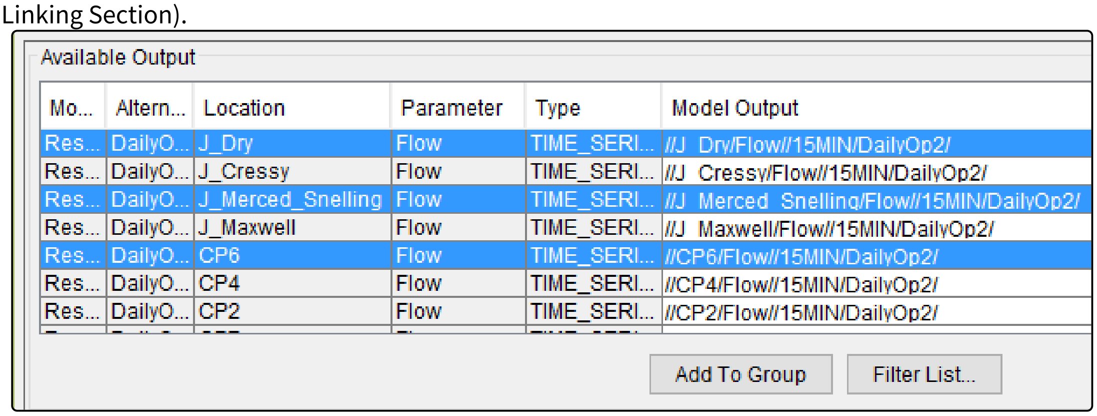
1. From the **File** box (Figure 2), click . An **Open** browser will open, browse to the location of the appropriate file, click on the filename, click **Open**. The **Open** browser will close, and you will be returned to the **New Post Group** dialog (Figure 2).
2. Select the F-Part substitution for the output records. From the **F-Part** list (Figure 2) you have several options to select from, the default is **As Specified**.
3. If you want your post group to run automatically during a post process, select **Select in Run Post List** (Figure 2).
4. Specify the **Time Window** to post the data from the **Start** and **End** lists; available options are: **Extract Start, Start of Simulation, Forecast Time, End of Simulation**. Offsets for the start and end times can also be included, available options are **hours**, **minutes**, or **seconds**. In the **Offset** box, you will enter the offset increment. This time window is relative to the forecast time window the you defined when you created the forecast.
5. When you are done, click **OK,** the **New Post Group** dialog will close (Figure 2). The new post group will display at the bottom of the **Post Groups** box, and the information about the post group will display.

*Note: Post groups are run in the order they are displayed in the **Post Groups** box. If you run more than one post group during the post process and there are records that are mapped in more than one of the selected post groups, the last post group mapping will replace all previous mappings.*

22.7.2 Adding Output Data to Post Groups

Once a post group is created, you can select which model alternative output records you want to include in in the post group. To add model alternative output records to a post group:

1. From the **Post Groups** box (Figure 1), select the name of a post group, the **Post Linking** table (Post Editor - Post Linking Section) will display information about the selected post group.
2. In the **Available Output** section (Figure 3), select all the output records that you want to add to the post group from the table. You can select multiple records by clicking on one record row in the table, then pressing your **CTRL** key and clicking on any additional records you want to add (Figure 3).
3. Click **Add to Group** (Figure 3), the records selected will display in the **Post Linking** table (Post Editor - Post Linking Section).



You can select how much information you want to view in the **Post Linking** table by clicking . Refer to Post Editor for further details on formatting the **Post Linking** table. *Note: If you have the **Hide Assigned Destinations** selected, the selected records will not display in the **Available Output** list.*

22.7.3 Selecting Destination Records

When **Available Output** records are added to a post group, the name of the **Available Output** record is displayed under the **To <*data_source*>** column in the **Post Linking** table (Post Editor - Post Linking Section). These **To <*data_source*>** record pathnames need to be replaced with the destination time series record name.

To select a time series record for the **To <*data_source*>** record:

1. From the **Post Linking** table, double-click the row in the **To <*data_source*>** column you would like to set the time series record for.
2. Click  , which appears in the right section of the cell with the **From <*data_source*>** name. The **DSS Time Series Record Chooser** will open.
3. From the **File** menu, click **Open**, an **Open HEC-DSS File** browser will open. Browse to the location of the DSS file that contains the records needed, click on filename, click **Open**. The **Open HEC-DSS File** browser will close, and the **DSS Time Series Record Chooser** will display its time series records.
4. You can filter the time series pathnames displayed in the table by using the **Search By Parts Parameters** at the top of the **DSS Time Series Record Chooser**.
5. To select a time series record to replace the one displayed in the selected row in the **Post Linking** table, select the time series record row in the **DSS Time Series Record Chooser**, click **Set Pathnames**.
6. The time series record you selected will display in the Post Linking table in the row you original selected.

To set another time series record, from the **DSS Time Series Record Chooser**:

1. Click on the next row in **Post Linking** table that you would like to replace, click on the time series record you want to use in the **DSS Time Series Record Chooser**, click the **Set Pathname**.
2. When all of the time series records have been set in the **Post Linking** table, from the **DSS Time Series Record Chooser**, from the **File** menu, click **Close**.

22.7.4 Ad-Hoc Records

From the **Available Output** table, you can add additional time series records to the outgoing data source using the Ad-Hoc record feature in a post group. An Ad-Hoc record allows you to save *forecast.dss* time series record data to a DSS file even if those records were not listed in the **Available Output** table.

To add an Ad-Hoc record:

1. From the **Extract Editor**, click **Add Extract Data Set**, the **Adhoc: DSS Time Series Record Chooser** will open.
2. From the **Search By Parts** area (**Adhoc: DSS Time Series Record Chooser**), use the various filter lists to locate the desired time series record.
3. Select the pathname row for the record, click **Set Pathname**.
4. Once you have selected all the pathnames for all the necessary time series records, from the **File** menu, click **Close**. The **Adhoc: DSS Time Series Record Chooser** will close.

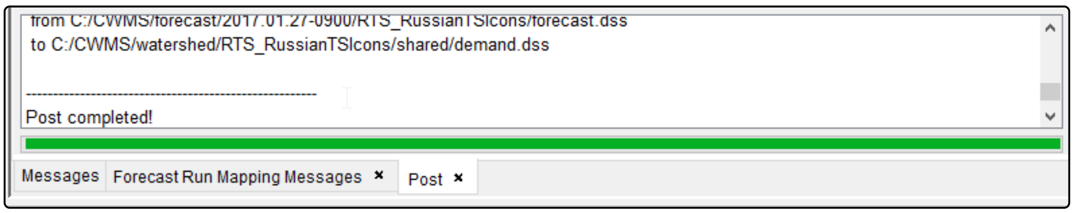
22.8 Running Posts

There are two ways to run a post group. A post group can be set to run automatically after a successful forecast compute or when a forecast is open in the Modeling module.

22.8.1 Automatic Posts

After a forecast is created and before the forecast is computed, you can automatically have the post process run after the forecast compute is completed. From the **Modeling** tab (make sure a forecast is open), from the **Forecast** tabs, click the **Actions** tab. Select **Run Post after Successful Compute**, now your post process will happen automatically once the forecast compute has finished.

Once a post process starts, from the **Message Pane**, a **Post** tab (Figure 1) will display. These messages include how many values were retrieved for each time series, where the time series were retrieved from, and the pathname that the time series will be stored under in the post file.

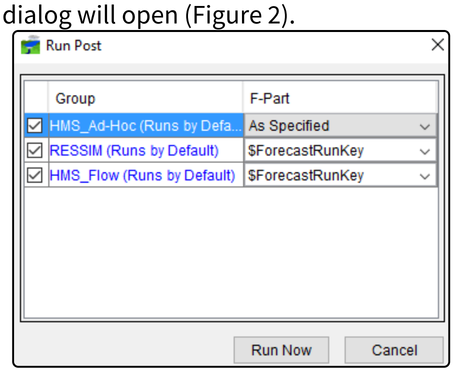


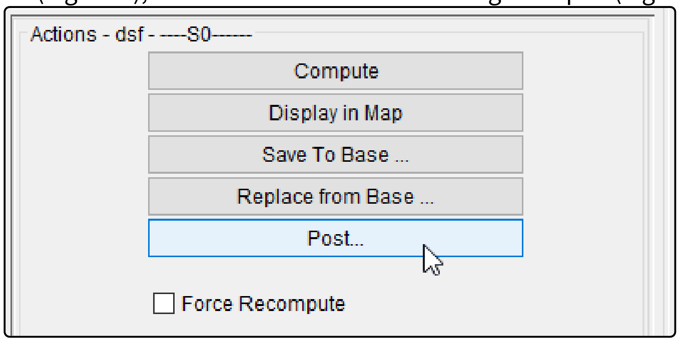
120 Figure 1 Post Message Tab

22.8.2 Manual Posts

To run a post group manually:

1. From the **Modeling** module (be sure that a forecast is open), from the **Models** menu, click **Post**, the **Run Post** dialog will open (Figure 2).



2. Alternatively, from the **Modeling** tab (make sure a forecast is open), from the **Forecast** tabs, click the **Actions** tab (Figure 3), click **Post**. The **Run Post** dialog will open (Figure 2).


3. Select the post groups you would like to run, click **Run Now** (Figure 2).
4. The **Run Post** dialog will close, and the post process will start. Once a post process starts, from the **Message Pane**, a **Post** tab (Figure 1) will display. These messages include how many values were retrieved for each time series, where the time series were retrieved from, and the pathname that the time series will be stored under in the post file.

23 Team Modeling

Team modeling facilitates collaboration between modelers working with the same watershed and forecasts. Multiple modelers may be sharing responsibility within a complex watershed containing several models. One modeler may only be responsible for the HEC-HMS model, while another oversees HEC-ResSim and HEC-RAS. Team modeling allows each modeling team member to contribute expertise and participate in real-time synchronized activities.

Team modeling consists of a master watershed and its associated forecasts, which are stored in a shared location (such as a networked drive). Team members share data by uploading and downloading changes to and from a master watershed, interacting with a local copy that resides on their computer. When watershed files are updated on the master watershed, team members working on the same watershed are notified of the changes by a real-time notification system.

The master watershed keeps detailed records that provide information on what files are changed, when the changes were made, and who modified the data. This information includes changes made to base and forecast data. **Base data** refers to the starting copies of the model alternatives that are used in forecasts. **Forecast data** represents the working copies of the model alternatives, including observed and forecast boundary condition data.

This chapter describes how to utilize the team modeling features, how to set up a master watershed; how to work with base data; and, how forecasts work in a team environment. Further additional details important to understanding team modeling, are detailed in Technical Notes on Versioning.

23.1 Master Watersheds

A **master watershed** is a watershed shared by multiple modelers. The master watershed consists of the watershed's base and forecast data. The master watershed needs to reside somewhere accessible by all participating team members. If team members intend to contribute updates to the data or models, then that team member must have write privileges on the shared drive (shared network drive).

The master watershed is not modified directly on the shared location. Instead, a copy of the master watershed is downloaded to each team member's computer. The team members edit their local copy of the master watershed, then upload changes back to the master watershed. Only files and DSS records that have been changed are uploaded and downloaded, this speeds up the process significantly since files that have not been changed do not need to be transferred.

A master watershed keeps track of the changes that each team member makes and notifies other team members of those changes. If the master watershed is modified by a member while others are working on the master watershed, the other members will receive a notification on the lower right-land side of their desktop alerting them that the files on their local copy of the master watershed are now out-of-date. If the watershed is closed at the time of the change, an out-of-sync notification will appear when the modeler next opens the watershed.

Any HEC-RTS (Version 3.1 or higher) study can become a master watershed. The master watershed may begin as a watershed on your computer or already exist on a networked drive. Once the master watershed has been set up, pre-existing forecasts may be turned into team forecasts. This is further outlined in Team Forecasting.

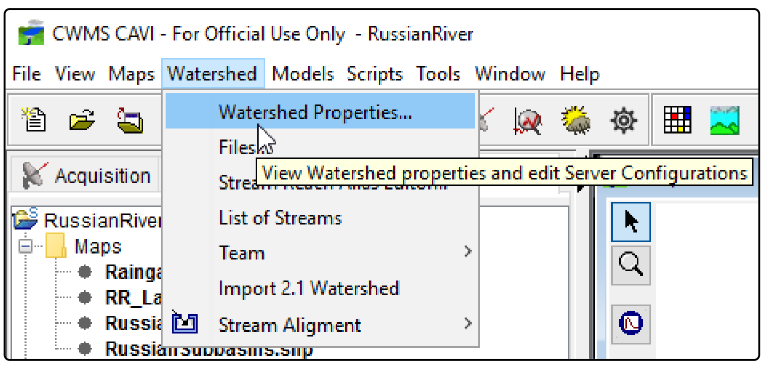
Regular watersheds will not have access to the team modeling features that master watersheds provide until they are upgraded to a master watershed. The following topics provide detailed instructions on how to set up a master watershed and walk you through the different setup scenarios.

23.1.1 Create a Master Watershed

Any HEC-RTS (Version 3.1 or higher) study can become a master watershed. You can turn a regular watershed on your computer into a master watershed and then upload it to a networked drive. After the master watershed is uploaded, other modelers will have access to it. You can also turn a regular watershed that already exists on a networked drive into a master watershed. The following sections go through each of these processes in detail.

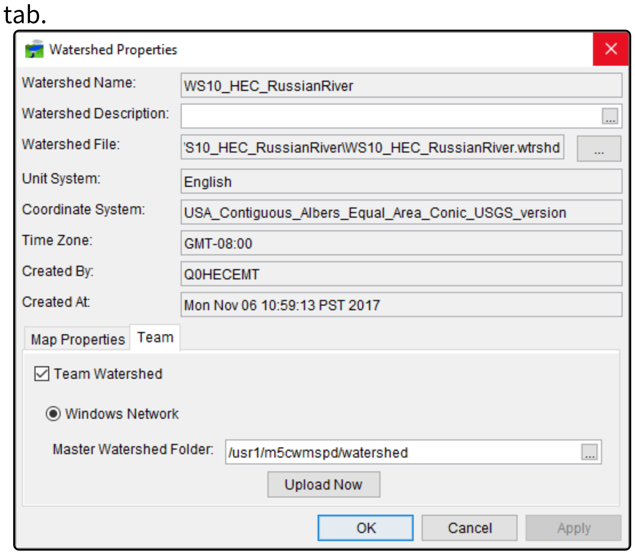
23.1.1.1 Create Master Watershed from Watershed on Your Computer

A regular HEC-RTS (Version 3.1 or higher) watershed located on your computer can be turned into a master watershed and then uploaded to a network drive. To turn a watershed into a master watershed, open the watershed you would like to convert to a master watershed in HEC-RTS (Version 3.1.1), select the **Setup** tab. From the **Watershed** menu, click **Watershed Properties** (Figure 1)



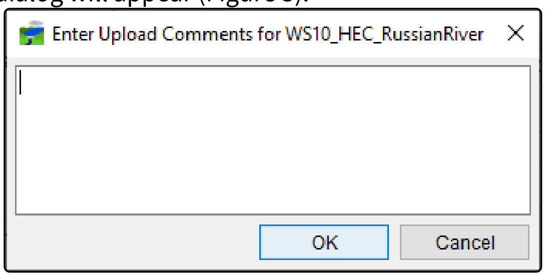
121 Figure 1 Setup Module - Watershed Menu

1. The **Watershed Properties** dialog will open (Figure 2). From the **Watershed Properties** dialog, select the **Team** tab.

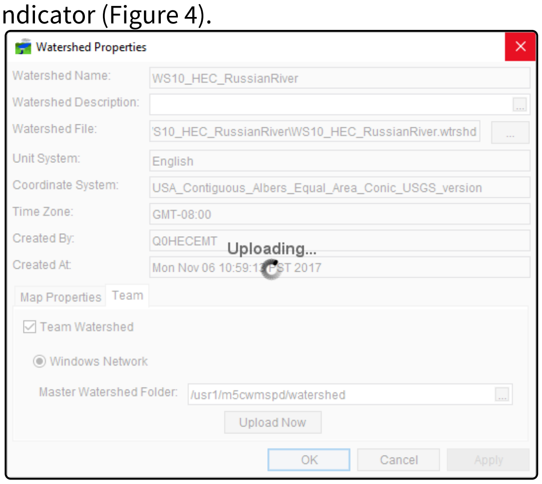


2. In the **Team** tab, select **Team Watershed** (Figure 2).
3. To upload the master watershed to a network location, click **Windows Network**.

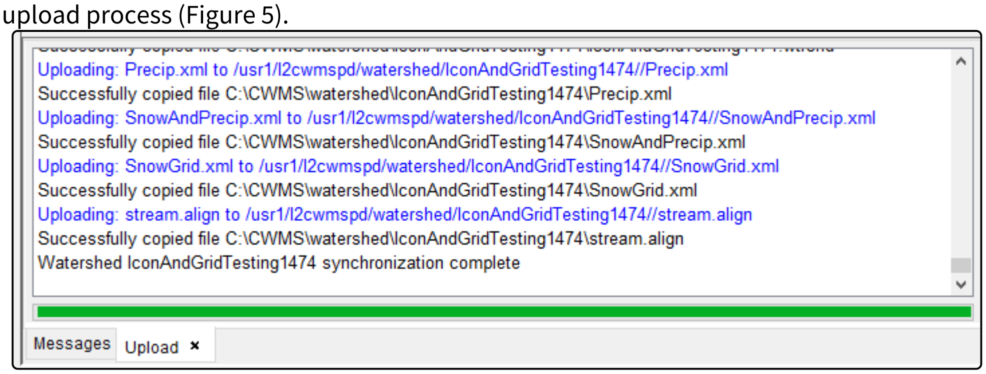
4. When uploading to a local network location, the file path will need to be selected. To the right of the **Watershed File** box, click ..., an **Open** browser will open. Browse to the folder location on the shared network where you would like to upload the master watershed.
5. Click **Open**, the Open file browser will close, and the selected file pathname will display in **Master Watershed Folder** box (Figure 2).
6. Once the upload location has been selected and defined, click **Upload Now**. The **Enter Upload Comments** dialog will appear (Figure 3).



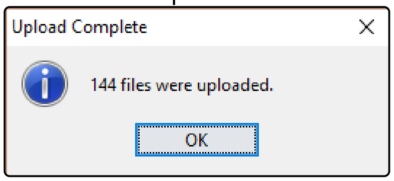
7. In the **Enter Upload Comments** dialog, type in a description for the upload and then click **OK**. The **Enter Upload Comments** dialog will close, and the upload will start. **Note:** The comments are part of the versioning for team modeling and cannot be changed later.
8. The **Watershed Properties** dialog will become inactive during the upload process and display an **Uploading** indicator (Figure 4).



9. An **Upload** message tab will appear along the bottom of the CAVI screen providing status updates during the upload process (Figure 5).



10. When the upload has completed, an **Upload Complete** message will appear (Figure 6), indicating the number of files that were uploaded for the master watershed.

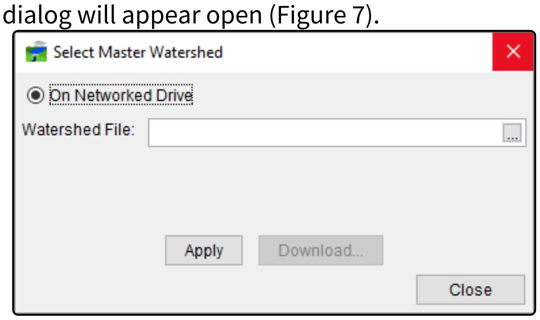


Once the watershed is uploaded to the network drive, other team members can access it and download copies of the files to their own computers. For more instructions on this process see

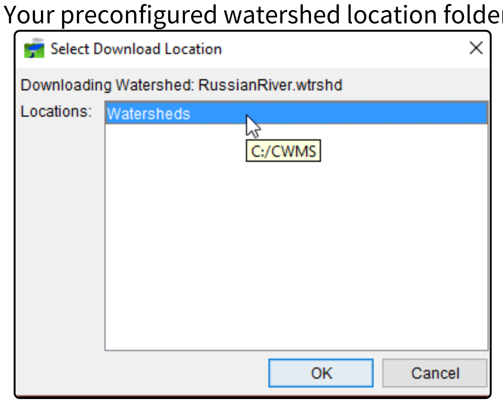
23.1.1.2 Create a Master Watershed from a Watershed on a Networked Drive

Watersheds that already exist on a networked drive can be turned into master watersheds and shared with the team. The following steps, show how you would convert a pre-existing HEC-RTS study into a master watershed:

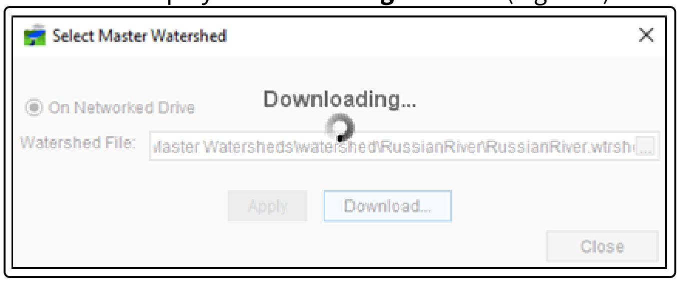
1. Start HEC-RTS Version 3.1.1, but do **not** open a watershed. From the HEC-RTS main window, click the **Setup** tab.
2. From the **Watershed** menu, point to **Team**, click **Download Master Watershed**. The **Select Master Watershed** dialog will appear open (Figure 7).



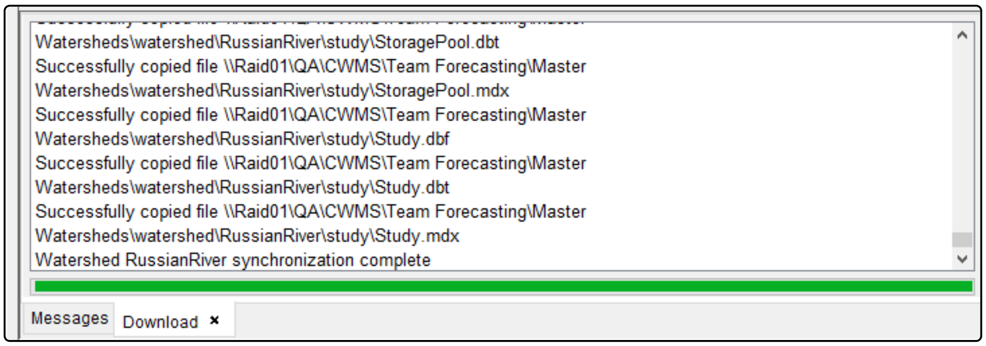
3. Select **On Networked Drive** (Figure 7), to the right of the **Watershed File** box, click . An **Open** browser will open.
4. From the **Open** browser, browse to the location of the watershed, select the watershed you want to make a master watershed, click **Open**. The **Open** browser will close, the location and name of the selected watershed will display in the **Watershed File** box (Figure 7).
5. From the **Select Master Watershed** dialog (Figure 7), click **Apply**. A confirmation window will display confirming the watershed is now a master watershed. Now you can download the master watershed to your computer.
6. Close the confirmation window, click **Download**, the **Select Destination Location** dialog will open (Figure 8). Your preconfigured watershed location folder is displayed in the **Locations** list (Figure 8).



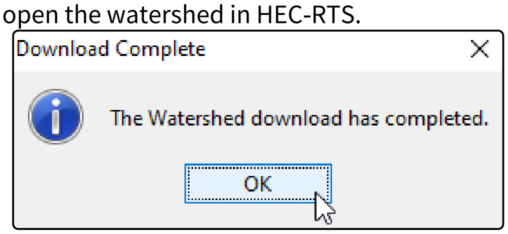
7. Select the appropriate watershed location folder from the **Locations** list, click **OK**, the download of the master watershed will begin. The **Select Master Watershed** dialog (Figure 7) will become inactive during the download process and display a **Downloading** indicator (Figure 9).

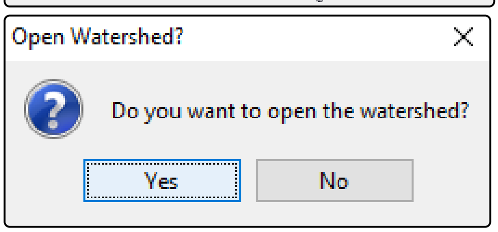


8. A **Download** tab will display in the **Message Pane** (Figure 10), providing status updates during the download process.



9. When the download is finished a **Download Complete** window (Figure 11) will display. Click **OK**, the **Download Complete** window will close. An **Open Watershed** window will display (Figure 12) asking if you would like to open the watershed in HEC-RTS.



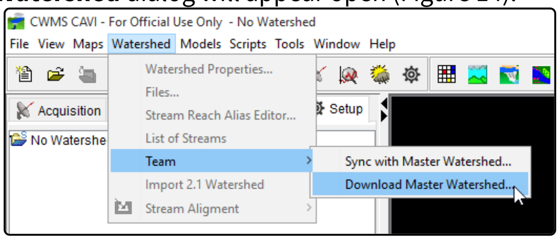


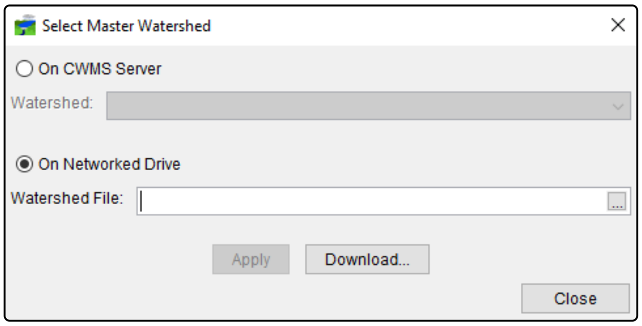
23.1.2 Download an Existing Master Watershed

When you are ready to share the master watershed with other team members, they will need to download a copy of the master watershed to their computers.

To download an existing master watershed:

1. Start HEC-RTS Version 3.1.1, but do **not** open a watershed. From the CWMS CAVI main window, click the **Setup** tab.
2. From the **Watershed** menu, point to **Team**, click **Download Master Watershed** (Figure 13). The **Select Master Watershed** dialog will appear open (Figure 14).





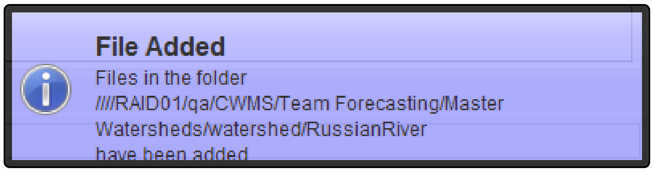
3. To continue downloading the master watershed from a networked drive see the next section – **Download from a Networked Drive**

23.1.2.1 Download from a Networked Drive

The following steps will guide you through downloading the master watershed from a network drive:

1. From the **Select Master Watershed** dialog, click **On Networked Drive** (Figure 14), to the right of the **Watershed File** box, click ... . An **Open** browser will display.
2. From the **Open** browser, browse to the location of the watershed, select the watershed you want to make a master watershed, click **Open**. The **Open** browser will close, the location and name of the selected watershed will display in the **Watershed File** box (Figure 14).
3. From the **Select Master Watershed** dialog (Figure 14), click **Apply**. If the watershed is already a master watershed, a **No Changes Made** window will display. If the watershed is not a master watershed, the watershed will be set as a master watershed and a message window will state that the watershed was successfully updated. Close the message window.
4. From the **Select Master Watershed** dialog (Figure 14), click **Download**. A **Select Download Location** dialog will display (Figure 8). Your preconfigured watershed location folder is displayed in the **Locations** list (Figure 8).
5. Select the appropriate watershed location folder from the **Locations** list, click **OK**, the download of the master watershed will begin. The **Select Master Watershed** dialog (Figure 14) will become inactive during the download process and display a **Downloading** indicator (Figure 9).
6. A **Download** tab will display in the **Message Pane** (Figure 10), providing status updates during the download process.
7. When the download is finished a **Download Complete** window (Figure 11) will display. Click **OK**, the **Download Complete** window will close.

8. The real-time messaging systems will also display a message (Figure 15) indicating that files have been added for the master watershed.



9. When the download is done, An **Open Watershed** window will display (Figure 12), asking if you would like to open the watershed in HEC-RTS. To open the watershed, click **Yes**.

23.1.3 Turning Team Modeling Off

If you no longer need to use team modeling with the local copy of the watershed you can turn the feature off. Turning team modeling off will convert the watershed into a regular watershed and it will no longer communicate with the master watershed. The watershed will behave as a normal HEC-RTS study and all team modeling options will be disabled.

To turn team modeling off for the local copy of the watershed:

1. First, from the HEC-RTS main window, open the master watershed that is on your computer, click the **Setup** tab.
2. From the **Watershed** menu, click **Watershed Properties** (Figure 1), the **Watershed Properties** dialog will open (Figure 2).
3. Click on the **Team** tab (Figure 16), uncheck **Team Watershed**. Click **OK**, the **Watershed Properties** dialog will close.

4. From the HEC-RTS main window, from the **Toolbar**, click  , this will save all changes made to the watershed.

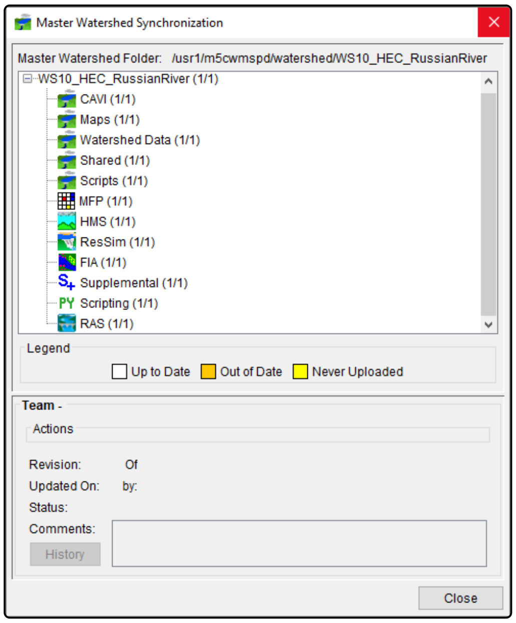
23.2 Master Watershed Base Data

Data that resides in the watershed's main folder is called **base data**. Base data includes all the data that supplies information displayed in the **Setup**, **Acquisition**, and **Visualization** modules. This data encompasses all data needed to create a forecast. Base data includes maps, layers, layouts, reports, scripts, extract post information, all model alternatives, shared and study information, and HEC-RTS data. Base data does not include data for the forecast, more detailed information on forecast data is described in Team Forecasting.

This section walks through how to determine which local base data files are up-to-date, how to download base data from the master watershed, and how to upload revised files back to the master watershed to share with other team members working on the watershed.

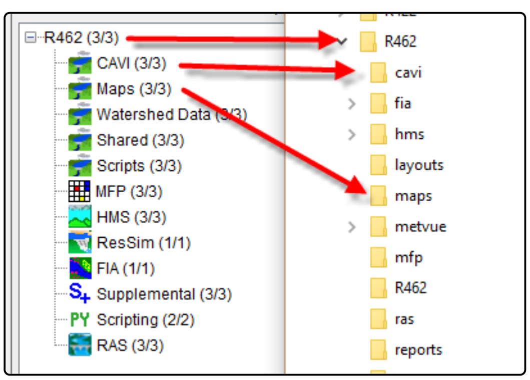
23.2.1 Master Watershed Synchronization Dialog

The **Master Watershed Synchronization** dialog (Figure 1) is used to upload or download base data to and from the master watershed. From the HEC-RTS main window, click on the **Setup** tab, from the **Watershed** menu, point to **Team**, click **Sync with Master Watershed**, the **Master Watershed Synchronization** dialog will open (Figure 1).



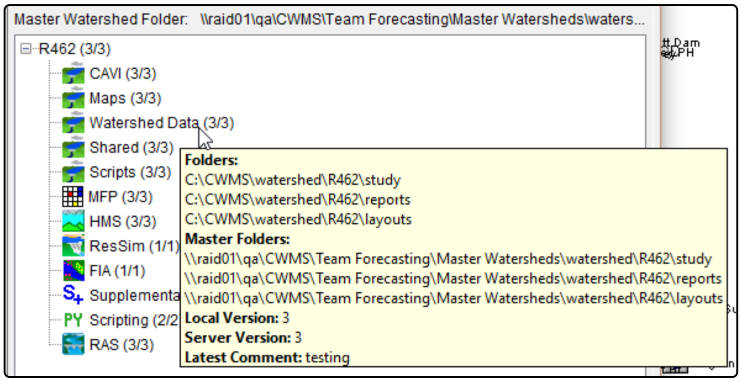
122 Figure 1 Master Watershed Synchronization Dialog

On the **Master Watershed Synchronization** dialog (Figure 1), the base data is displayed in an expandable tree called the **Synchronization Tree**. Each node in the **Synchronization Tree** represents a data group for the watershed. The data groups reflect the folder structure of the current watershed folder (Figure 2).



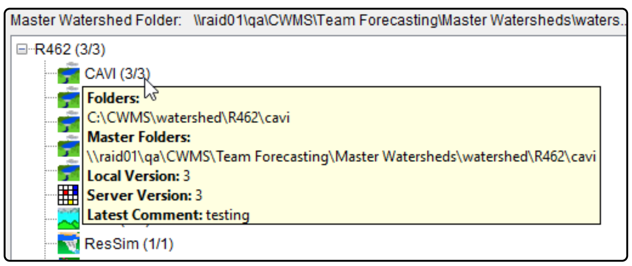
123 Figure 2 Watershed Folders

The **Watershed Data** node (Figure 3), is a special data group. This data group represents three folders in the watershed's folder structure - study, reports, and layouts. Hovering the mouse over a node in the **Synchronization Tree** displays a tooltip (Figure 3), that provides information on what watershed folder(s) represent the data group; what master watershed folder(s) represent the data group; what the version is for the local copy of the watershed; what the version is for the server copy of the watershed; and comments.



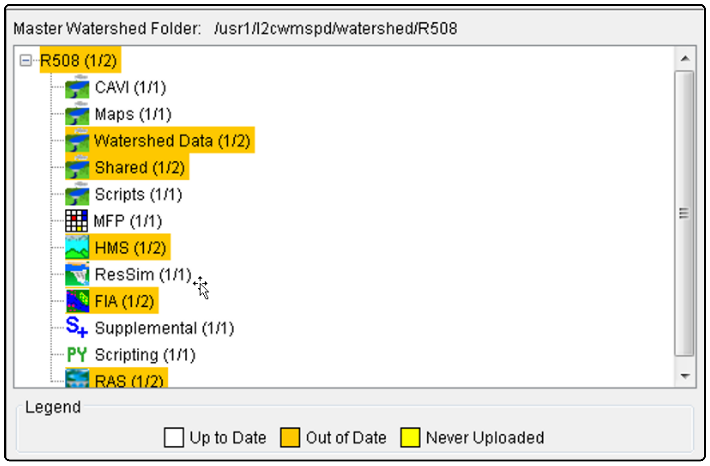
124 Figure 3 Watershed Data Synchronization Tree Node

For example, in Figure 4, the top node is **R462 (3/3)**. This is the name of the watershed. The top node represents all the data groups under the watershed's main folder. If you upload or download this node, **all** watershed base data will be transferred.



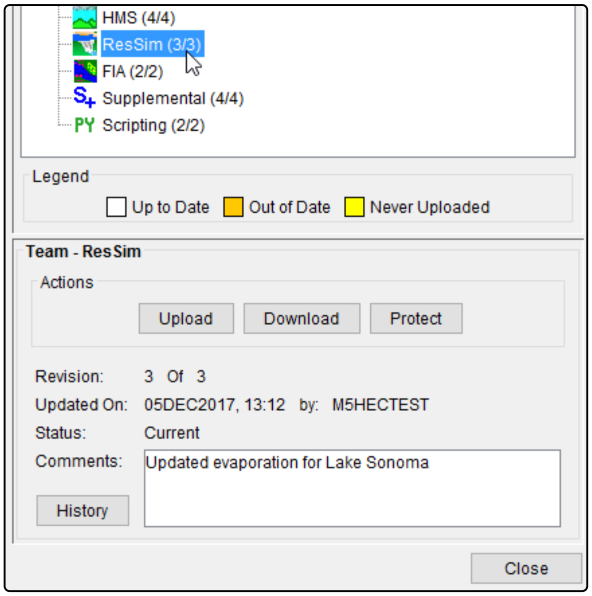
Next to a data group node, there are two numbers shown in parenthesis. For example, in Figure 5, from the **Synchronization Tree**, the **Shared folder** has **(1/2)** displayed. These numbers indicate what version of the folder is on your computer and what version is on the master watershed. The first number (top) represents the watershed that is on your computer. The second number (bottom) represents the version of the master watershed. If the top number is less than the bottom number, the master watershed folder has been updated and the changes need to be downloaded to your local copy. For example, **Shared (1/2)**, informs you that the **Shared** folder on the local copy of the watershed is Version 1, and the master watershed is Version 2. If the numbers were "(0/0)" or "(Na/Na)", this means the folder has never been uploaded to the master watershed.

In addition to the displayed version numbers, the **Synchronization Tree** nodes will highlight in different colors depending on the node's revision state (Figure 5). The different colors and their meaning are listed in the **Legend** area (Figure 5) below the **Synchronization Tree**. The highlight colors represent the following information:



125 Figure 5 Synchronization Tree Node Color Indicators

White	Nodes highlighted in **white** means that the local watershed copy is up to date with the master watershed copy for those data groups. The version number displayed next to the node name will have both numbers matching, for example **CAVI (3/3)**.
Orange	Nodes highlighted in **orange** mean that the master watershed has a newer version of the files in a data group than the local watershed copy. The files will need to be downloaded from the master watershed. The version numbers displayed next to the node name will show the top number is less than the bottom number. For example, **CAVI (1/3)** indicates the local watershed copy version is 1, and the master watershed version is 3. To download the files, select a highlighted node (orange), the **Master Watershed Synchronization** dialog (Figure 6) now displays an **Actions** box, from that box, click **Download**.
Yellow	Nodes highlighted in **yellow** mean that the data group has not been uploaded to the master watershed. For example, if the FIA data group displays **FIA (0/0)** or **FIA (Na/Na)**, this means there is no version for the data group on the mater watershed.



126 Figure 6 Master Watershed Synchronization Dialog - Actions Box

To upload the files, select a highlighted node (yellow), the **Master Watershed Synchronization** dialog (Figure 6) now displays an **Actions** box, from that box, click **Upload**.

In the **Team** section of the **Master Watershed Synchronization** dialog (Figure 6), when you select a data group, information about that data is displayed. In addition, the **Actions** box (Figure 6) provides you with commands that allow you to interact with the selected data group:

Upload	allows you to upload the selected data group node to the master watershed.
Download	allows you to download the latest version of the selected data group node from the master watershed.
Protect	is only available when the status of the selected data group node is **Unprotected**. This option allows you to prevent other team members from uploading changes for a data group to the master watershed. If the data group node is already protected, you will see an **Unprotect** option. Selecting **Unprotect**, will allow others to upload changes for that data group node to the master watershed.
Unprotect	is only available when the status of the data group node is **Protected**. Selecting **Unprotect**, will allow sharing of the master watershed so you can upload changes for the selected data group.

Below the **Action** box (Figure 6) is the revision information for a selected data group:

Revision	displays the revision number of the local watershed on your computer and the master watershed as "**# Of #**", respectively. The first number represents the version number of the selected data group for the local copy of the watershed on your computer. The second number is the revision number of the selected data group on the master watershed. If the two numbers are the same, this means that you have the most recent copy of the master watershed for the selected data group. If they are not the same, then you are not working on the most recent copy of the selected data group and need to download the updated data group from the master watershed.
Updated On	reports the date, time, and username of the person who last updated the selected data group on the master watershed (format - "DDMMMYYYY, HH:MM **by:** <username>"). The date is set by the team member who made the revision's (computer date and time). The username displayed is that team member's workstation login username.
Status	reports if the version of the selected data group from the local copy of the watershed on your computer is **Current** or **Out of Date**. The **Current** status means that your computer's version of the selected data group and the master watershed version are the same. The **Out of Date** status means that the current master watershed version is newer than the copy of the selected data group that is currently on your computer.
Comments	displays any comments made on the last revision on the master watershed.
History	opens the **Revision History** dialog which provides detailed information on all the revisions made to the master watershed data (further details about the **Revision History** dialog is available in the **Revision History** section of this chapter).

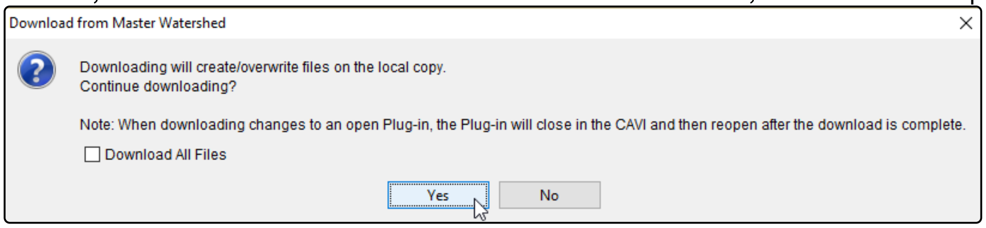
23.2.2 Downloading Base Data

If base data has been changed for the master watershed or you would like to get the latest version, you can download the data from the **Master Watershed Synchronization** dialog (Figure 1).

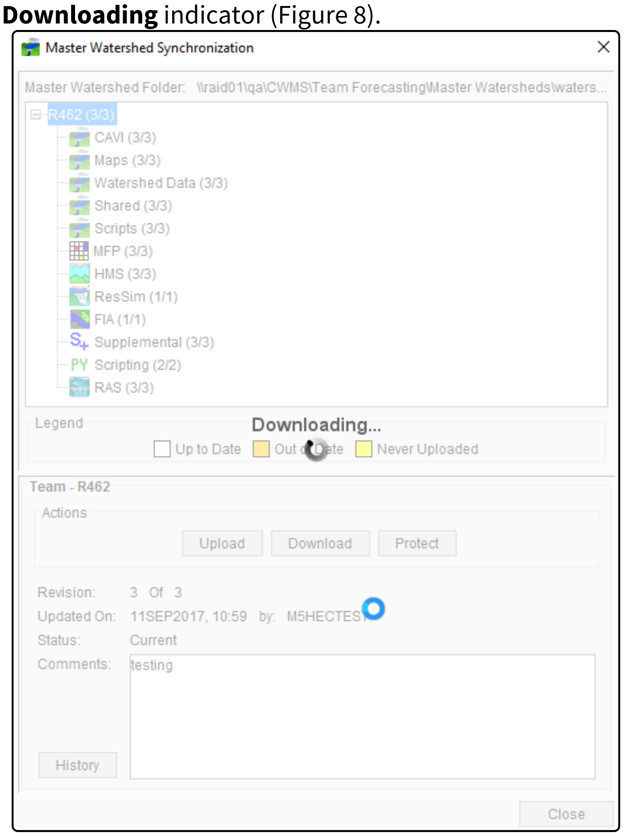
To download base data from the master watershed:

1. First, from the HEC-RTS main window, open the master watershed that is on your computer, click the **Setup** tab.
2. From the **Watershed** menu, point to **Team**, click **Sync with Master Watershed**. The **Master Watershed Synchronization** dialog will open (Figure 1).
3. From the **Synchronization Tree** (Figure 5), select the data group node that you need to download files. Hovering the mouse pointer over the nodes will display information about the data group node that will be downloaded.
4. From the **Master Watershed Synchronization** dialog (Figure 1), click **Download**, the **Download from Master Watershed** window will open (Figure 7).

5. Click **Yes**, the **Download from Master Watershed** window will close, and the download process will start.



6. During the download process, the **Master Watershed Synchronization** dialog will disable and display a **Downloading** indicator (Figure 8).



7. A **Download** tab will display in the **Message Pane**, providing status updates during the download process.
8. When the download is complete, a notification message will display (Figure 9), and the **Master Watershed Synchronization** dialog will be active again (Figure 1).

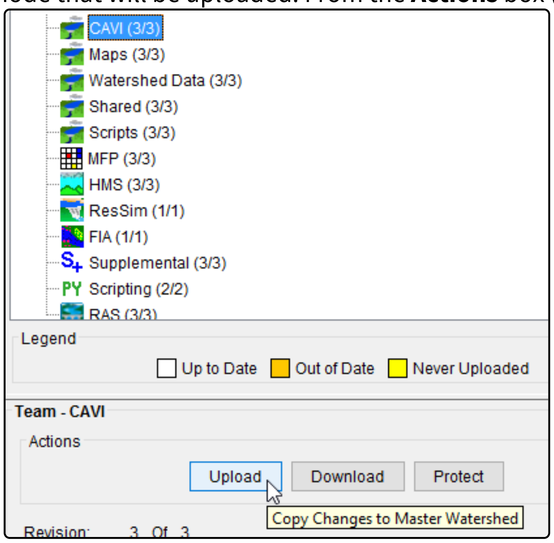
9. When you have downloaded all of the files that you need, close the **Master Watershed Synchronization** dialog (Figure 1).

23.2.3 Uploading Base Data

Changes made to the base data on the local copy of the master watershed can be uploaded to the master watershed's base data and then shared with other team members. If other team members have the same HEC-RTS study open during the upload, the team members will receive real-time notification messages indicating that the master watershed has been updated after the upload is complete.

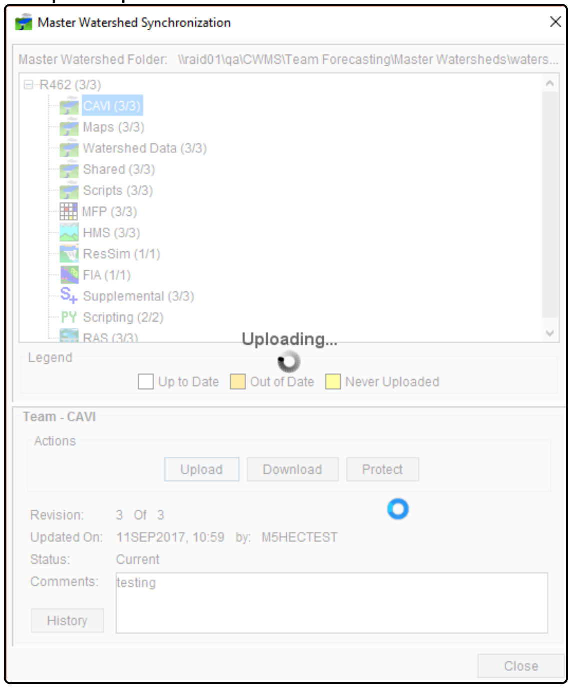
To upload changes to the master watershed's base data:

1. First, from the HEC-RTS main window, open the master watershed that is on your computer, click the **Setup** tab.
2. From the **Watershed** menu, point to **Team**, click **Sync to Master Watershed**. The **Master Watershed Synchronization** dialog will open (Figure 1).
3. From the **Synchronization Tree** (Figure 5), select the data group node that you need to upload the files for in the master watershed. Hovering the mouse over the data group node will display information about the data group node that will be uploaded. From the **Actions** box (Figure 10) click **Upload**.

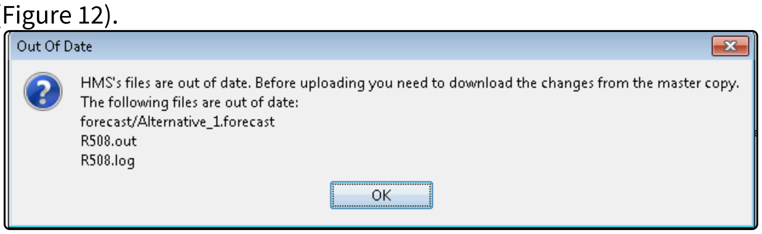


4. The **Enter Upload Comments** dialog will display. Enter comments about the files that you are uploading. These comments will be seen by other team members as part of the revision.
5. Click **OK,** the upload process will start. The watershed will close in the HEC-RTS main window and the **Master Watershed Synchronization** dialog will become inactive and display an **Uploading** indicator (Figure 11).

6. An **Upload** tab will display in the **Message Pane** (Message Pane - Upload Tab), providing status updates during the upload process.



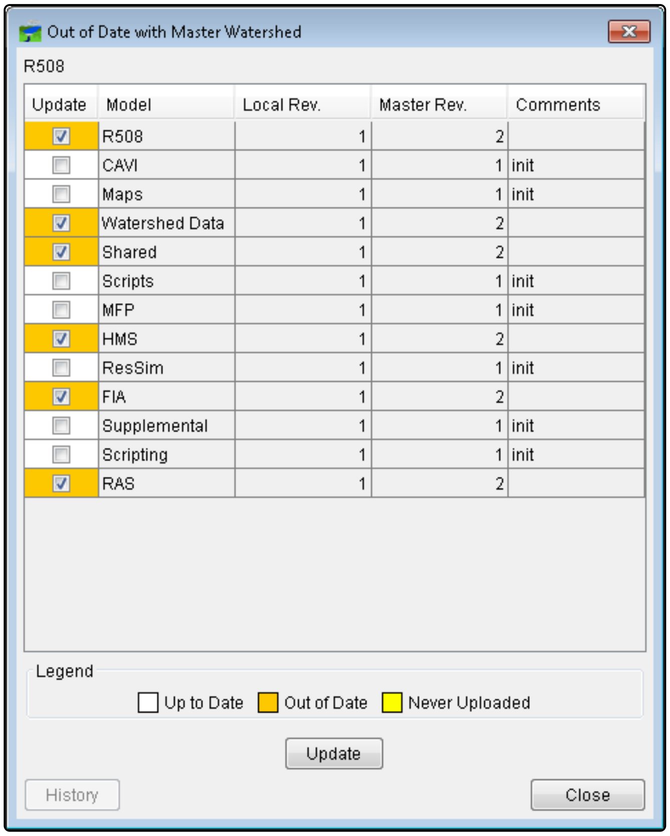
7. When the upload is complete, the watershed will reopen in the CWMS CAVI main window and the **Master Watershed Synchronization** dialog (Figure 11) will be active again. A **Real-Time Notification Message** will display (like Download Complete - Real-Time Notification Message) indicating what files have been uploaded to the master watershed. Any team members that have the same master watershed open on their desktops will also receive a notification.
8. If you do not have the most recent files, and then try to upload changes, you will get an **Out of Date** message (Figure 12).



9. This is to prevent you from accidentally deleting another team member's changes. Click **OK**, then download the outdated files. Downloading files is further explained in the section titled **Downloading Base Data**.

23.2.4 Out of Date with Master Watershed

If new files have been uploaded to the master watershed since you closed the watershed an **Out of Date with Master Watershed** dialog will display (Figure 13) when you open the watershed, prompting you to update any files that are out of date.



127 Figure 13 Out of Date with Master Watershed Dialog

The **Out of Date with Master Watershed** dialog (Figure 13) consists of the **Data Group** table and, **Update** and **History** buttons. The out-of-date data groups will have a checkmark in the **Update** column (Figure 13) of the **Data Group** table by default. To download the latest master watershed version of the data groups, click **Update** (Figure 13). All the data groups in the **Data Group** table with checkmarks will be updated. The **Out of Date with Master Watershed** dialog (Figure 13) and your watershed will close as the files will begin updating.

The columns of the **Data Group** table (Figure 13) represent the following information:

Update	Checking the box in the **Update** column (Figure 13) will select the row's data group that will be uploaded to the master watershed when uploading begins.
Model	The **Model** column (Figure 13) displays the data group name. To view the watershed folders associated with the data group, hover the mouse over a data group name in the **Model** column row, a tooltip will display showing the associated data for that data group.
Local Rev.	displays the local revision number for the data group that is on your computer's copy of the master watershed.

Master Rev.	displays the master watershed's revision number for the data group.
Comments	displays the last comment provided for the revision on the master watershed.

The highlight colors in the table are defined in the **Legend** box (Figure 13). The colors indicate the following states:

White	means that data group is currently up to date. No further action needs to be taken.
Orange	represents data groups that are out-of-date with the master watershed, which means the master watershed version is newer the watershed on your computer. To resolve this, select a row that is highlighted in orange, then click **Update**.
Yellow	represents a data group that has never been uploaded to the master watershed. To upload the data group, select a row that is highlighted in yellow, then click **Upload**.

At the bottom of the **Out of Date with Master Watershed** dialog (Figure 13) is a **History** button. The **History** button is active when a row is selected in the table. Clicking the **History** button displays the **History** dialog for the selected row. The **History** dialog displays a list of detailed information on the versions for the data group. For information on the **Revision History** dialog, see the section titled **Revision History**.

23.3 Team Forecasting

Team forecasting allows multiple modelers to work on the same forecast simultaneously. Each team member has a copy of the master watershed forecast on their computer. Changes that are uploaded to the master watershed forecast can be downloaded and modified by other team members.

The following sections go over retrieving a team forecast from the master watershed, creating a new team forecast, uploading, and downloading team forecast changes, and the available team tools to help with this process.

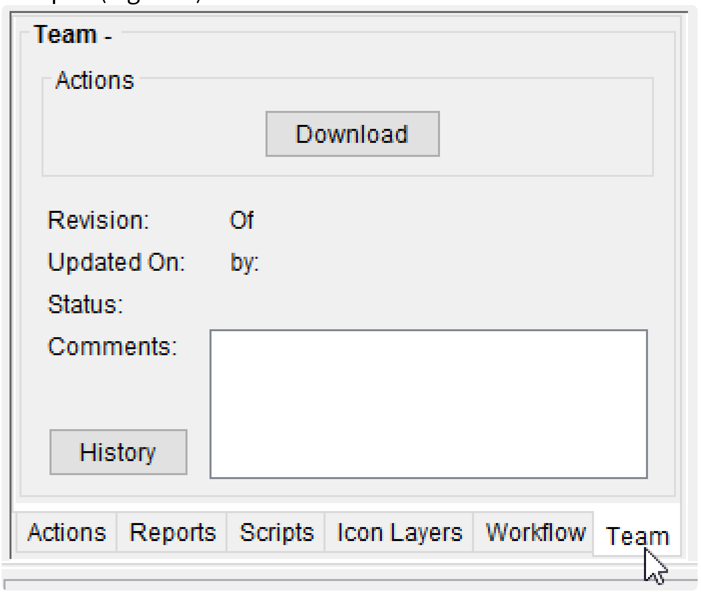
23.3.1 Retrieving a Team Forecast

Once a forecast has be uploaded to the master watershed, it can be shared with other team members as a team forecast. **Note:** When retrieving team forecasts from the master watershed, a forecast cannot be open in the **Modeling** module. If a forecast is open, you will be unable to download a team.

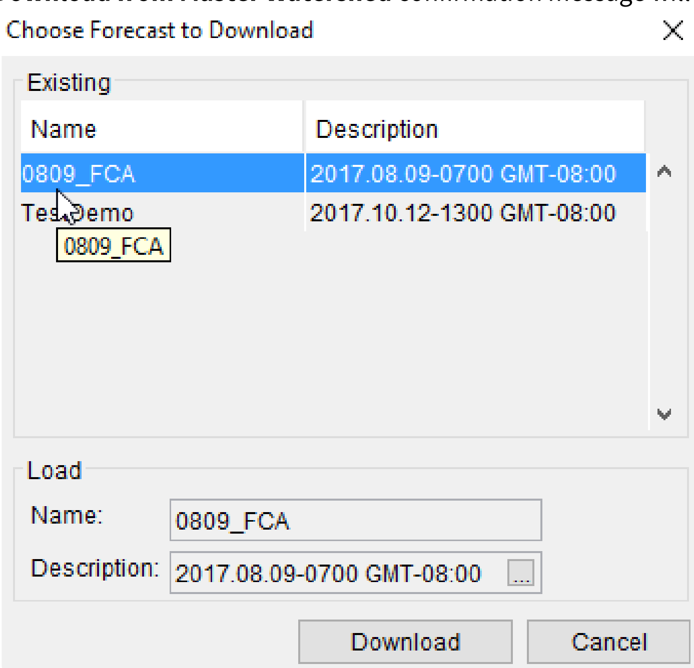
To download a team forecast from a master watershed:

1. First, from the HEC-RTS main window, open the master watershed that is on your computer, click the **Modeling** tab. Do not open a forecast, if you have a forecast open, close the forecast.

2. From the **Forecast Tabs** (Figure 1), click on **Team**. Click **Download**, the **Choose Forecast to Download** dialog will open (Figure 2).



3. From the **Existing** box (Figure 2), select the forecast you would like to download from the list, click **Download**. A **Download from Master Watershed** confirmation message will display.



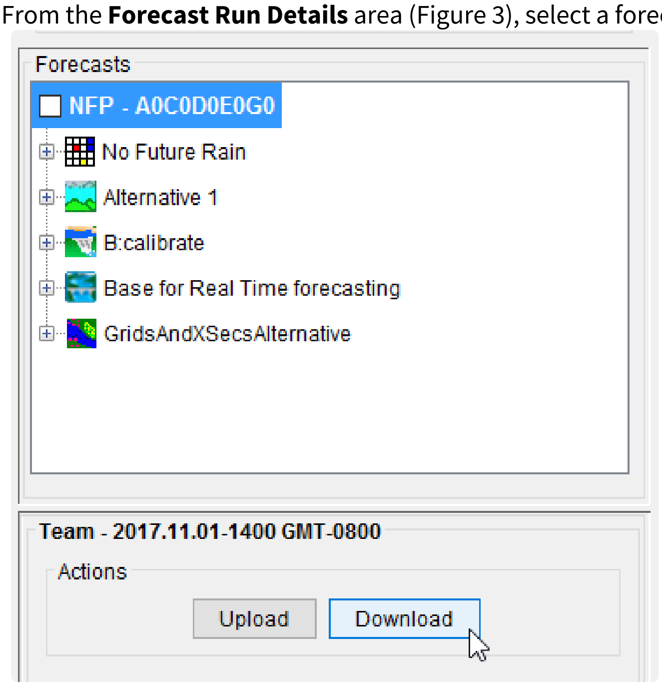
4. Click **Yes,** the download will begin. A **Download** tab will display in the **Message Pane**, providing status updates during the download process.
5. When the download is finished a **Download Complete** window will display. Click **Yes**, the forecast will open.

23.3.2 Creating a Team Forecast

To create a team forecast, use your local copy of the master watershed, and create a forecast as you normally would. Once the forecast is created you can upload it to the master watershed before or after you compute it.

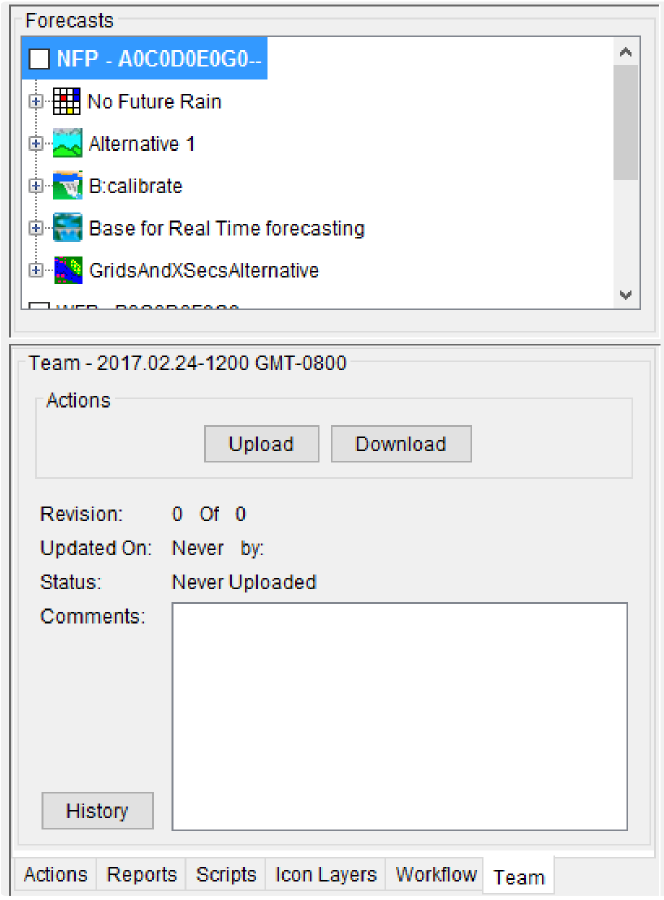
To create a team forecast:

1. First, from the HEC-RTS main window, open the master watershed that is on your computer, click the **Modeling** tab. Steps for creating a new forecast are detailed in HEC-RTS Forecasts.
2. From the **Forecast Run Details** area (Figure 3), select a forecast. From the **Forecast Tabs** (Figure 1), click **Team**.



3. Click **Upload** (Figure 3), the **Enter Upload Comments** dialog will appear open.
4. In the **Enter Upload Comments** dialog, enter information about the upload click **OK**.
5. The forecast will temporarily close in HEC-RTS and the upload will begin. Once the upload is complete, the forecast will display in the **Modeling** tab (Figure 4). More team forecasting information will display on the **Team**

tab now that the forecast is a team forecast.



23.3.3 Uploading Forecast Changes

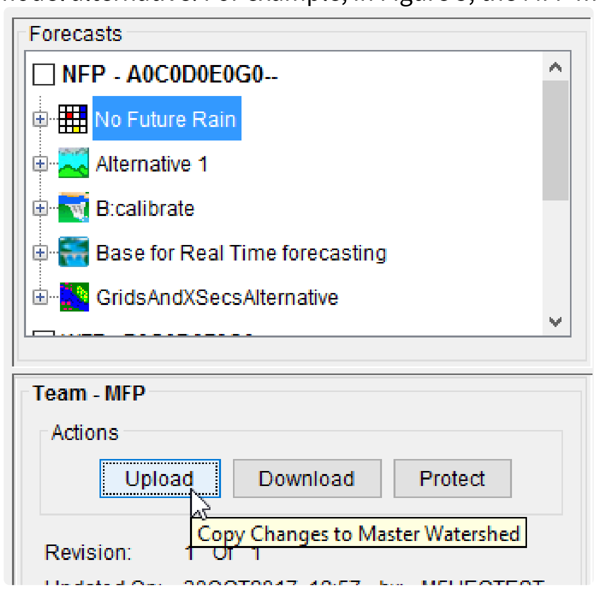
Changes made to the team forecast on your computer can be uploaded to the master watershed and shared with other team members. If other team members have the forecast open in HEC-RTS, the team members will receive a notification message that the master watershed has been updated. The uploaded forecast changes can include all the forecast data or a single model alternative's data. When uploading a single model alternative's data, only the model alternative's data and the model alternative's DSS records will be uploaded to the master watershed. If uploading the entire forecast, all model alternatives and the entire *forecast.dss* file will be uploaded.

Note: Before making changes and uploading forecast data, the **status** displayed in the **Team** tab needs to read **Current**. The **Current** status means you have a copy of the most recent changes from the master watershed on your computer. This copy can be changed once it is downloaded. Best practice would be to download the forecast to the status **current**, and then make changes to the model alternatives. This prevents you from unintentionally erasing data that has been changed since you last updated your local data files.

To upload the changes for the team forecast to the master watershed:

1. First, from the HEC-RTS main window, open the master watershed that is on your computer, click the **Modeling** tab. From the **Forecast** menu, open a team forecast.

2. From the **Forecast Run Details** area (Figure 5), select a forecast. From the selected forecast tree, click on a model alternative. For example, in Figure 5, the MFP model alterative has been selected.



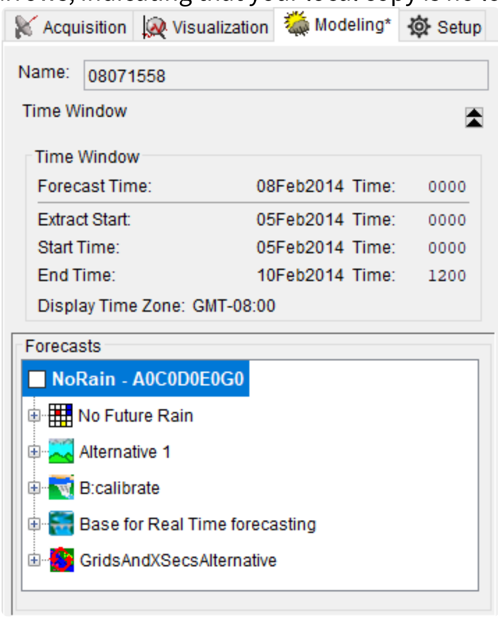
3. From the **Actions** box (Figure 5), click **Upload**, the **Enter Upload Comments** dialog will open. Enter comments on the upload in the text field. When you are done, click **OK**, the upload will begin.

Note: You can upload ALL data for the selected forecast by checking the **Upload All Files** checkbox located on the bottom left-hand side of the **Enter Upload Comments** dialog. This will override all the data for that forecast model in the master watershed, not just the changes.

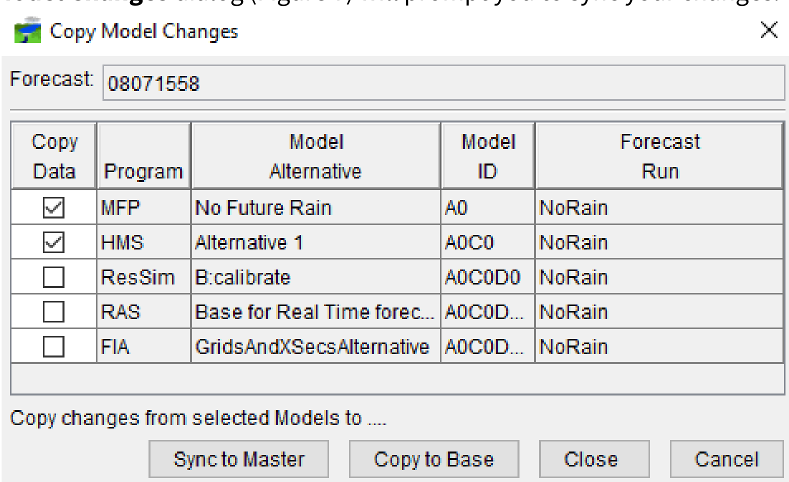
If you try to upload changes for a model alternative and the current local revision number does not match what is on the master watershed, you will be notified that your model alternative is out-of-date and prompts you to download the latest changes from the master watershed.

1. When the upload starts, the forecast will close and an **Upload** tab will appear in the **Message Pane** on the CAVI. The **Upload** tab displays information on the files uploading.
2. When the upload is complete, the real-time notification messages will appear on the bottom right-hand side of your monitor screen and the forecast will be re-opened and displayed again in HEC-RTS.

3. If changes have been made within a team forecast, the affected model alternative's icon will display two red arrows, indicating that your local copy is no longer current (Figure 6).



4. When attempting to close a team forecast without first uploading changes to the master watershed, the **Copy Model Changes** dialog (Figure 7) will prompt you to sync your changes.



To upload the changes to the master watershed from the **Copy Model Changes** dialog (Figure 7):

1. Select the model alternatives to sync with the master watershed by selecting the check next to their name. Any model alternatives that do not match the copies in the master watershed will be checked by default. The MFP and HMS model alternatives need to be synced with the master watershed in the example (Figure 7).
2. **Copy to Base** updates changes in model files from the forecast folders back to the watershed "base" model folders.
3. Click **Sync to Master**, an **Enter Upload Comments** dialog will open for each model alternative selected to sync.
4. Enter upload comments and click **OK** to start the upload process. An **Upload** tab will display in the **Message Pane**. The tab will display the status message during the upload.

5. If your local files are not current, you will receive an **Out Of Date** warning reminding you to sync with the master copy before making and uploading changes.

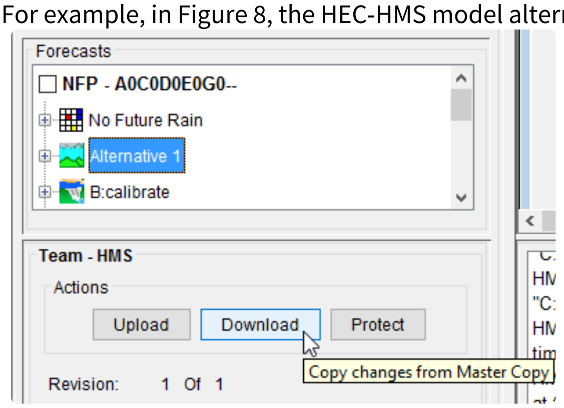
23.3.4 Downloading Forecast Changes

Changes made to the team forecast on the master watershed can be downloaded to your computer's team forecast copy. You can download the entire forecast or a single model alternative's data. When downloading a single model's data, only the model alternative's data and DSS records will be downloaded to your local copy. If downloading the entire forecast, all model alternatives and the entire *forecast.dss* file will be copied.

Note: It is important to download any files that have been updated before you start revising the files on your copy. This is to ensure you do not accidentally override another modeler's changes.

To download team forecast changes to your local copy of the forecast:

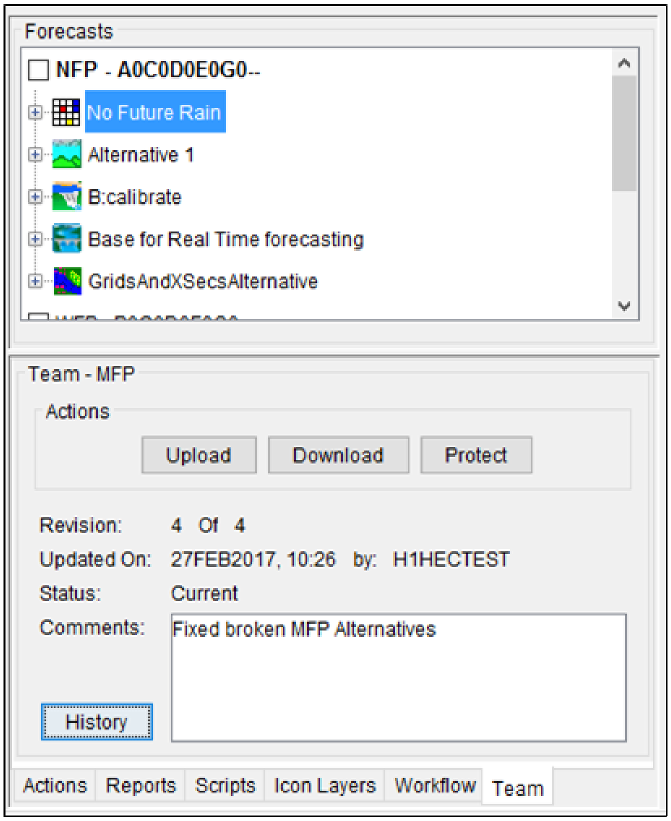
1. First, from the HEC-RTS main window, open the master watershed that is on your computer.
2. Select the forecast run or model alternative node in the **Forecast Tree** you would like to download changes for. For example, in Figure 8, the HEC-HMS model alternative is selected.



3. From the **Team** tab, click the **Upload** button in the **Actions** (Figure 8).
4. The **Download from Master Watershed** message will appear.
5. Click **Yes** to continue with the download. If you would like to override all files for the selected forecast node you are downloading, check the **Download All Files** checkbox before clicking **Yes**.
6. The forecast will close in the HEC-RTS main window and the download will start. A **Download** tab will display in the **Message Pane** that displays information about the download.
7. When the download is complete, the forecast will re-open in the HEC-RTS main window and a real-time message will display on the bottom right-hand side of your screen (Download Notification Message).
8. The download will replace any out-of-date files for the selected **Forecast Tree** node on your computer with the files from the master watershed and the **Revision** information on the **Team** tab will update. Only the files that have had a revision update on the master watershed will be downloaded.

23.3.5 Team Forecast Tab

The **Team** tab (Figure 9) is located at the bottom of the **Modeling** tab and provides access to team forecasts, and allows you to upload, download, and protect forecast data. It also provides detailed information on the forecast data's revision information and history. Each of these features are explained in more detail in the following sections.

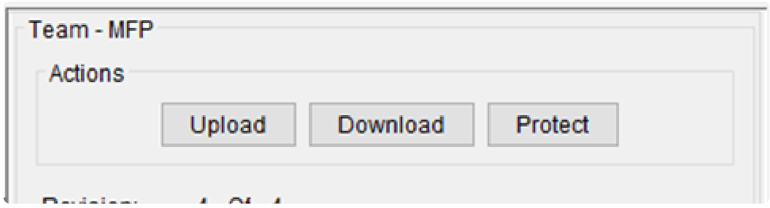


The **Forecasts** box (Figure 9) displayed at the top of the **Team** tab displays the **Forecast Tree**, which displays the different forecast runs and model alternatives for the forecast. When a node is selected in the **Forecast Tree**, the bottom half of the tab will populate with information about forecast teaming on that selected forecast data. For example (Figure 9), if a forecast run node is selected, the **Team** area reflects the information for that specific forecast run. If a model alternative is selected in the tree, the **Team** tab will display information on that model alternative in the forecast run. If there are no team forecasts open and the watershed is a master watershed, the **Team** tab will display a **Download** button that allows you to select a pre-existing team forecast on the master watershed and download it to your computer.

For example, in Figure 9, displays the MFP model alternative "*No Future Rain*" selected in the **Forecast Tree** and the **Team** tab displaying information on that model alternative.
The following sections go over the different features provided in the **Team** tab in detail.

23.3.5.1 Action Buttons

Depending on what is selected in the **Forecast Tre**e, the **Actions** area provides **Upload**, **Download**, **Protect**, and **UnProtect** buttons (Figure 10). These different options allow you to interact directly with the master watershed forecast.



The **Actions** box includes the following:

Upload	allows you to upload a new forecast or changes made to an existing team forecast to the master watershed. When a forecast run is selected in the **Forecast Tree**, the **Upload** button will upload all changed data for the forecast run to the master watershed. If a single model alternative is selected in the **Forecast Tree**, the **Upload** button will upload only changes made to that model alternative.
Download	allows you to download the whole team forecasts or its individual model alternative updates, depending on what is selected in the **Forecast Tree**. If a forecast is not opened in the Modeling module, the **Download** button allows you to download team forecasts available for the master watershed.
Protect	prevents others from uploading any changes to the forecast on the master watershed. You can protect a single model alternative or the entire forecast.

If a forecast or model alternative is already protected, you will see an **UnProtect** option.

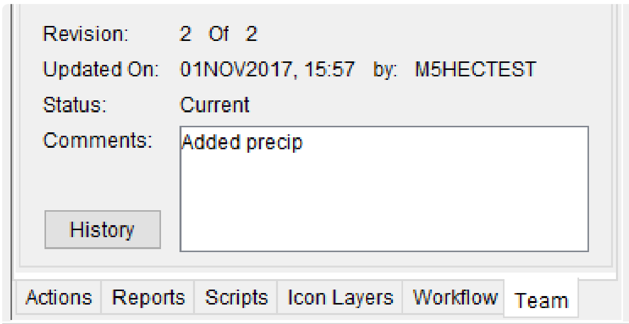
UnProtect	allows you to begin uploading changes to the forecast or model alternatives on the master watershed again.

Once you unprotect the data, **Protect** becomes available again.

If a forecast is not open on the **Modeling** tab, only the **Download** button will be available to download team forecasts from the master watershed.

23.3.5.2 Revision Pane

The **Team** tab also displays information on team forecast revisions. This information can be found directly underneath the **Actions** box (Figure 11).

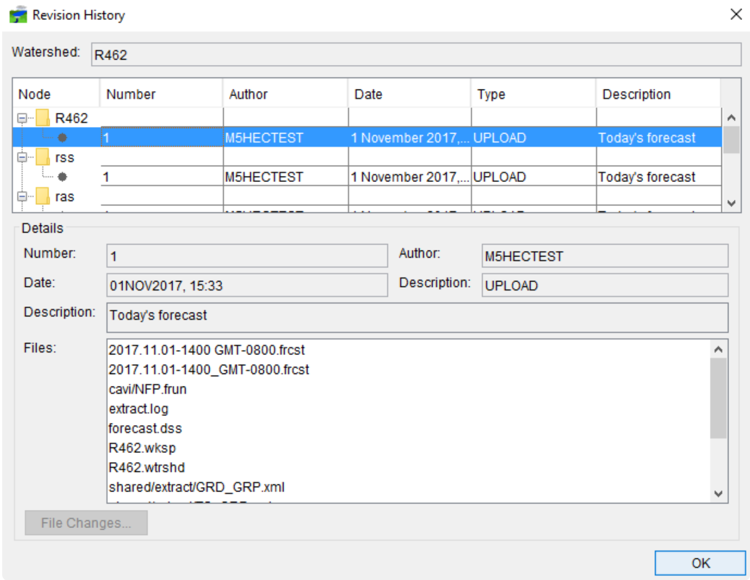


The revision information displays for the selected node in the **Forecast Tree**. The information includes the following:

Revision	The **Revision** displays **# Of #.** The first number represents the version number you have on your local computer's copy. The second number is the revision number of the forecast data on the master watershed copy. If the two numbers are the same, this means that you have the most recent copy of the master watershed data. If the first number is smaller than the second, the master watershed's copy is newer than your copy.
Updated On	The **Updated On** information, in the format "DDMMMYYYY, HH:MM **by:** <username>", reports the date, time, and username of the person who last updated the forecast data on the master watershed. The date is set by the team member who made the revision's computer date and time. The username displayed is that team member's workstation login username.
Status	The **Status** reports if the version on your computer is **Current** or **Out of Date**. The **Current** status means that your computer's version of the watershed data and the master watershed data are the same. The **Out of Data** status means that the master watershed is newer than the copy that is currently on your computer.
Comments	this text area displays any comments made on the last revision on the master watershed.
History	The **History** button will open the **Revision History** dialog which provides detailed information on all the revisions made to the master watershed data. The Revision History dialog is described in further detail in the next section, **Revision History**.

23.3.5.3 Revision History Dialog

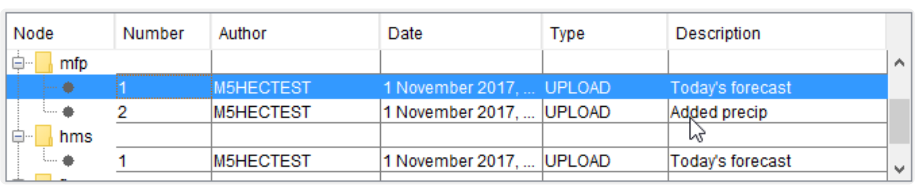
The **Revision History** dialog (Figure 12) is available on the forecast's **Team** tab (Figure 1) from the **History** button. It is also available on the **Out of Date with Master Watershed** dialog. It displays detailed information on the versions for the watershed data selected in the forecast or **Synchronization Tree**.



Clicking on a row of the table populates the bottom section of the **Revision History** dialog with expanded information from the columns: **Node, Number**, **Author**, **Date**, **Type,** and **Description**. Expanded descriptions of the features of the **Revision History** dialog can be found in the following sections.

23.3.5.4 Revision Table and Details Panel

The **Revision Table** (Figure 13) displays information on a revision that has been made. The revisions are grouped by forecast and then model alternative. When a row is selected in the table (Figure 13), the information on the **Details** panel (Figure 12) of the **Revision History** dialog is populated.



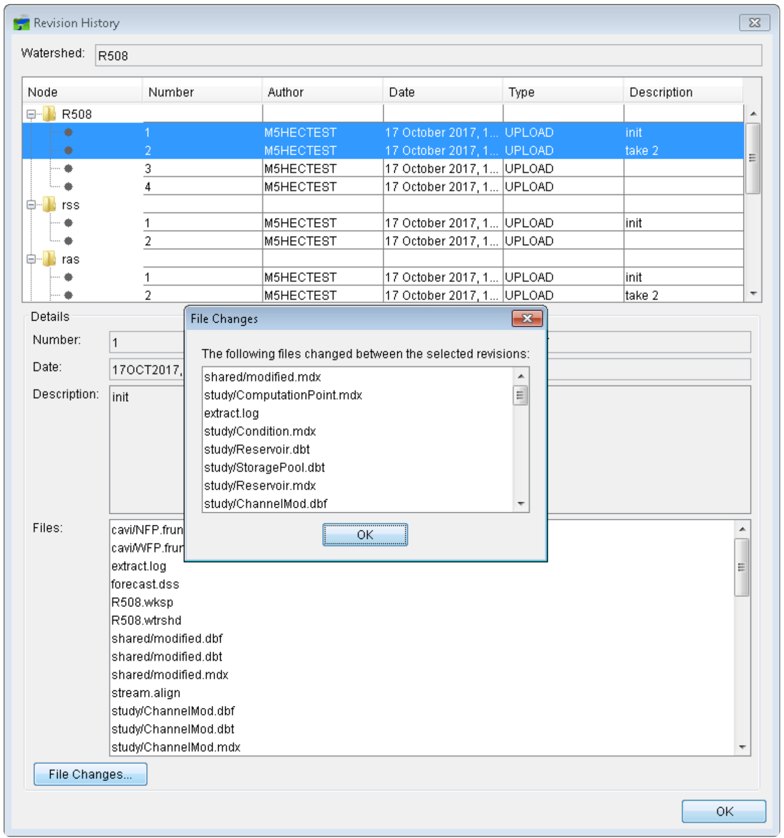
The table (Figure 13) and panel (Figure 12) contain the following information:

Node	is the first column of data in the **Revision Table** (Figure 13) and provides information about the data group that was revised. Revisions are made and grouped by forecast run and models.
Num ber	displays the revision the row represents. The number assigned is based on the order that that revision to the master watershed was made.
Auth or	this column tells you which modeler submitted the revision to the master watershed data.

Date	date of the revision made by the team member who made the revision's (computer's date and time).
Type	indicates what caused the revision change. There are five types that can cause a revision change: **UPLOAD**, **DOWNLOAD**, **PROTECTED**, and **UNPROTECTED**. The **UPLOADED** option indicates the revision changed when the modeler uploaded new data to the master watershed. The **PROTECTED** option displays when the modeler changed the status of the data to Protected Mode. It prevents others from uploading changes to the master watershed data. The **UNPROTECTED** option displays when a modeler has changed the status of the data out of the **Protected Mode**. This means that others can start to upload changes to the master watershed again.

Additional information at the bottom of the **Revision History** dialog (Figure 13) includes: **Description**, **Files**, and **File Changes**.

Description	The **Description** box (Figure 12) is filled in by the team member that made that revision. The description cannot be changed later.
Files	The **Files** box (Figure 12) contains a list of the modified files in the text field. The **Files** box only populates with the information of the revision that has been selected above in the **Revision Table** (Figure 13).
File Changes	To display a list of files that were changed between the selected revisions in the **Revision Table** (Figure 12), from the **Revision History** dialog (Figure 12), click **File Changes**. The **File Changes** dialog (Figure 14) will open displaying the files that were changed between the selected revisions. **File Changes** is available to you when a revision row has been selected in the **Revision Table** (Figure 13).



23.4 Protecting Data

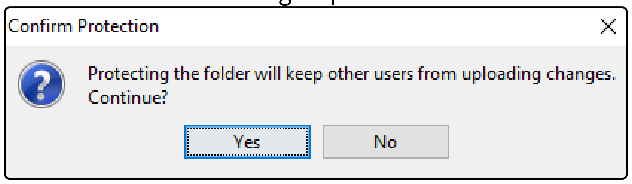
Once the base data or team forecast is in a state where you do not want others to uploaded changes to it on the master watershed you can set the **Protect** flag on the data. Setting the **Protect** flag on watershed data will stop anyone from uploading changes to the master watershed. You can still download the watershed to the local computer and make changes to it, but you will not be able to upload the changes to protected data groups.

23.4.1 Protecting Base Data

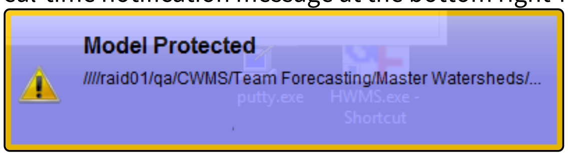
To set protect on base data:

1. From the **Setup** tab, from the **Watershed** menu, point to **Team**, click **Sync to Master Watershed**, the **Master Watershed Synchronization** dialog will open.
2. On the **Master Watershed Synchronization** dialog, select a data group node in the **Synchronization Tree**. The bottom of **Master Watershed Synchronization** dialog) will display information on the selected group.

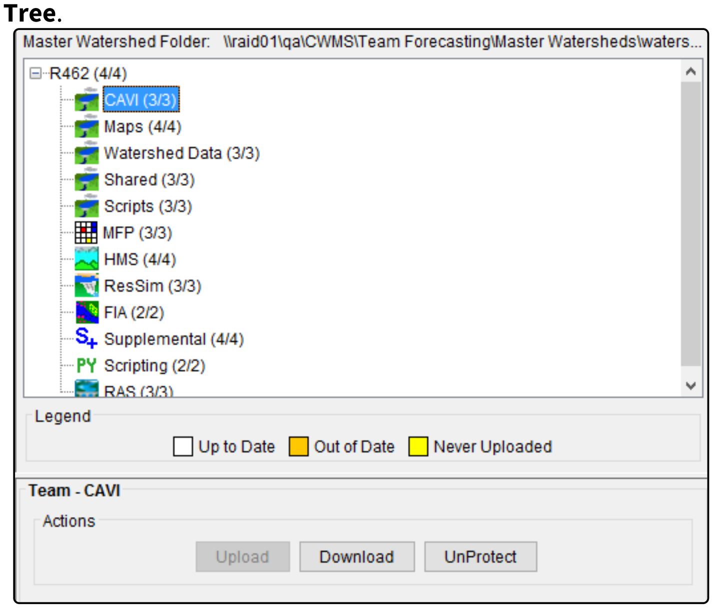
3. Click **Protect**, the **Confirm Protection** window (Figure 1) will display. The window is asking you would like to protect the selected data group's folder.



4. Click **Yes**, the **Confirm Protection** window will close and when the protection process is complete, you will see a real-time notification message at the bottom right-hand of your screen (Figure 2).



5. The button's label will change to "UnProtect", meaning the data is now protected from uploads and the **Upload** button, pictured in Figure 3, will no longer be available when the data node is selected in the **Synchronization Tree**.

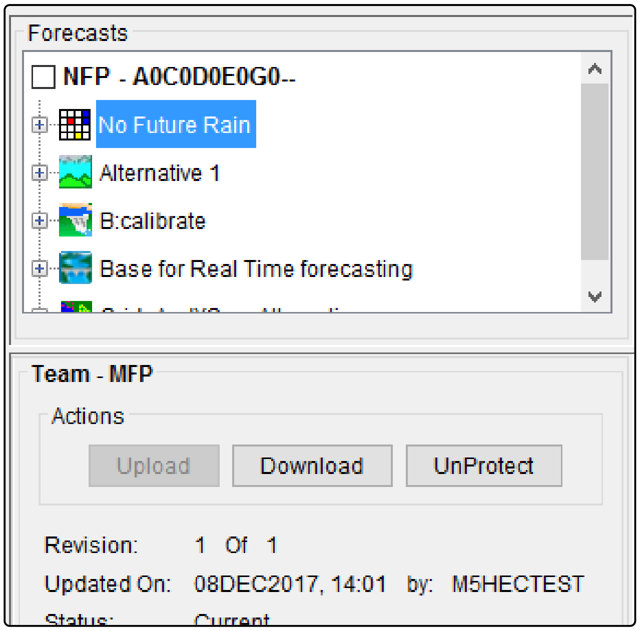


23.4.2 Protecting Forecast Data

To set **Protect** on forecast data:

1. Open a team forecast in the **Modeling** tab.
2. Select the model alternatives or forecast run in the **Forecast Tree** that you would like to protect. The **Team** tab located in the bottom tab group will fill with information on the selected data.
3. Click the **Protect** button. A **Confirm Protection** window (Figure 1) will appear, confirming you would like to protect the forecast or model alternative folder.
4. Continue with the protection process by clicking **Yes** on the **Confirm Protection** dialog (Figure 1).The dialog will close and when the protection process is complete, you will see a real-time notification message at the bottom right-hand of your screen (like Figure 2).

5. The button's label will change to "UnProtect", meaning the data is now protected from uploads and the **Upload** button, pictured in Figure 4, will no longer be available when the forecast or model alterative is selected in the **Forecast Tree**.

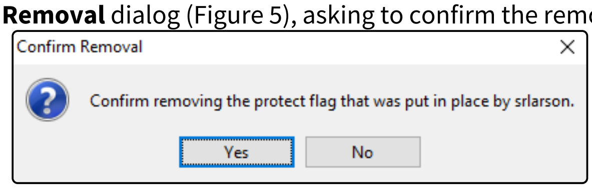


23.4.3 Unprotecting Data

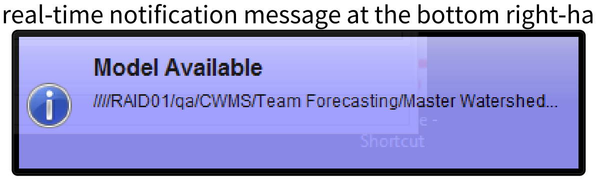
Unprotecting data is like protecting data, data can be set as "Unprotected". When **Unprotect** is set, the master watershed will accept uploads to the data group again. Unprotect is only available if the data has been previously protected.

The process is like protecting the data and is described previously. The **Protect** button displays as an **UnProtect** button when the protected node is selected in the synchronization or forecast trees. To unprotect data:

1. First open the editors you found the Protect button in by following the steps in the previous two sections, Protecting Base Data or Protecting Forecast Data.
2. Select the data you would like to turn the protection off for. Then click the **UnProtect** button. The **Confirm Removal** dialog (Figure 5), asking to confirm the removal of the protection flag will appear.



3. Clicking **Yes**, the **Confirm Removal** dialog will close. When the unprotect process is complete, you will see a real-time notification message at the bottom right-hand of your screen (Figure 6).



4. The button's label will change to "Protect", meaning the master watershed is now accepting uploads for that data. The **Upload** button will be available when the data is selected in the tree.

23.5 Technical Notes on Versioning

When a watershed is configured to be a master watershed, HEC-RTS will keep track of the different versions of the watershed that exist, the main copy on the network drive and the local copies on the team member's workstations. HEC-RTS manages all the changes made to HEC-RTS files, but not files associated with the model alternatives (i.e., HEC-HMS, HEC-ResSim, HEC-RAS). HEC-RTS does not know what changes have been made since editing is done through the native models' interfaces. If you update your local copy, the version number on the local copy will not change, and the team synchronization dialogs will not indicate that your forecast or watershed files are more recent than the master. Future development goals include providing more complete information about differences between the local and master files.

When new files for a watershed are uploaded, HEC-RTS compares the local files with those on the master watershed. The size and sum of the local files are compared with the size and sum of the files on the server. If there are any differences, the files are updated. For downloads, the local and master watershed files are compared. If the version numbers are different, the system updates all of files that have undergone revisions between your local version and the version on the network drive. The following sections provide further details on these concepts.

23.5.1 Master Watershed Team Modeling Files

As part of the upload process for master watershed, there are new files created to track versions. In the master watershed, the main watershed directory and all sub-directories will contain a file labeled *.block.revision.dss*. These *.block.revision.dss* files contain a running history of all the file changes that have been made for the directory. Each time an upload occurs for a selected folder, the *.block.revision.dss* file will log a new entry that contains the name of the files that were changed on the master watershed, the user that performed the upload action, the date and time of the upload, and any comments entered at time of upload.

23.5.2 Local Watershed Team Modeling Files

In addition to the files that are created in the master watershed at the time of the upload, there are new files created for the local copy in the main watershed directory on your computer and all sub-directories as well. These files are labeled *.block.revision*. Unlike the DSS files on the master watershed, the local copy *.block.revision* files are XML files. The *.block.revision* files will contain information about the last update to the specified local folder, instead of a complete history like on the master watershed.

23.5.3 Version Information

CWMS manages the different master watershed *.block.revision.dss* and local *.block.revision* files to determine the proper versioning for your local copy of the master watershed. The **Master Watershed Synchronization** dialog (described in the section titled, **Master Watershed Synchronization Dialog**) will display information from both the local and master watershed revision files. When a team member selects a node in the **Master Watershed Synchronization** dialog, it will display the current revision number as defined locally and on the master watershed, the last updated date and time, the username who made the changes, Status, and Comments. If you click the **History** button at the bottom of the window, a **Revision History** dialog will launch. This dialog will display information from the master watershed's

.block.revision.dss file, which contains the complete version history for the selected node. From the **Revision History** dialog, you can select two different uploads and click the **File Changes** button to see the differences between the two revision numbers. Please see the section titled **Revision History** for more information on using this dialog.

24 Glossary

The following are definitions of terms used in this manual. The terms are defined in context with HEC-RTS.

Term	**Definition**
Acquisition module	The Acquisition module in HEC-RTS is the grouping of commands for data acquisition. These are commands for monitoring data acquired by HEC-RTS (and DSS files for gridded data), validating the quality of acquired data, and editing the data, as needed.
Arc shapefile	Arc shapefiles is the native data structure for the ArcGIS® software. Shapefiles store non-topological geometry and attribute information for the spatial features of a data set. A shapefile is a group of files that share a common base name and different extensions. At a minimum a shapefile has three component files: *.dbf*, *.shp*, and *.shx*. The *.shp* file contains the shapes (point, polylines, or polygons) that are displayed in the layer. Each shape has a record in the *.dbf* database file containing descriptive data called "attributes". The *.shx* file contains an index that links each shape to its record in the *.dbf* file.
Base data	Base data for an HEC-RTS study refers to the original set of data from which copies are made to create forecasts.
Baseflow method	The baseflow method is a model for calculating the subsurface flow contribution to total runoff flow in a subbasin.
Basin model	Basin models are one of the main components in an HEC-HMS project. Basin models are composed of subbasins, routing reaches, junctions, and other elements. Their principal purpose is to predict how meteorologic inputs are transformed into streamflow at specific locations in the watershed.
Cache	A cache is a temporary storage area where recently accessed watershed and data files are copied and stored.
Calibration	Calibration is the systematic adjustment of model parameters to produce model results that more accurately reflect watershed and channel behavior.
Common schematic	A common schematic consists of a stream alignment, map layers, time series icons, computation points, impact areas, and projects that allow sharing of common data between HEC software (e.g., HEC-HMS and HEC-ResSim) for a watershed. Use of a common schematic ensures consistency of computation nodes, routing reaches, and so on.
Computation point	A computation point is a location where time series data are exchanged between analysis applications.

Coordinated Universal Time (UTC)	UTC is a time scale that is the basis for the worldwide system of civil time. To provide a consistent method of assigning time to data, the HEC-RTS default time zone is UTC. No matter what time zone the data are originally measured in, the data will always be stored in UTC in the database. The time assigned to data can be converted to local time or another time zone when storing or retrieving data from the database.
Corps Water Management System (CWMS)	CWMS is a real-time decision support system developed by HEC. CWMS makes readily available to USACE staff information about the current state of watersheds, likely future state of watersheds, and consequences of management actions. The information helps water managers make wise operation decisions.
Culvert	A culvert is a conduit that conveys stream flow through a roadway embankment or past some other type of flow obstruction.[4]
Data Storage System (DSS)	DSS is a database system developed by HEC to store and retrieve efficiently scientific data that are typically sequential. Such data types include, but are not limited to, time series data, curve data, spatial-oriented gridded data, and others. HEC designed the system to make it easy for users and application programs to retrieve and store data. DSS is incorporated into most of HEC's major software applications.
Data Storage System (DSS) Pathname	A DSS pathname is a unique reference to a record or time series in a DSS file. The pathname is structured so that it is easy to group related data together. A pathname is separated into six parts (separated by slashes) labeled "A" through "F", as follows: /A/B/C/D/E/F/.
Data Storage System Visual Utility Engine (HEC-DSSVue)	HEC-DSSVue is a software utility developed by HEC that provides a graphical user interface for viewing, editing, and manipulating data in DSS files.
Data validation	Data validation is the process of reviewing data and editing or deleting data that are erroneous.
Datum	Datum refers to the reference specifications of a measurement system, usually a system of coordinate positions on a surface (a horizontal datum), or heights above or below a surface (a vertical datum).[3]

Directory	A directory catalogs the computer files in a nested folder-like structure. Files are organized by storing related files in folders in the same directory.
Diversion	A diversion removes and redirects water from its normal course in a channel.
Domestic Satellite Receive-Only Terminal (DROT)	DROT is software that facilitates receipt of satellite data.
Easting	Easting is the x coordinate of a location in the user-selected World Coordinate System.
Flood Impact Analysis (HEC-FIA)	HEC-FIA is an HEC analysis application that assesses consequences of flow or water surface elevations in the system. Inputs include computed or observed flows or water surface elevations at critical locations. Outputs include economic, life loss, or other measures of impact, or optionally, information on actions to be taken in response to flows or elevations that will be experienced.
Flow blending	Flow blending is the capability of CWMS to transition smoothly from observed flows during the "lookback period" to simulated flows during the forecast period. The lookback period is the period occurring prior to the time of forecast.
Forecast	A CWMS forecast is a simulation of watershed processes and consequences of flooding based on input data and hydrologic, reservoir operation, hydraulic, and impact analysis models configured in CWMS. Forecast results include flow and stage in the channel from watershed runoff, reservoir release schedules, floodplain inundation maps, floodplain consequence reports, and reports listing actions for emergency responders to take. These results inform water management decision making.
Forecast alternative	A forecast alternative is a single forecast scenario defined by a specific set of data and model alternatives. A second forecast alternative would be defined by different data or model alternatives.
Freeboard	Freeboard is the vertical distance from the water surface to the top of the channel, levee, or floodwall at design condition, and serves as a factor of safety for containing water in the stream without overtopping the channel, levee, or floodwall.[1]
Geographic Information System (GIS)	GIS is an integrated collection of software and data used to view and manage information about geographic places, analyze spatial relationships, and model spatial processes.[3]

Geo-referenced grid	A geo-referenced grid is a set of geographic data that are represented as a raster and have been linked to a coordinate system for viewing, querying, and analyzing with other geographic data.
Gridset	A gridset is a sequence or time series of georeferenced grids that describe properties or states of a watershed as they vary spatially and through time. Examples include sequences of precipitation or air temperature grids.
HEC-GridUtil	HEC-GridUtil is a software application developed by HEC that allows you to view, process, and analyze gridded data sets stored in DSS format.
Hydrograph	A hydrograph is a relationship of stage, discharge, velocity, or other state of water with respect to time at a certain location.
Hydrologic Engineering Center (HEC)	HEC is a USACE designated Center of Expertise in the technical areas of surface and groundwater hydrology, river hydraulics and sediment transport, hydrologic statistics and risk analysis, reservoir system analysis, planning analysis, real-time water control management, and a number of other closely associated technical subjects. HEC is the developer of CWMS.
Hydrologic Modeling System (HEC-HMS)	HEC-HMS is an HEC analysis software application that simulates watershed response to precipitation. Inputs may include observed or forecasted precipitation, temperature, snowpack, and other environmental conditions. Outputs include flows throughout the watershed, including inflows to reservoirs.
Impact area	In HEC-RTS, an impact area is a distinct portion of a watershed in which rising stage in a stream, river, lake, or reservoir has a consequence that is to be assessed. The boundaries of an impact area are defined by a closed polygon. You characterize the effects of rising stage by describing the property and population at risk in the area bounded by the polygon as input to HEC-FIA.
Inundation	Inundation is a condition in which water covers normally dry land, primarily caused by severe events along rivers or the coast.[6]
Java Archive (JAR, .jar)	JAR is a platform-independent file format that aggregates many files into one. Multiple Java applets and their requisite components (*.class* files, images, and sounds) can be bundled in a JAR file. The JAR format also supports compression, which reduces file size.[8]
Layers	Layers in HEC-RTS are like transparencies laid one on top of the other. Each includes representations of static physical images such as roads, county and state boundaries, rivers, and subbasins. These are layered in the display area as color pictures. See Chapters 3, 4, and 5 for more information about layers.

Layer tree	A layer tree is a list representing the hierarchical arrangement of layers displayed in HEC-RTS.
Levee	A levee is a manmade barrier built along a stream to reduce flooding in the adjacent floodplain from rising water in the stream.
Log file	A log file is a file that holds messages produced by the HEC-RTS software. The types of messages include information-only messages and error messages.
Lookback period	The lookback period is the period occurring prior to the time of forecast.
Loss method	The loss method is a model for calculating infiltration and other abstractions of precipitation in a subbasin.
Map layer	A map layer is a digital map of watershed features that is configured to display in HEC-RTS. A map layer is displayed as a color picture. Map layers help you visualize your watershed.
Map projection	A map projection uses mathematical formulas to relate spherical coordinates on the globe to flat, planar coordinates.[2]
Map Window	The map window is the window in the HEC-RTS that displays the graphical (and usually georeferenced) representation of the watershed.
Meteorologic Forecast Processor (MFP)	MFP is an HEC analysis software application that processes meteorological forecasts for input to HEC-HMS. Inputs are forecasted meteorological data such as precipitation and temperature. Outputs are forecasted meteorological time series formatted for compatibility with HEC-HMS.
Meteorologic model	Meteorologic models are one of the main components in an HEC-HMS project. Their principal purpose is to prepare meteorologic boundary conditions for subbasin runoff computations.
Model alternative	A model alternative is a single model configuration with a specific set of input and parameters. A second model alternative would be defined by a different configuration or different input or parameters.
Modeling module	The Modeling module in the CAVI is the grouping of commands for model execution. These are commands for setting up and executing analysis applications and viewing results.

Module	Commands in the CAVI are organized by function: watershed setup, data acquisition, data visualization, and model execution. A module is the grouping of commands related to a function. The CAVI contains the Setup module for watershed setup, the Acquisition module for data acquisition, the Visualization module for data visualization, and the Modeling module for model execution. Each module has menus, tools, and a display area specific to its function.
Northing	Northing is the y coordinate of a location in the user-selected World Coordinate System.
National Weather Service (NWS)	NWS is a Federal agency that makes and issues meteorologic and hydrologic forecasts. NWS forecasts may be used as input for HEC-RTS simulations.
Off-channel storage area	In HEC-RAS, an off-channel storage area is a modeling element used to represent ponded water in interior areas, adjacent ponds and lakes, urban areas next to rivers, etc.
Parameter	A parameter is a representation of a watershed condition in a model of the watershed. You can set parameter values using HEC-RTS. Varying parameters may vary analysis results.
Precipitation grid	A precipitation grid is a spatial representation of precipitation over the whole watershed for an interval of time.
Project benefit	A project benefit is a reduction in adverse flood consequences attributable to a project or projects. Project benefit commonly is computed as the difference between the without- and with-project consequences.
Projects	Projects are man-made structures such as reservoirs or levees that are built to control water.
Quality color bar	The quality color bar is a thick, segmented line displayed in the HEC-RTS map window that represents the history of the data quality at a specific location within the time window you have established.
Quantitative precipitation forecast (QPF)	QPF is a spatial and temporal precipitation forecast for a specified region or area.[7]
Raw data	Raw data are data that are acquired, transmitted, and received, but not yet subjected to quality tests and revised. They are preserved in an HEC-RTS DSS file and are identified as raw data.

Regulated flow	Regulated flow is flow that has been changed from its natural state due to the influence of hydraulic constrictions or manmade influences.
Reservoir	A reservoir is a structure built to control water for uses such as flood control, hydropower generation, and water supply. Storing water in a reservoir can reduce flood consequences by regulating flow to reduce flood stages.
Reservoir System Simulation (HEC-ResSim)	HEC-ResSim is an HEC analysis software application that simulates behavior of reservoirs and linking channels, following user-specified operations for reservoir release decision making. Inputs include flows into reservoirs and unregulated flows downstream of reservoirs. Outputs include reservoir releases, downstream regulated flows, and reservoir storage conditions.
Revised data	Revised data are data that are validated and transformed for use in modeling. They are saved to the CWMS database and are identified as potentially revised data.
River Analysis System (HEC-RAS)	HEC-RAS is an HEC analysis software application that simulates, in one-dimension, behavior of channels and adjacent floodplains. This permits determination of water surface elevations corresponding to flows computed by HEC-HMS or HEC-ResSim. Inputs include flows, and outputs include water surface elevations.
River Analysis System Mapper (RAS Mapper)	RAS Mapper is tool that is included in the HEC-RAS software application that allows you to use and visualize floodplain geospatial information along with HEC-RAS analysis results.
Routing reach	A routing reach is a portion of a stream network. In HEC-HMS and HEC-ResSim, a routing reach is a modeling element used to compute behavior of flow in a channel.
Scripting	Scripting is the process of automating the actions of a program, so a prescribed sequence of actions can be executed with a single action.
Setup module	The Setup module in HEC-RTS contains the watershed setup commands. These are commands for configuring inputs, models, and outputs that describe a watershed's behavior.
Simulation	A simulation is a mathematical representation of the behavior of a system, given inputs and initial conditions.
Stage	Stage is the vertical distance of a water surface above or below a local or national datum.

Stream alignment	A stream alignment is a representation of the stream network in the watershed. The alignment shows routing reaches and indicates where confluences and bifurcations occur and provides a sense of distance and scale.
Stream element	A stream element is part of the representation of the stream network in the watershed. Specifically, a stream element is a segment of the stream alignment that is composed of vertex points (points with known coordinates). The beginning and ending vertex points of a stream element are stream nodes. Stream elements typically begin or end at a confluence, bifurcation, or at the boundary of the river system.
Stream junction	Stream junctions are part of the representation of the stream network in the watershed. Specifically, a stream junction is a point where two stream elements meet (two stream nodes at one location).
Stream node	Stream nodes are part of the representation of the stream network in the watershed. Specifically, stream nodes are the beginning and ending points for each stream element. Stream nodes are points where stream elements meet (stream junctions).
Stream stationing	Stream stationing is a reference system for locating features with distance along a stream. HEC-RTS creates and labels stream stations (or river stations) along each stream element. The station at the downstream end of an element is Station 0 (zero), and the stationing increases upstream along the element according to the linear distance between vertices on the stream element in the watershed coordinate system.
Subbasin	A division of a watershed typically delineated based on the terrain within the watershed. The entire precipitation within a subbasin drains to a common point.
Supplemen tal program	A supplemental program is a program or script that is not included with HEC-RTS. The program or script is developed by the user and may be added to the program order so it will be executed from HEC-RTS.
Threshold color bar	The threshold color bar is a thick, segmented line shown in an HEC-RTS map window that represents a linear histogram that compares current data against threshold values set for a location.
Time series	A time series is an ordered sequence of values of a property or state at equally spaced time intervals. DSS files also support time series composed of irregularly spaced increments.
Time series icon	A time series icon is a symbol in the display area that provides quick access to time-series data assigned to the icon. The location of the time series icon on the watershed map corresponds to the location to which the data pertains. Time series icons can be used to view data, images, web pages, scripts, and document files.

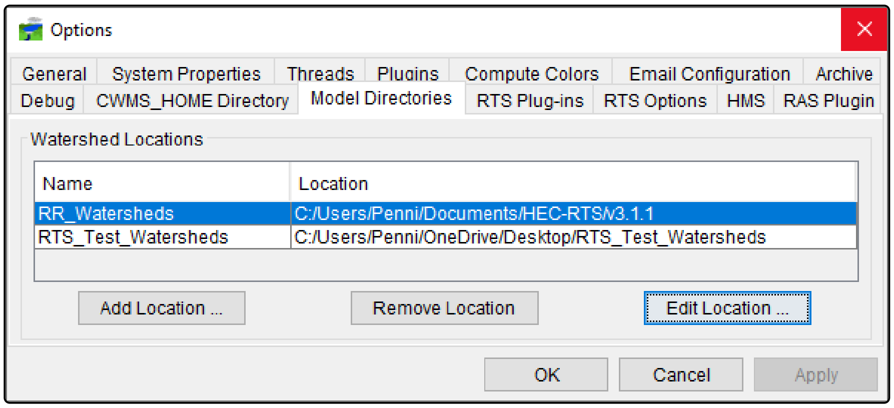
Time window	A time window is the duration that will be simulated. A time window is defined by starting and ending dates and times.
Transform method	The transform method is a model for calculating surface runoff from precipitation on a subbasin.
U.S. Army Corps of Engineers (USACE)	USACE is a Federal agency that operates about 700 water control projects in the U.S. as part of its water control management mission. These projects include reservoirs, navigation locks and dams, and levee and bypass systems with closure and diversion structures.
United States Geological Survey (USGS)	USGS is a Federal agency that collects hydrologic data with a network of gages and reports the data in real-time.
Validation list	A validation list is a way to organize data in HEC-RTS DSS files for editing or validation.
Vertex point	A vertex point is a point with known coordinates on a stream element in the stream alignment.
Visualization module	The Visualization module in HEC-RTS contains commands for data visualization. These are commands for displaying observed and forecast data to evaluate the hydrometeorological state of the watershed.
Watershed	In HEC-RTS, a watershed is a set of data, models, and images that represent watershed lands and the channels, gages, and water control features within the watershed.
Watershed pane	The watershed pane in the HEC-RTS refers to the set of Visualization, Acquisition, Modeling, and Setup module tabs (tabbed menus). Clicking a module tab opens that module in HEC-RTS.
World Coordinate System (WCS)	To maintain a geographic reference (also called a geo-reference), HEC-RTS uses a user-selected and customizable coordinate system, called the World Coordinate System. This superimposes a grid on layer features to establish x and y coordinates in WCS for each point on the layer. The x coordinate is referred to as "easting", and the y coordinate is referred to as "northing". Possible choices for the WCS include but are not limited to State Plane Coordinates and Universal Transverse Mercator (UTM) coordinates.
Zone	A set of subbasins in the watershed.

Definition Sources:

1. California Department of Water Resources. (2012). Urban Levee Design Criteria. Accessed December 20, 2013.
   http://www.water.ca.gov/floodsafe/leveedesign/ULDC_May2012.pdf
   ESRI. ArcGIS® Help Version10.2. Accessed December 20, 2013.
2. http://resources.arcgis.com/en/help/main/10.2/index.html#//003r0000000/q000000
   ESRI Press. (2006). A to Z GIS. Redlands, CA.
3. Federal Highway Administration. (1985). Hydraulic Design of Highway Culverts, Chapter 1, HDS 5 Introduction. Accessed December 18, 2013.
   http://www.fhwa.dot.gov/engineering/hydraulics/pubs/hds5si.pdf
4. National Institute of Standards and Technology (NIST)/SEMATECH.
   e-Handbook of Statistical Methods, Section 6.4.1 Definitions, Applications and Techniques. Accessed December 16, 2013.
   http://www.itl.nist.gov/div898/handbook/pmc/section4/pmc41.htm
5. National Oceanic and Atmospheric Administration. "NOAA and Inundation". Accessed December 18, 2013.
   http://inundation.noaa.gov/
6. National Weather Service. NOAA's National Weather Service: Glossary. Accessed December 20, 2013.
   http://forecast.weather.gov/glossary.php?word=qpf
7. Oracle. Java SE Documentation. Accessed December 20, 2013.
   http://docs.oracle.com/javase/6/docs/technotes/guides/jar/index.html
8. U.S. Naval Observatory. "Universal Time". Accessed December 18, 2013.
   http://aa.usno.navy.mil/faq/docs/UT.php

25 HEC-RTS Application Settings

You can configure several HEC-RTS application settings through the **Options** dialog (Figure 1). You can set the following: the directory where your HEC-RTS studies are stored; list of plug-ins related to the software applications (i.e., MFP, HEC-FIA) that are part of an HEC-RTS study; reopen last forecast; set the forecast directory structures; add the *maps* folder to the **Watershed Tree**; allow multiple program orders; run debug for HEC-HMS and HEC-RAS; set several general options for HEC-RTS; create/edit/delete HEC-RTS system properties; review the active HEC-RTS threads; review the list of enabled HEC-RTS plug-ins; adjust the default colors for HEC-RTS computes; setup a scheme for archiving; and, adjust the debug level for DSS. This appendix provides details on the available settings that can be adjusted.



128 Figure 1 Options Dialog

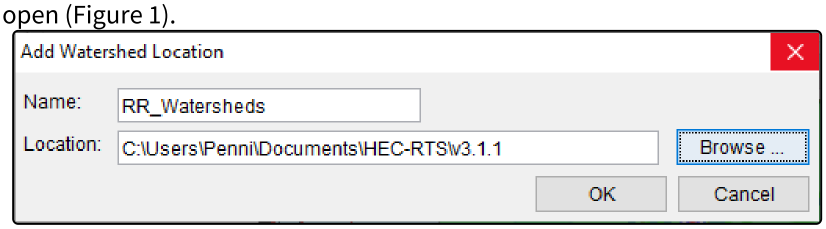
25.1 Model Directories Tab

From the **Model Directories** tab (Options Dialog), you need to tell HEC-RTS where to look for watersheds. From the **Model Directories** tab, you can create locations, delete (remove), and edit locations. You will need to create at least on watershed location, the defined watershed locations display in the **Open Watershed** dialog (Opening an Existing Watershed).

25.1.1 Add a Watershed Location

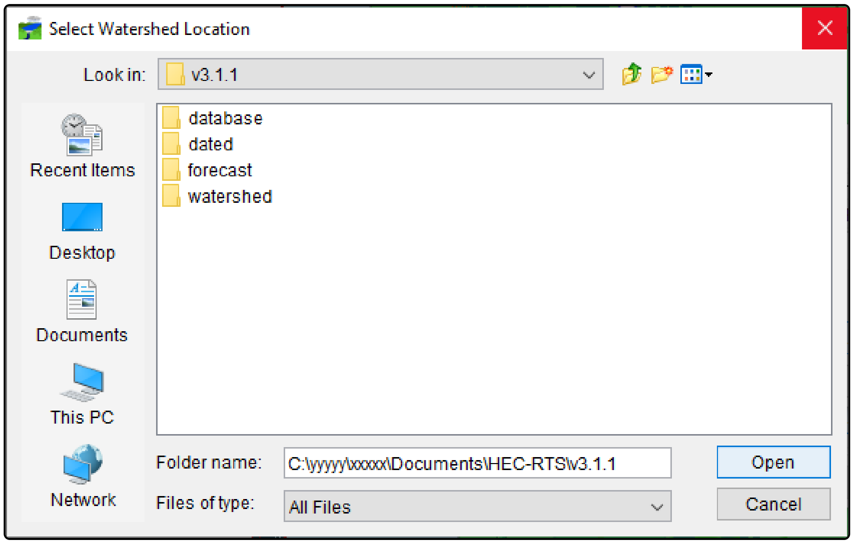
To create a watershed location:

1. From the HEC-RTS main window, from the **Tools** menu, click **Options,** the Options Dialog opens.
2. Click the **Model Directories** tab (Options Dialog), click **Add Location**. The **Add Watershed Location** dialog will open (Figure 1).

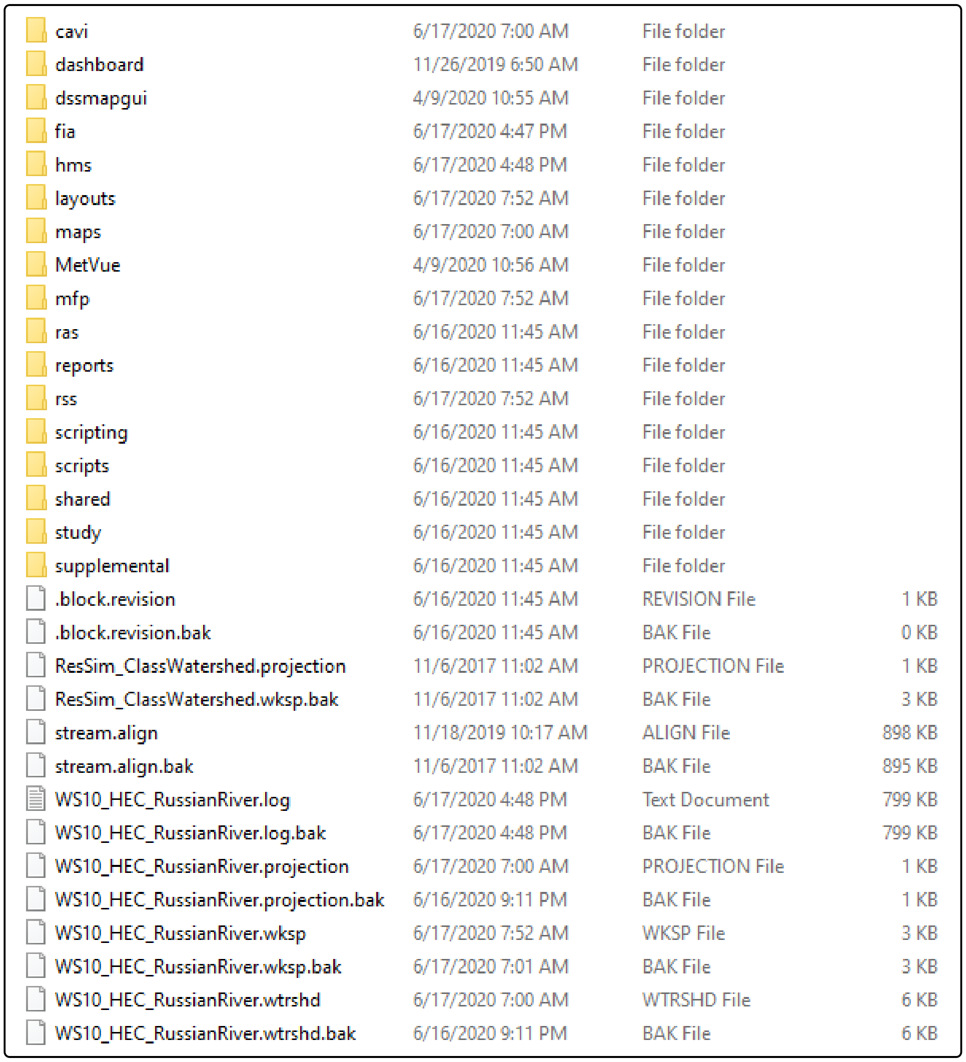


3. Enter a name for the watershed location in the **Name** box (Figure 1). You can either enter the location of the directory in the **Location** box (Figure 1) or click **Browse**. A **Select Watershed Location** browser will open (Figure 2). Navigate to the directory on your computer or network where you want a watershed stored. Click **Open**, the

Select Watershed Location browser closes.



4. The selected directory displays in the **Location** box (Figure 1). Click **OK**, if the directory does not exist, a **Warning** window will open asking you if you want to create the directory, click **Yes**. Both the **Warning** window and the **Add Watershed Location** dialog (Figure 1) will close. From the **Watershed Locations** table (Options Dialog) the name and location of the watershed location is displayed.
5. In the location folder, *C:\yyyyy\xxxxx\Documents\HEC-RTS\v3.1.1*, three folders are created - ***database***, ***forecast***, and ***watershed***. When a new watershed is created (i.e., *WS10_HEC_RussianRiver*), and a watershed location is selected - *RR_Watersheds* - the new watershed (i.e., *WS10_HEC_RussianRiver*) will be stored under the ***watershed*** directory.
6. When you create a new watershed, HEC-RTS will generate a directory structure, and stores all files associated with the watershed inside that structure. The watershed directory is named according to the name you have given the watershed; for example, in Figure 3, the watershed directory is *WS10_HEC_RussianRiver*. Figure 3 shows schematically how HEC-RTS organizes the directories and files it creates in the watershed's directory ({_} C:\yyyyy\xxxxx\Documents\HEC-RTS\v3.1.1\watershed_*RR_NorthFork*). The watershed's directory configuration files and data for the watershed, including maps and model alternative data will be stored.



cavi	6/17/2020 7:00 AM	File folder	
dashboard	11/26/2019 6:50 AM	File folder	
dssmapgui	4/9/2020 10:55 AM	File folder	
fia	6/17/2020 4:47 PM	File folder	
hms	6/17/2020 4:48 PM	File folder	
layouts	6/17/2020 7:52 AM	File folder	
maps	6/17/2020 7:00 AM	File folder	
MetVue	4/9/2020 10:56 AM	File folder	
mfp	6/17/2020 7:52 AM	File folder	
ras	6/16/2020 11:45 AM	File folder	
reports	6/16/2020 11:45 AM	File folder	
rss	6/17/2020 7:52 AM	File folder	
scripting	6/16/2020 11:45 AM	File folder	
scripts	6/16/2020 11:45 AM	File folder	
shared	6/16/2020 11:45 AM	File folder	
study	6/16/2020 11:45 AM	File folder	
supplemental	6/16/2020 11:45 AM	File folder	
.block.revision	6/16/2020 11:45 AM	REVISION File	1 KB
.block.revision.bak	6/16/2020 11:45 AM	BAK File	0 KB
ResSim_ClassWatershed.projection	11/6/2017 11:02 AM	PROJECTION File	1 KB
ResSim_ClassWatershed.wksp.bak	11/6/2017 11:02 AM	BAK File	3 KB
stream.align	11/18/2019 10:17 AM	ALIGN File	898 KB
stream.align.bak	11/6/2017 11:02 AM	BAK File	895 KB
WS10_HEC_RussianRiver.log	6/17/2020 4:48 PM	Text Document	799 KB
WS10_HEC_RussianRiver.log.bak	6/17/2020 4:48 PM	BAK File	799 KB
WS10_HEC_RussianRiver.projection	6/17/2020 7:00 AM	PROJECTION File	1 KB
WS10_HEC_RussianRiver.projection.bak	6/16/2020 9:11 PM	BAK File	1 KB
WS10_HEC_RussianRiver.wksp	6/17/2020 7:52 AM	WKSP File	3 KB
WS10_HEC_RussianRiver.wksp.bak	6/17/2020 7:01 AM	BAK File	3 KB
WS10_HEC_RussianRiver.wtrshd	6/17/2020 7:00 AM	WTRSHD File	6 KB
WS10_HEC_RussianRiver.wtrshd.bak	6/16/2020 9:11 PM	BAK File	6 KB

129 Figure 3 HEC-RTS Watershed Directory Structure - C:\yyyyy\xxxxx\Documents\HEC-RTS\v3.1.1\watershed\WS10_HEC_RussianRiver

7. Furthermore, when you create a new forecast for a watershed, you will give that forecast a name. Under the ***forecast*** directory ({_}C:\yyyyy\xxxxx\Documents\HEC-RTS\v3.1.1_*forecast*) for the watershed's location, HEC-RTS will create a directory based on the dated forecast name. For example, the directory name ***2017.01.26-0400*** corresponds to a forecast created for *26 January 2017*. Under that directory ({_}C:\yyyyy\xxxxx\Documents_*HEC-RTS\v3.1.1\forecast*{*}*2017.01.26-0400*), HEC-RTS will create a directory based on the watershed's name (*WS10_HEC_RussianRiver*) and will automatically copy all of the directories and files for your watershed (except for the *maps* directory) into the dated forecast directory (*C:\yyyyy\xxxxx\Documents\HEC-RTS\v3.1.1\forecast*{*}* *2017.01.26-0400_ _WS10_HEC_RussianRiver*). This facilitates archiving of forecast information and ensures consistency in your model results.

Delete (Remove) a Watershed Location

To delete (remove) a watershed location:

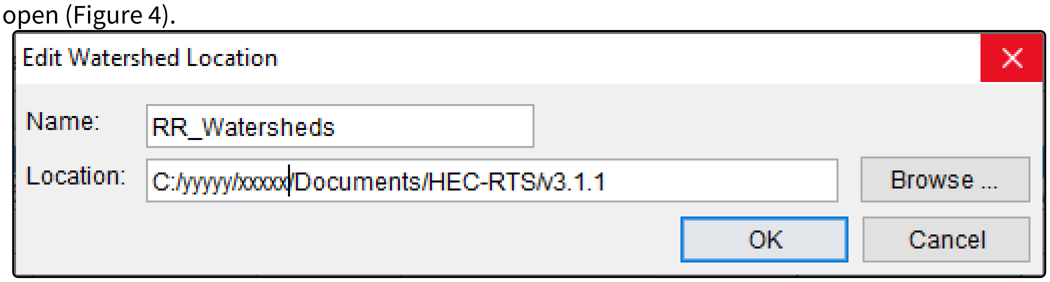
1. From the HEC-RTS main window, from the **Tools** menu, click **Options,** the Options Dialog opens.

2. Click the **Model Directories** tab (Options Dialog), from the **Watershed Locations** table (Options Dialog), select a location. Click **Remove Location** (Options Dialog), a **Confirm Removal** window will open, asking you to confirm the removal of the selected location. Click **Yes**, the **Confirm Removal** window will close, and the selected location will no longer display in the **Watershed Locations** table (Options Dialog).
3. The directory structure for the selected watershed location remains on the hard drive, nothing is deleted.

25.1.2 Edit a Watershed Location

To edit a watershed location:

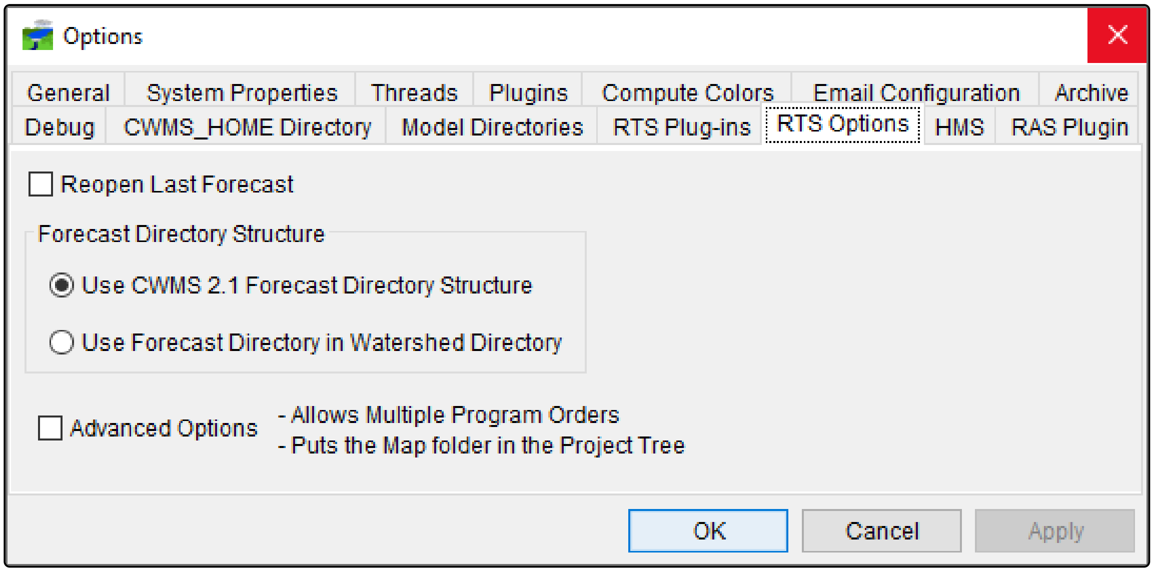
1. From the HEC-RTS main window, from the **Tools** menu, click **Options,** the Options Dialog opens.
2. Click the **Model Directories** tab (Options Dialog), click **Edit Location**. The **Edit Watershed Location** dialog will open (Figure 4).



3. You can change the name of the watershed location in the **Name** box (Figure 4). You can either change the location of the directory in the **Location** box (Figure 4) or click **Browse**. A **Select Watershed Location** browser will open (Figure 2). Navigate to the directory on your computer or network where you want a watershed stored. Click **Open**, the **Select Watershed Location** browser closes.
4. The selected directory displays in the **Location** box (Figure 4). Click **OK**, if the directory does not exist, a **Warning** window will open asking you if you want to create the directory, click **Yes**. Both the Warning window and the **Edit Watershed Location** dialog (Figure 4) will close. From the **Watershed Locations** table (Options Dialog) the edits will be displayed.

25.2 RTS Options Tab

From the **Options** dialog, from the **RTS Options** tab (Figure 1), you can have the last forecast you had open in HEC-RTS open automatically the next time; set the structure of the forecast directories; display the ***Maps*** folder in the **Watershed Tree**; and, allow multiple programs order.



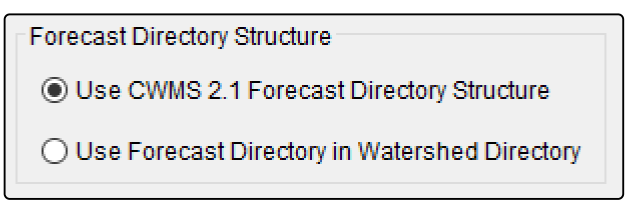
130 Figure 1 Options Dialog - RTS Options Tab

25.2.1 Reopen Last Forecast

To reopen the last forecast, from the HEC-RTS main window, from the **Tools** menu, click **Options,** the **Options** dialog opens. Click the **RTS Options** tab (Figure 1), the available HEC-RTS settings are displayed. To have HEC-RTS automatically open the last forecast that you had open during a previous HEC-RTS session, select **Reopen Last Forecast**. Click **OK**, the **Options** dialog will close (Figure 1), and the next time you open HEC-RTS, the forecast you have currently open, will be automatically opened by HEC-RTS.

25.2.2 Forecast Directory Structure

By default, the forecast directory structure for a watershed in HEC-RTS, is the forecast directory structure that was developed in HEC-RTS Version 1.2. If the **Use CWMS 2.1 Forecast Directory Structure** option is selected (Figure 2), Team Modeling will not work.



131 Figure 2 Options Dialog - RTS Options Tab - Forecast Directory Structure Box

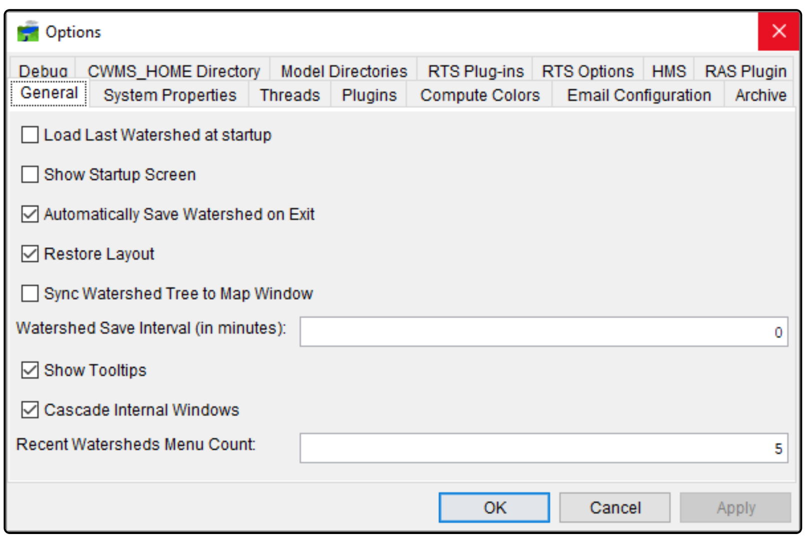
25.2.3 Advanced Options

From the **Options** dialog (Figure 1), if **Advanced Options** is selected, two things will occur when you restart HEC-RTS. From the **Watershed Tree**, the **Maps** folder will now display (Figure 3).

In addition, by default HEC-RTS only allows one **Program Order** (Multiple Program Orders). By selecting **Advanced Options**, you now have the capability to create multiple program orders (further details are provided in Multiple Program Orders). Care must be exercised when using multiple programs orders. When a model alternative is linked to a program for its input information, it remembers the position of the model it is receiving its data from, not the name or program of the specified model alternative. This can lead to potential conflicts if a model alternative is used in two different program orders. The recommended practice when using multiple programs orders is to configure a different model alternative for use in each different program order.

25.3 General Tab

From the **Options** dialog, from the **General** tab (Figure 1), the user can set the last study to be loaded during the startup of HEC-RTS; display a dialog box that allows the user to open or create a study; save the study automatically when exiting HEC-RTS; restore the default layout; synchronize the **Study Tree** to the active **Map Window**; set an interval (minutes) for automatically saving a study; display tooltips; set default scheme for cascading windows; and, specify the number of HEC-RTS watersheds to display in the **Recent Studies** item under the **File** menu.

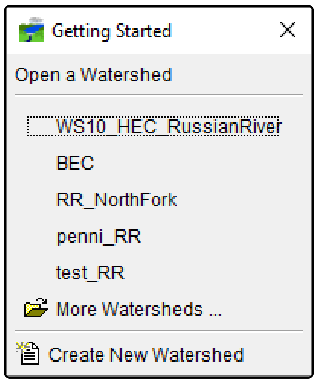


25.3.1 Load Last Watershed at Startup

HEC-RTS by default does not automatically load the last watershed that has been opened. If you want the last watershed you opened to automatically load when HEC-RTS startups, from the **Options** dialog, from the **General** tab (Figure 1), click **Load Last Watershed at startup**. The next time you re-start HEC-RTS, the last watershed that was open will automatically load.

25.3.2 Show Startup Screen

HEC-RTS by default does not automatically display the **Getting Started** dialog (Figure 2) on startup. The **Getting Started** dialog, provides you with a list of the last five HEC-RTS watersheds that were open; more watersheds can be accessed from the **Open Watershed** dialog (Opening an Existing Watershed); and, create new watersheds (Creating a Watershed).



132 Figure 2 Getting Started Dialog

To open a recent watershed, click on a watershed name, the **Getting Started** dialog will close, and the watershed will display in the HEC-RTS main window. To access watersheds not listed, click **More Watersheds** (Figure 2), the **Open Watershed** dialog will display (Opening an Existing Watershed). Select a watershed, the **Open Watershed** dialog will close, the watershed will display in the HEC-RTS main window, and the **Getting Started** dialog will close.

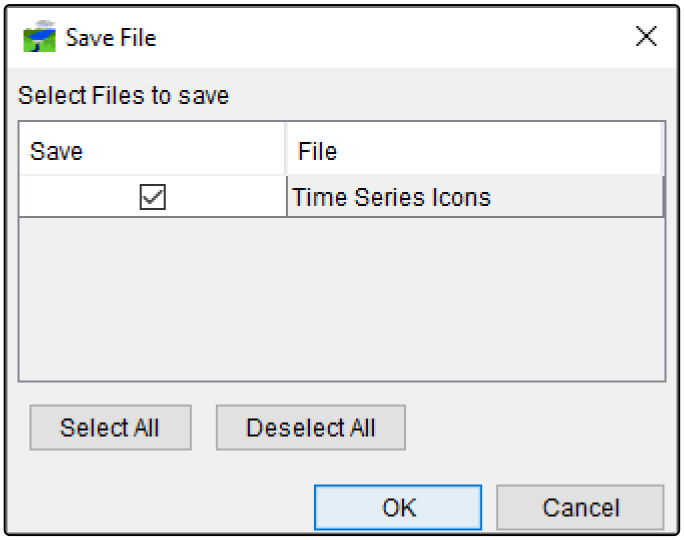
From the **Getting Started** dialog, you can create a new watershed, click **Create New Watershed** (Figure 2), the **Create New Watershed** dialog will open (details for creating a watershed are detailed in Creating a Watershed). Click **OK**, the **Create New Watershed** dialog will close, the new watershed will display in the HEC-RTS main window, and the **Getting Started** dialog will close.

If you want the **Getting Started** dialog (Figure 2) displayed when starting HEC-RTS, from the **Options** dialog, from the **General** tab (Figure 1), click **Show Startup Screen**. The next time you re-start HEC-RTS, the **Getting Started** dialog will display.

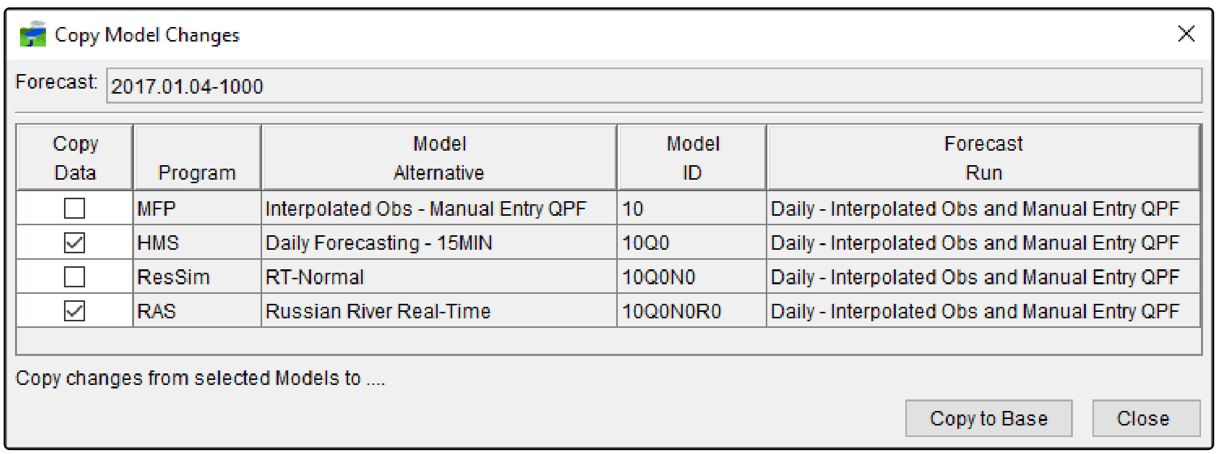
25.3.3 Automatically Save Watershed on Exit

HEC-RTS by default automatically saves the open watershed on exit. If you do not want HEC-RTS to save the current watershed on exit, from the **Options** dialog, from the **General** tab (Figure 1), un-select **Automatically Save Watershed on Exit**.

For example, when making changes to the watershed, you change the name of a time series icon, and you make change to two of the model alternatives. When closing this watershed, the **Save File** dialog will open (Figure 3) letting you know that there is a change to the time series icons layer - do you want to save that change. The **Copy Model Changes** dialog will open (Figure 4) because of the changes you made to model alternatives that are part of the watershed – do you want to save those changes.



133 Figure 3 Save File Dialog



134 Figure 4 Copy Model Changes Dialog

25.3.4 Restore Layout

The capability in HEC-RTS to restore layouts is on by default, which means that information about the current layout of the map windows is automatically saved when exiting HEC-RTS. Layouts can be saved, managed, and restored. From the **Options** dialog, from the **General** tab (Figure 1), if you do not want the capability to restore layouts, un-select **Restore Layout**.

25.3.5 Sync Watershed Tree to Map Window

By default, in HEC-RTS, the watershed tree of the current watershed is synchronized with the **Map Window** being currently displayed in the HEC-RTS main window. From the **Options** dialog, from the **General** tab (Figure 1), if do not want the study tree to be synchronized with the active **Map Window**, un-select **Sync Study Tree to Map Window**.

25.3.6 Watershed Save Interval

HEC-RTS by default automatically sets the interval for a watershed save to zero, which means that a watershed is not automatically being saved while the watershed is open. From the **Options** dialog, from the **General** tab (Figure 1), if you want to automatically save watersheds, from the **Watershed Save Interval** box, enter a time in minutes.

25.3.7 Show Tooltips

HEC-RTS by default automatically displays tooltips in the HEC-RTS interface. If you do not want to view tooltips, from the **Options** dialog, from the **General** tab (Figure 1), un-select **Show Tooltips**. Now as you hover over items in the HEC-RTS interface you will no longer see tooltips.

25.3.8 Cascade Internal Windows

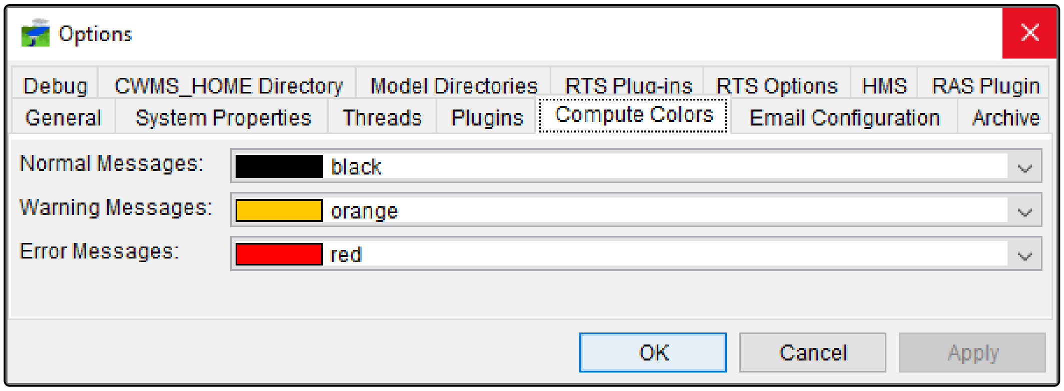
HEC-RTS by default the capability to cascade map windows is set. If you do not want map windows to cascade, from the **Options** dialog, from the **General** tab (Figure 1), un-select **Cascade Internal Windows**.

25.3.9 Recent Watersheds Menu Count

HEC-RTS by default automatically displays the last five watersheds that have recently been opened. If you want to view more or less watersheds, from the **Options** dialog, from the **General** tab (Figure 1), from the **Recent Watersheds Menu Count** box (Figure 1), enter a new value. The value must be 1 from 10. Any value entered that is less than the value in the **Recent Watersheds Menu Count** box (Figure 1), purges the list of recent watersheds. An upper limit of 10 is a recommendation, you can enter a large value, but be aware that the longer the list, how that might impact **Recent Watersheds** list.

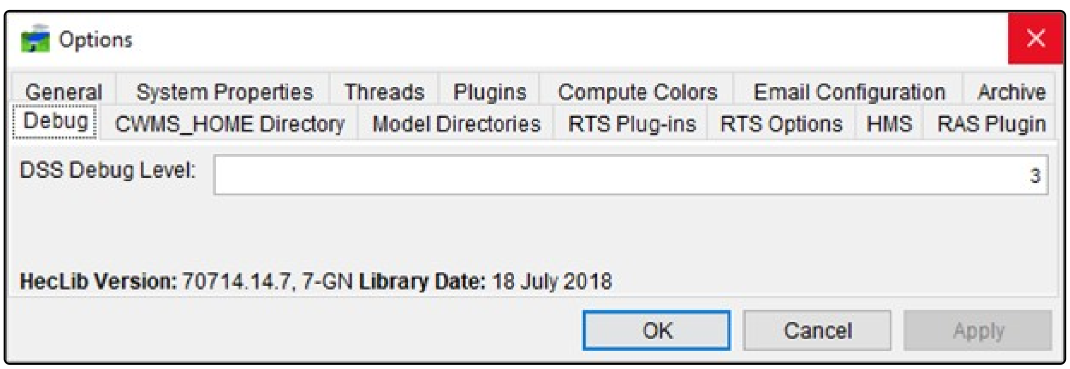
25.4 Compute Colors Tab

When you compute a forecast in the Modeling module, the **Compute Progress** dialog reports on the progress of the forecast and presents warning and error messages generated by the forecast compute. From the **Options** dialog, click the **Compute Colors** tab (Figure 1). You can set colors for **Normal Messages**, **Warning Messages**, and **Error Messages**. There are thirty-two color choices in each of the lists. Change the color of messages by selecting a color from the lists. For any of your changes to take effect, you must re-compute a forecast.



25.5 Debug Tab

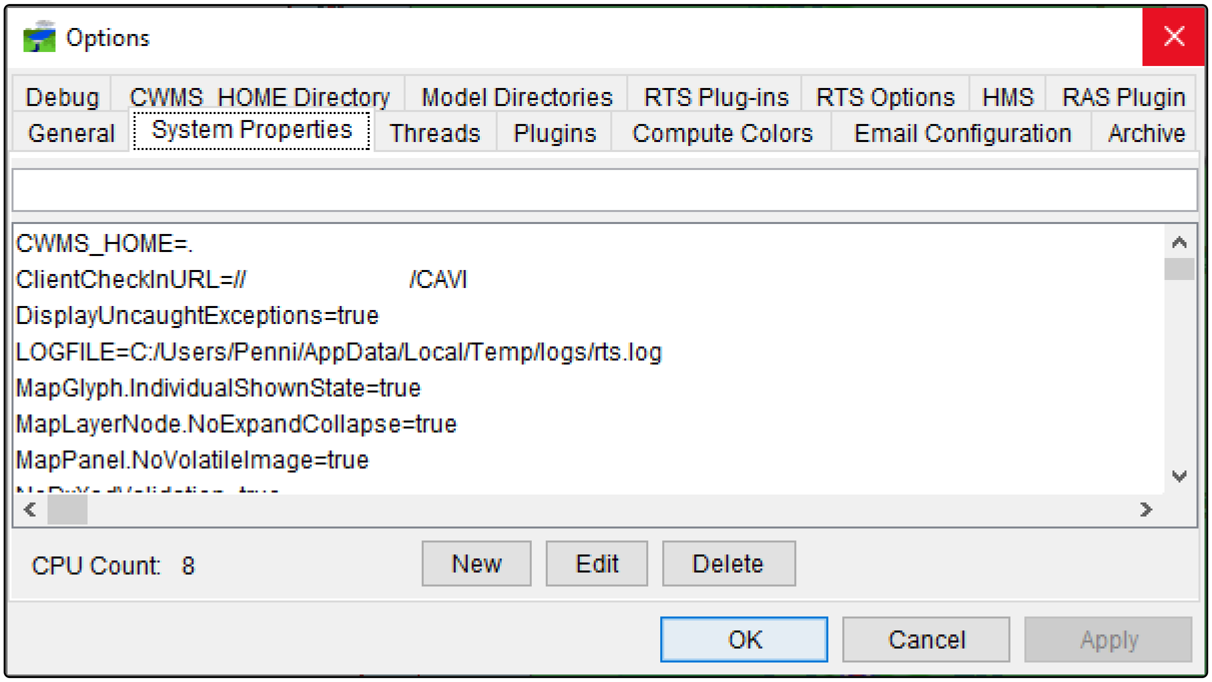
HEC-RTS by default automatically sets the debug level for the DSS log file at three (3). From the **Options** dialog, click the **Debug** tab (Figure 1), pertain to settings that deal with the HEC-RTS software. From some of these tabs you can make changes, it is recommended that you only use this information from these tabs for debugging and viewing purposes. If you make changes to system properties, any changes made are not persistent and are not saved permanently.



135 Figure 1 Options Dialog - Debug Tab

25.6 HEC-RTS Properties

The **System Properties**, **Threads**, and **CWMS_HOME Directory** tabs from the **Options** dialog (Figure 1), pertain to settings that deal with the HEC-RTS software. From some of these tabs you can make changes, it is recommended that you only use this information from these tabs for debugging and viewing purposes. If you make changes to system properties, any changes made are not persistent and are not saved permanently.



136 Figure 1 Options Dialog - System Properties Tab

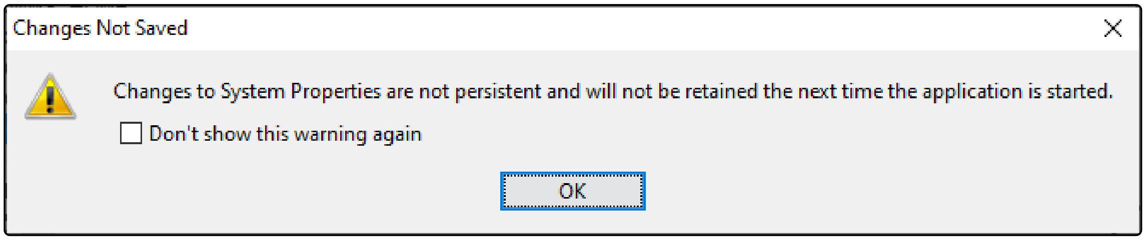
25.6.1 System Properties Tab

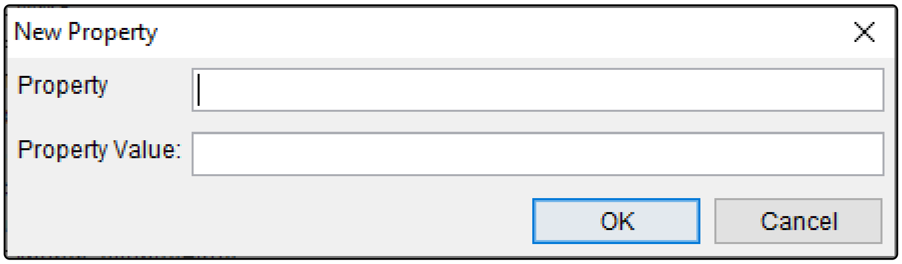
The **System Properties** tab (Figure 1) allows you to view the system properties values set for the HEC-RTS software. If you have the correct user access, you can edit some or all properties. The recommendation is that you only use this information for debugging and viewing purposes. Editing must be done with extreme caution as any changes you make can directly affect the HEC-RTS functionality.

25.6.2 Add a System Property

To add a system property:

1. If you have the correct user access levels, the **New** command will be available, and you can add system properties. Click **New**, a **Changes Not Saved** window will display (Figure 2), letting you know that any changes made are not persistent and are not saved permanently. Click **OK**, the **Changes Not Saved** window will close, and the **New Property** dialog (Figure 3) will open.

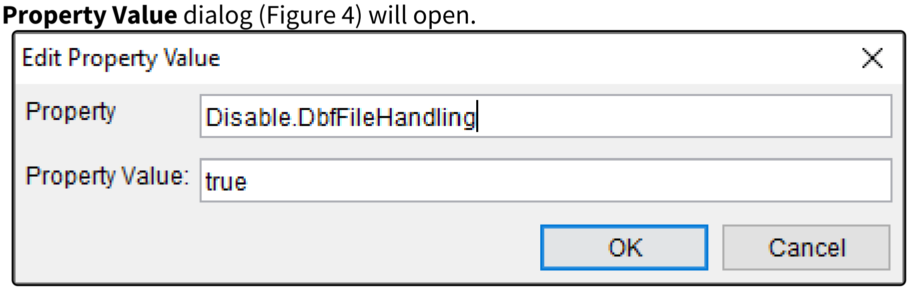




2. In the **Property** cell, enter the name of the new system property. For the parameters of the new system property, enter the parameters in the **Property Value** cell.
3. Click **OK**, and the **New Property** dialog closes. The new system property will display on the **System Properties Tab** (Figure 1).

25.6.3 Edit a System Property

1. If you have the correct user access levels then the **Edit** command will be available, and you can edit system properties. From the list, select a system property to edit.
2. Click **Edit**, a **Changes Not Saved** window will display (Figure 2), letting you know that any changes made are not persistent and are not saved permanently. Click **OK**, the **Changes Not Saved** window will close, and the **Edit Property Value** dialog (Figure 4) will open.

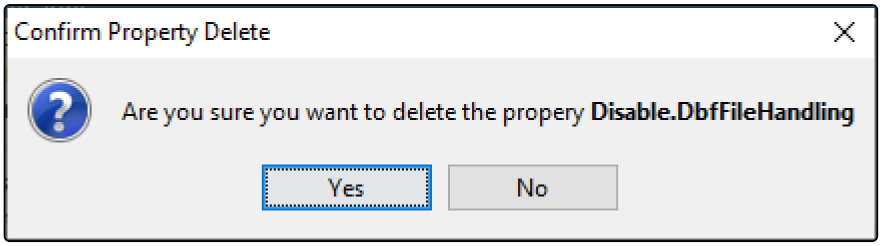


3. In the **Property** and **Property Value** boxes, edit the items, as you need to.
4. Click **OK**, the **Edit Property Value** dialog closes, and the changed system property will appear on the **System Properties Tab** (Figure 1).

25.6.4 Delete a System Property

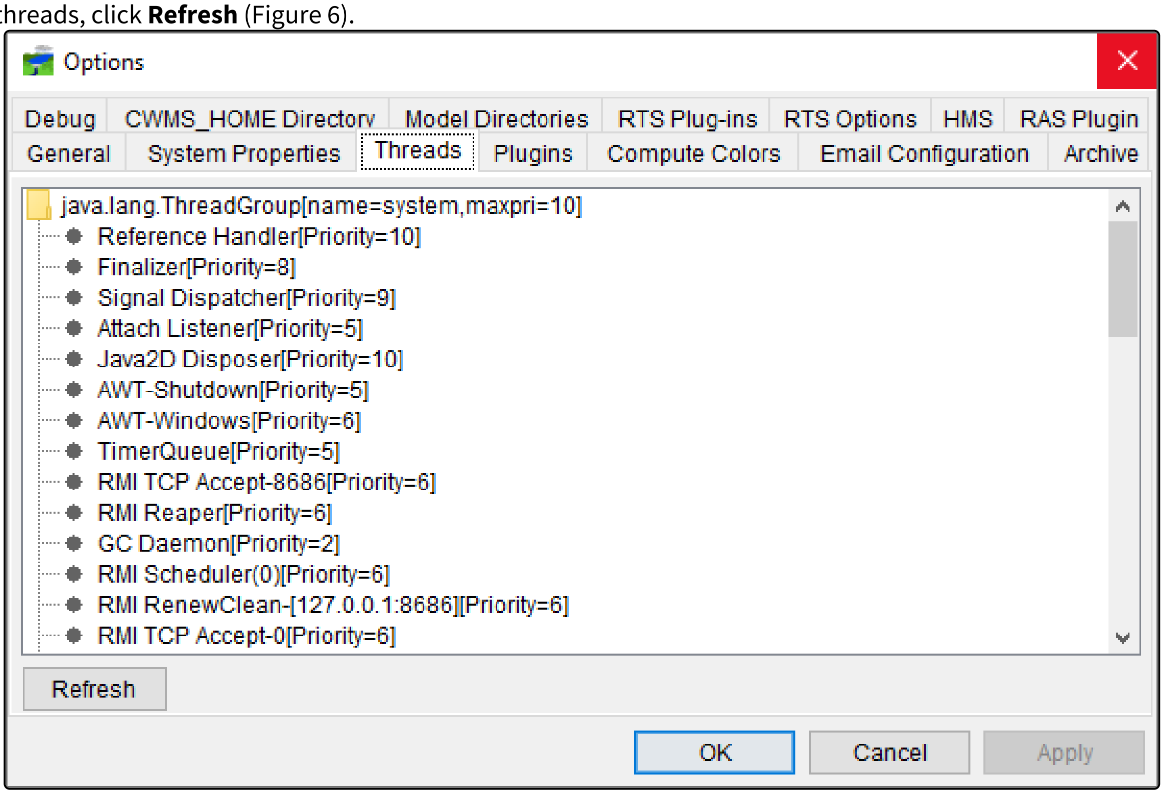
1. If you have the correct user access levels, then the **Delete** command will be available, and you can delete system properties**.**
2. From the list, select the system property you wish to delete. To select, click on the system property, it will now be highlighted in blue.

3. Click **Delete**, a **Confirm Property Delete** message will open (Figure 5). If you want to delete the selected system property, click **Yes**. The **Confirm Property Delete** message will close and the selected system property will be removed from the list.



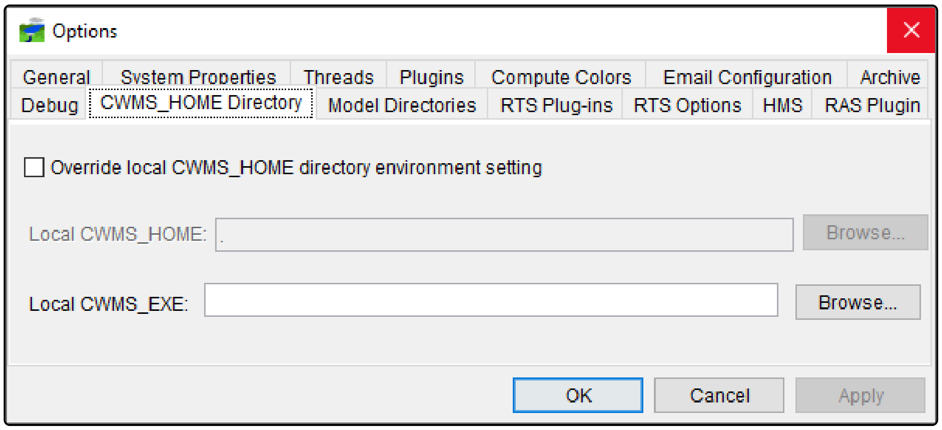
25.6.5 Threads Tab

A thread is a single sequential flow of control within software. Within Java the user can have multiple threads in a piece of software all running at the same time and performing different tasks. HEC-RTS is a multi-threaded software application, to view HEC-RTS threads, from the **Options**, click the **Threads** tab (Figure 6). The **Threads** tab displays information regarding the threads that are actively being used by HEC-RTS. To provide an up-to-date list of the active threads, click **Refresh** (Figure 6).



25.6.6 CWMS_Home Directory Tab

The **CWMS_HOME Directory** tab (Figure 7) pertains to two HEC-RTS system variables – CWMS_HOME and CWMS_EXE. In most cases, the normal user will not be doing anything to these two system variables. If you are dealing with client-side scripts, the tab provides an interface for overriding the values of CWMS_HOME and CWMS_EXE.



137 Figure 7 Options Dialog - CWMS_HOME Directory Tab

25.6.7 Local CWMS_HOME

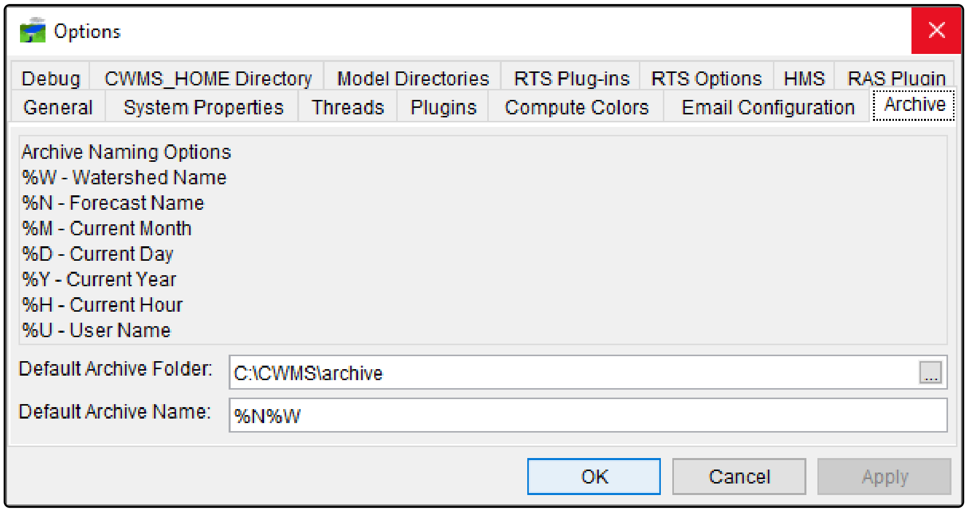
To edit the **CWMS_HOME** variable, select **Override local CWMS_HOME directory environment setting** (Figure B.21). The **Local CWMS_HOME** box is now active. **CWMS_HOME** only needs to be overridden if client-side scripts rely on **CWMS_HOME** as a pointer to where data and models reside on the client file system. For example, **CWMS_HOME** might be set to a "watershed location", following a common convention to keep gridded DSS records in the "database" folder that is sister to a folder containing the watersheds.

25.6.8 Local CWMS_EXE

You should only need to override **CWMS_EXE** if client-side scripts rely on **CWMS_EXE** to locate HEC-RTS executables or **_.jar_ files. If you need to override *CWMS_EXE**, from the **Local CWMS_EXE** box (Figure 7) enter the new value.

25.6.9 Archive Tab

You can archive a forecast; the archive is posted to the selected watershed's default directory as a **_.zip_ file. If you wish to override the default directory and or the name of the archive, from the *Options** dialog, click the **Archive** tab (Figure 8).



138 Figure 8 Options Dialog - Archive Tab

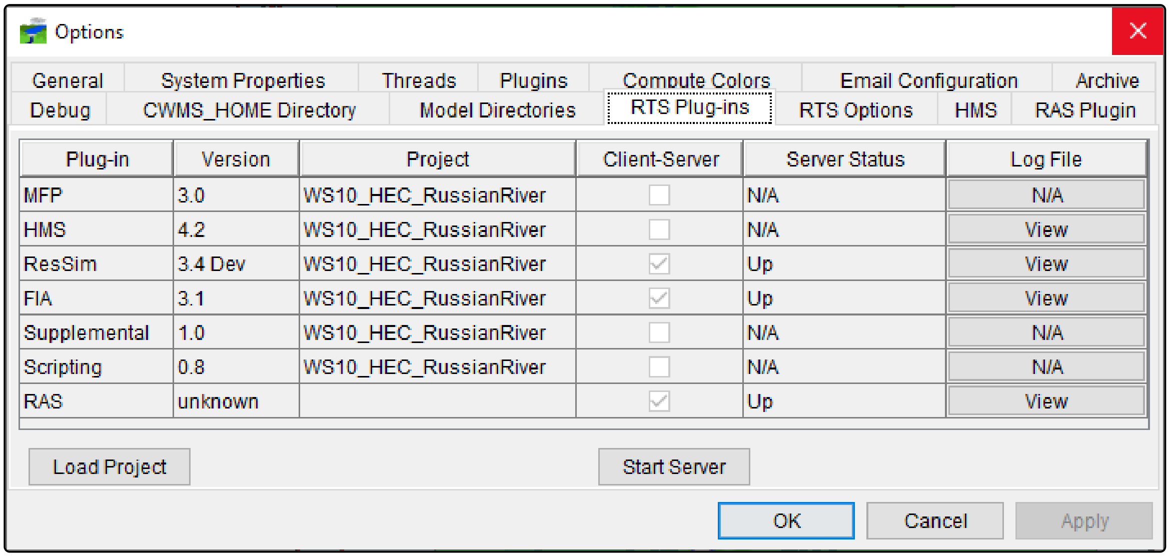
1. If you want to change the location of where the archive file is saved, you can enter a folder name in the **Default Archive Folder** box (Figure 8). another way to get a folder name, to the right of the **Default Archive Folder** box (Figure 8), click ... , an **Open** browser will open. Navigate to the location where you want archives saved, once the name displays, click **Open**. The **Open** browser will close, and name will display in the **Default Archive Folder** box (Figure 8).
2. To override the default archive file name, from the **Default Archive Name** box (Figure 8), enter the appropriate archive naming options which are listed on the **Archive** tab (Figure 8).

25.7 Plug-Ins

The **RTS Plug-Ins** tab provides information on the plug-ins that are currently being used in HEC-RTS for the four basic software applications in the default program order, plus the plug-ins for MFP, scripting, and supplemental software. The **RTS Plug-Ins** tab provides a list of the plug-ins available in HEC-RTS and from this tab you can enable and disable the list plugins.

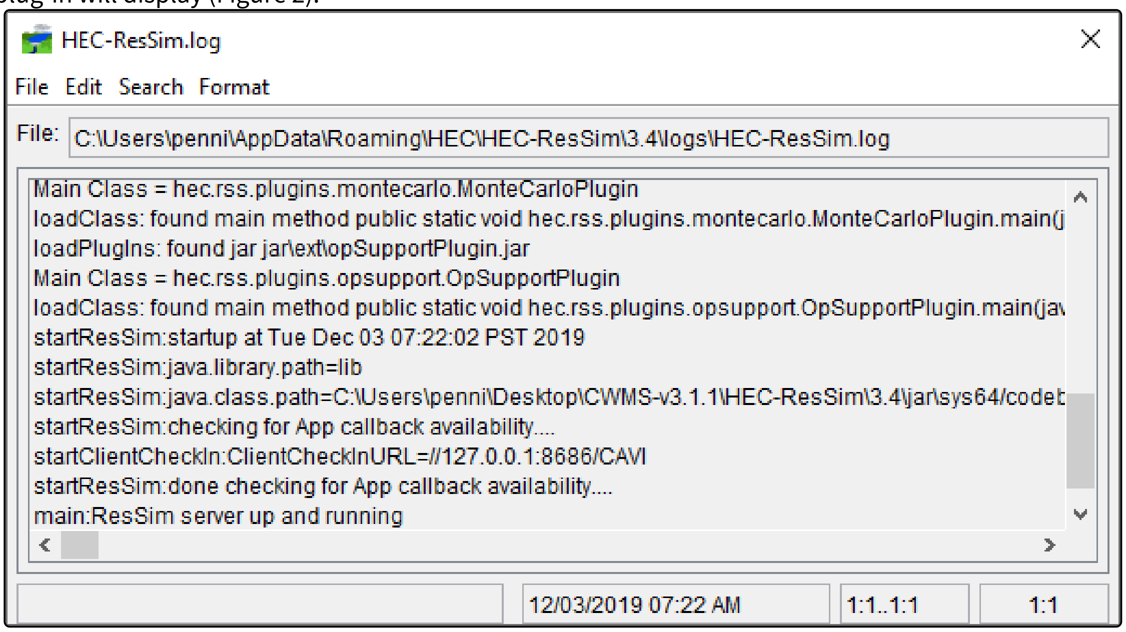
25.7.1 RTS Plug-Ins Tab

From the **Options** dialog, from the **RTS Plug-Ins** tab (Figure 1), you can view whether a software application is running; view a software application's log file; and, restart a software application if that application has stopped.



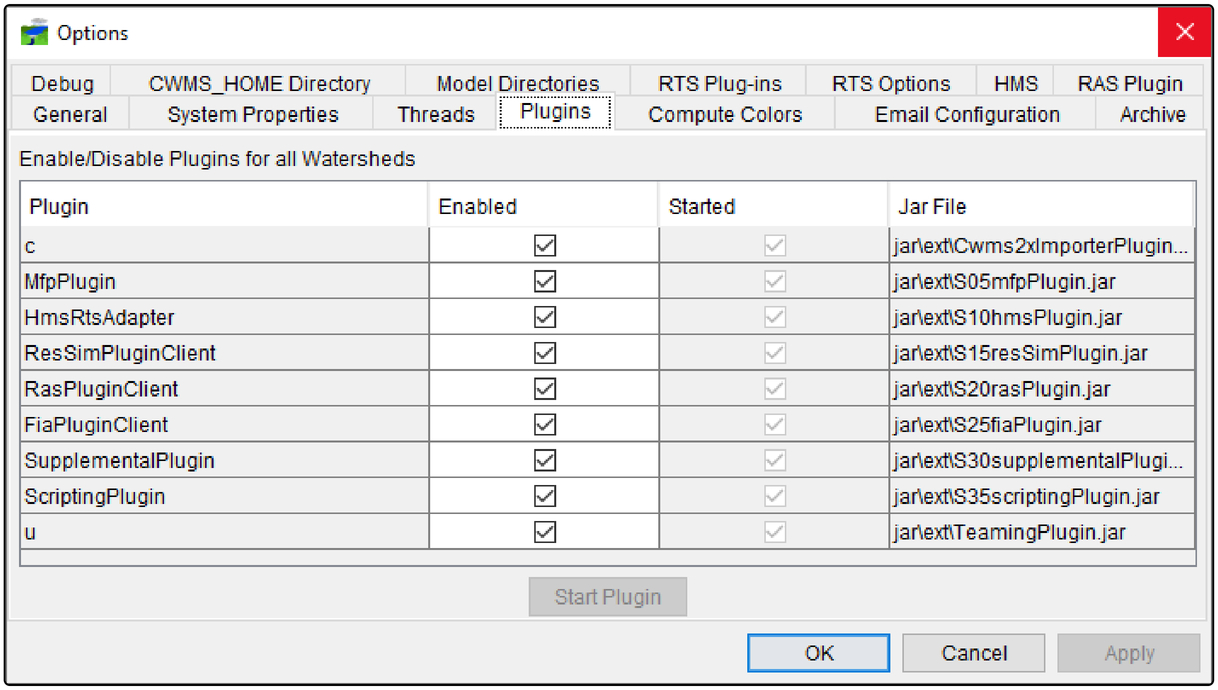
139 Figure 1 Options Dialog - RTS Plug-Ins Tab

1. From the **Options** dialog, click the **RTS Plug-Ins** tab (Figure 1), a table will display. The information contained in the table (Figure 1) is all about the available software applications, supplemental software, and scripting. Information includes the name of the software application; the version being used; the watershed (project) that is open; server status; and, the capability to view the log file.
2. As long as the plug-in for each item in the table (Figure 1) is providing you will see a version number; server status will be up; and, for a log file you will be able to view. If you see **unknown** or **N/A**, that does not mean there is something wrong, it just means that the required information needed by HEC-RTS is not being provided.
3. If the ResSim, FIA, or RAS plug-in's server status is **Down**, select that plug-in's row, click **Start Server** (Figure 1), HEC-RTS will re-start the selected plug-in.
4. In the **Log File** column (Figure 1), if **View** is displayed, then you can click **View** and the log file for the selected plug-in will display (Figure 2).



25.7.2 Plugins Tab

From the **Options** dialog, click on the **Plug-ins** tab (Figure 3), a table of the plug-ins that are enabled by default for HEC-RTS will display. Besides providing a list of the available plug-ins, you can also disable plug-ins for all watersheds. It is not recommended that you disable plugins.



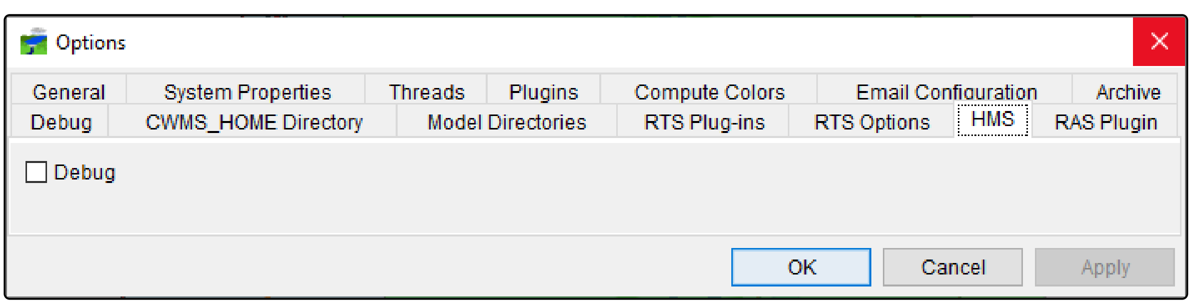
140 Figure 3 Options Dialog - Plugins Tab

25.8 Software Options

Further debugging information for a HEC-RTS forecast is created for two of the software applications that are part of the HEC-RTS default program order - HEC-HMS and HEC-RAS. The debugging information will display in the individual software applications' log files.

25.8.1 HMS Tab

Another debugging tool that is available to the user, is to debug the HEC-HMS software application. By default, the HEC-HMS debugging capability is turned off. From the **Options** dialog, click the **HMS** tab (Figure 1). To turn on HMS debugging, select **Debug** (Figure 1). Now from the HMS log file, the user will be able to view XML (eXtensible Markup Language) calls from the HMS software application.

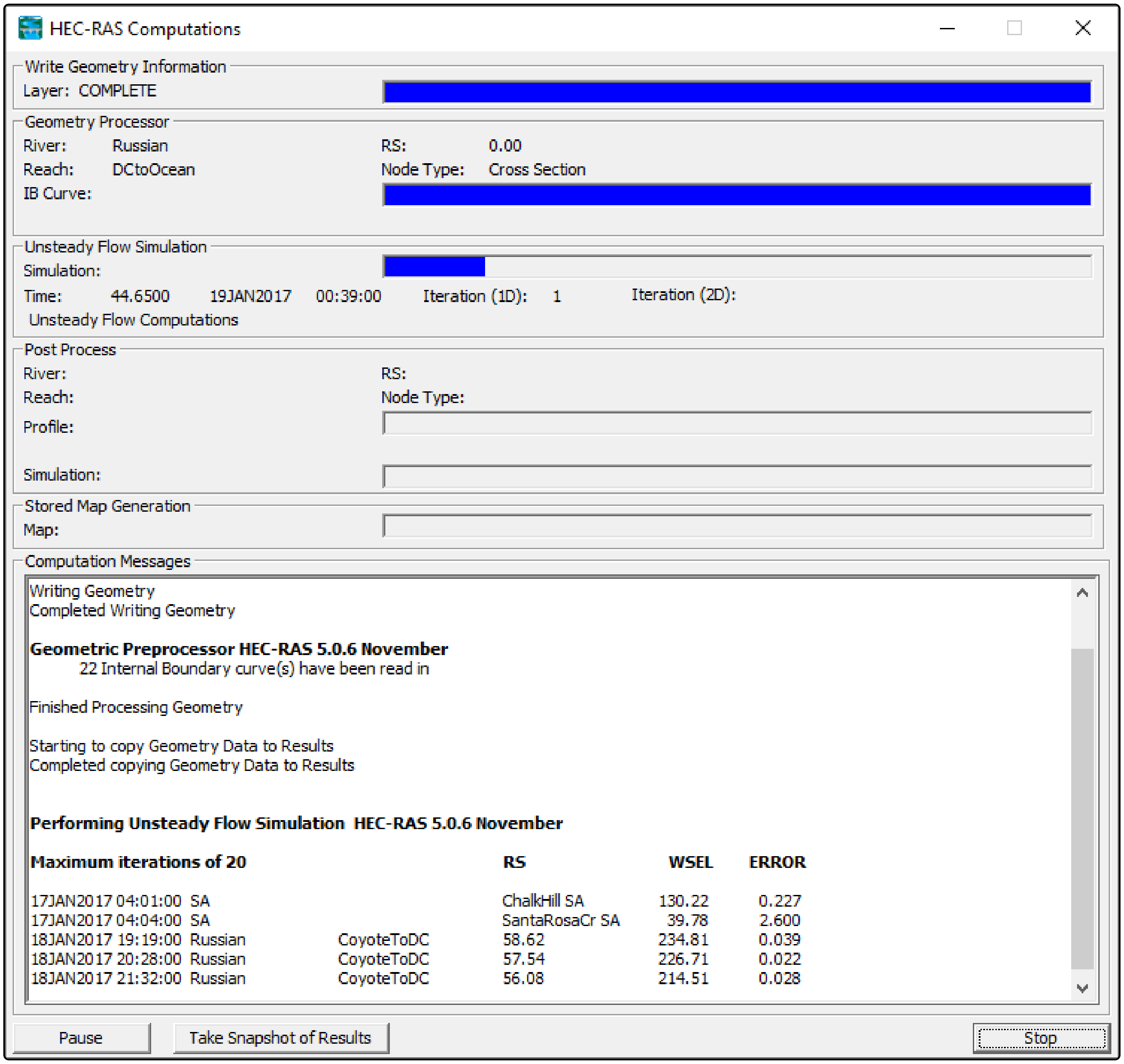


141 Figure 1 Options Dialog - HMS Tab

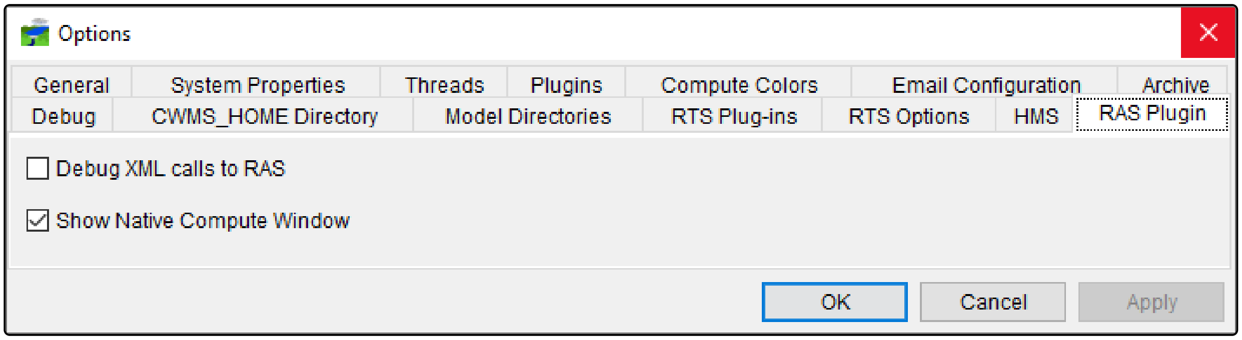
25.8.2 RAS Plugin Tab

For the HEC-RAS software application there are two debugging tools – display debug calls for the HEC-RAS software application and display the **HEC-RAS Computations** dialog (Figure 2). From the **Options** dialog, click the **RAS Plugin** tab (Figure 3).

To have debugging information included in the HEC-RAS log file, click **Debug XML calls to RAS** (Figure 3), this will debug XML calls in the HEC-RAS software application. To display further debugging information, click **Show Native Compute Window** (Figure 3), this will display the **HEC-RAS Computations** dialog (Figure 2) during a forecast run that includes HEC-RAS.



142 Figure 2 HEC-RAS Computations Dialog



143 Figure 3 Options Dialog RAS Plugin Tab

26 Setting Up the Coordinate System

HEC-RTS performs an "on-the-fly" coordinate system transformation and related software applications. This allows HEC-RTS to display map layers that were created in different coordinate systems in a unified view. For example, you can display your watershed with background maps from Internet sources such as Google® Maps and Open Street Map. This feature works correctly only if you define a geo-referenced coordinate system for the watershed and include a coordinate system definition (**.prj* file or equivalent) for each map layer included in the watershed. The positions of elements in the watershed's primary layers (time series icons, stream alignment, and study layers) are stored in the watershed coordinate system. You can set the watershed's coordinate system when creating a new watershed. This superimposes a grid on layer features to establish x- and y-coordinates in the World Coordinate System (WCS) for each point on the layer. The x-coordinate is referred to as "easting" and the y-coordinate is referred to as "northing" following customary use in surveying and mapping. The user can select the extent of this grid, the units of measurement of the grid, and even the location of the origin of the grid. This appendix provides you with information about the various coordinate systems that are supported by HEC-RTS.

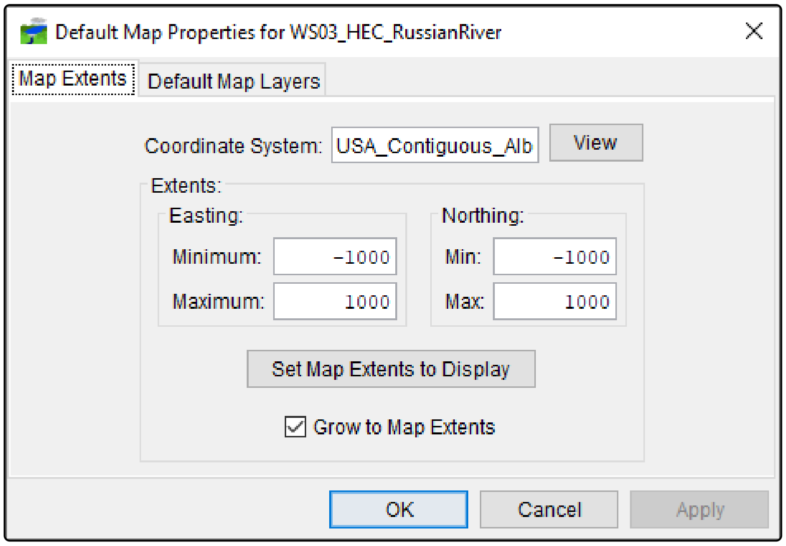
26.1 Geographic Referencing

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

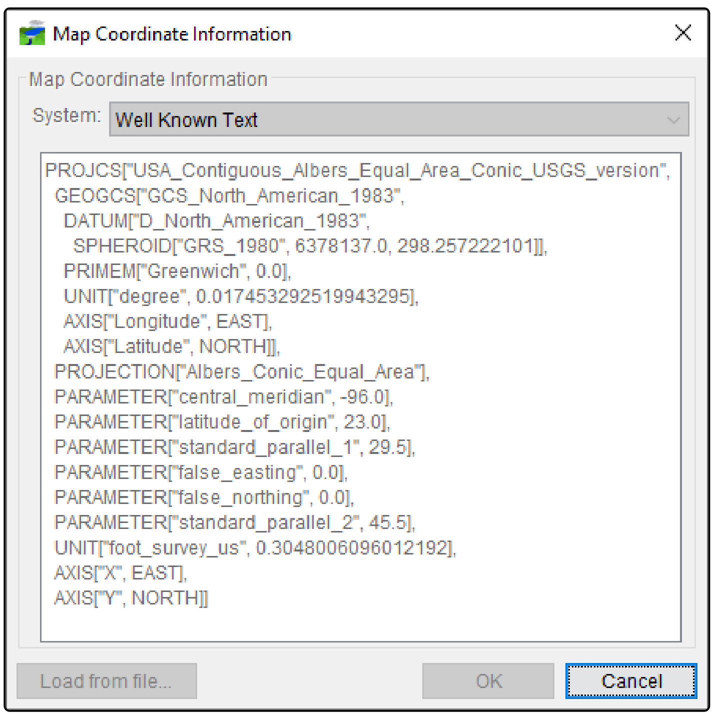
1. From the HEC-RTS main window, from **Maps**, click **Default Map Properties**. The **Default Map Properties** dialog (Figure 1) will open.
2. The **Default Map Properties** dialog contains two tabs: **Map Extents** and **Default Map Layers**. he **Map Extents** tab includes:

Coordinate System This field identifies the established coordinate system for the watershed. To view the coordinate system, click **View**. The **Map Coordinate Information** dialog (Figure 2) will open, providing you with information about the coordinate system that was established when the watershed was created. You are not able to edit this information once a watershed is created.

Extents Easting Minimum and **Maximum; Northing Minimum** and **Maximum**. These values (Figure 1) indicate the location of the left, right, bottom and top borders (respectively) of the grid in the map window.



144 Figure 1 Default Map Properties Dialog



145 Figure 2 Map Coordinate Information Dialog

Set Map Extents to Display This will set the limits of the map window. If you zoom in on an area and click **Set Map Extents to Display**, the extents on the **Default Map Properties** dialog (Figure 1) will change to the zoomed area.
Grow to Map Extents When selected, HEC-RTS automatically sets the geographic extents to define the smallest rectangle that encompasses all the objects in the watershed.

HEC-RTS can transform coordinates "on-the-fly" if the individual layers have coordinate systems that are defined properly in ways that the program can read, for example an ESRI projection (**.prj*) file. Therefore, you may need to use GIS tools to identify or transform the layers from one coordinate system to another before using them with HEC-RTS. Although HEC-RTS has the ability to project different coordinate systems "on the fly", using a consistent coordinate system may produce more accurate placement of map elements and may result in better performance in displaying large maps.

26.2 Coordinate System Types

A **coordinate system** is a method of representing points in a space of given dimensions by coordinates. There are several different types of coordinate systems; including geographic coordinate systems, which are based on latitude and longitude coordinates, and projected coordinate systems, which represent the projection of a geographic coordinate system on a plane and use linear measures (like feet or meters) for coordinates. A **spheroid** (ellipsoid) is the shape of the earth used in the calculations that transform positions on the curved surface of the earth to positions on a flat map. It is part of the horizontal datum, which approximates the curved surface of the earth over part of the globe.

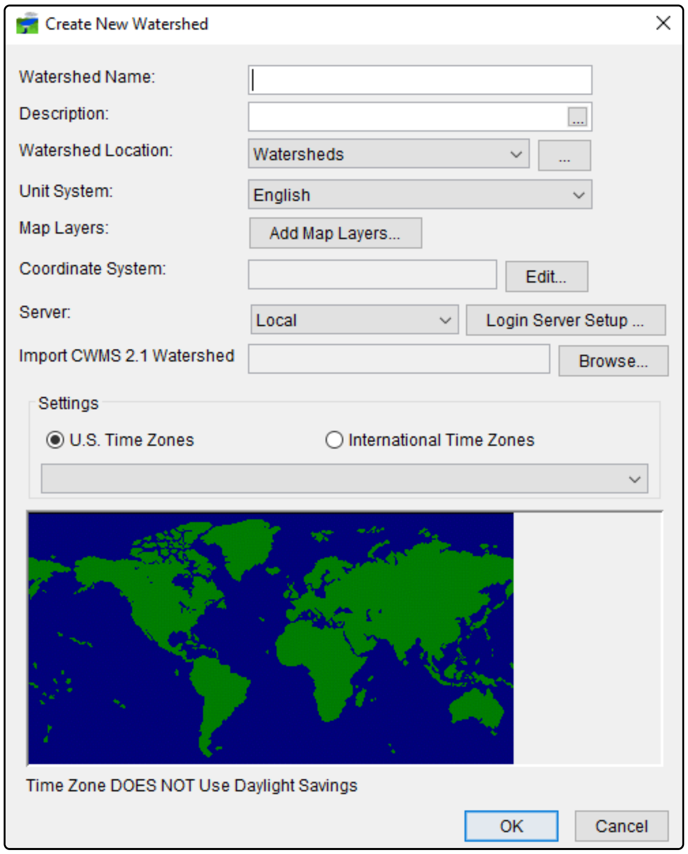
The coordinate systems available in HEC-RTS are shown in Table 1. This table also shows the units required for each coordinate system.

Table 1 Coordinate Systems Available in HEC-RTS

Coordinate Systems	**Units**
X-Y	Liner units (feet or meters)
Google/Bing Web Mercator	Liner units (feet or meters)
Geographic	Angular units (radians, degrees of arc, or seconds)
Universal Transverse Mercator	Liner units (feet or meters)
State Plane Coordinates	Liner units (feet or meters)
Albers Equal-Area Conic	Liner units (feet or meters)
Lambert Conformal Conic	Liner units (feet or meters)
Transverse Mercator	Liner units (feet or meters)
Albers Equal-Area Conic (SHG)	Liner units (feet or meters)
Polar Stereographic (HRAP)	Liner units (feet or meters)

Since Version 3.1.1 of HEC-RTS, the creation of a geo-referenced coordinate system for a watershed has been part of the creation of a HEC-RTS watershed and is done through "importing" information from coordinate system definition (**.prj* file or equivalent) for each map layer that is included in a watershed. The creation of a watershed is detailed in Creating a Watershed.

The following section provided information about each coordinate system that is recognized by HEC-RTS. Once a watershed is created, you cannot change the coordinate system. To view any of this information, from the HEC-RTS main window, from the **File** menu, click **New Watershed**. The **Create New Watershed** dialog (Figure 1) will open. To set the coordinate system of the watershed, to the right of the **Coordinate System** box (Figure 1), click **Edit**, the **Map Coordinate Information** dialog will open.



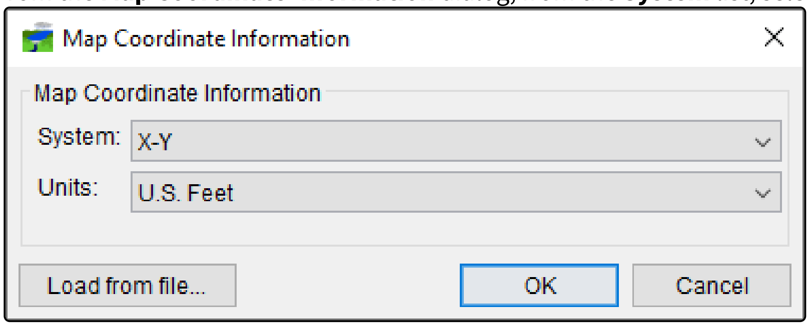
146 Figure 1 Create New Watershed Dialog

26.2.1 X-Y Coordinate System

The label "X-Y" can be applied to a coordinate system that is either not geo-referenced or for which the geo-referencing method is unknown. When the "X-Y" coordinate system is selected, HEC-RTS cannot project maps to the watershed's coordinate system.

To set parameters for the X-Y coordinate system:

1. From the **Map Coordinate Information** dialog, from the **System** list, select **X-Y** (Figure 2).

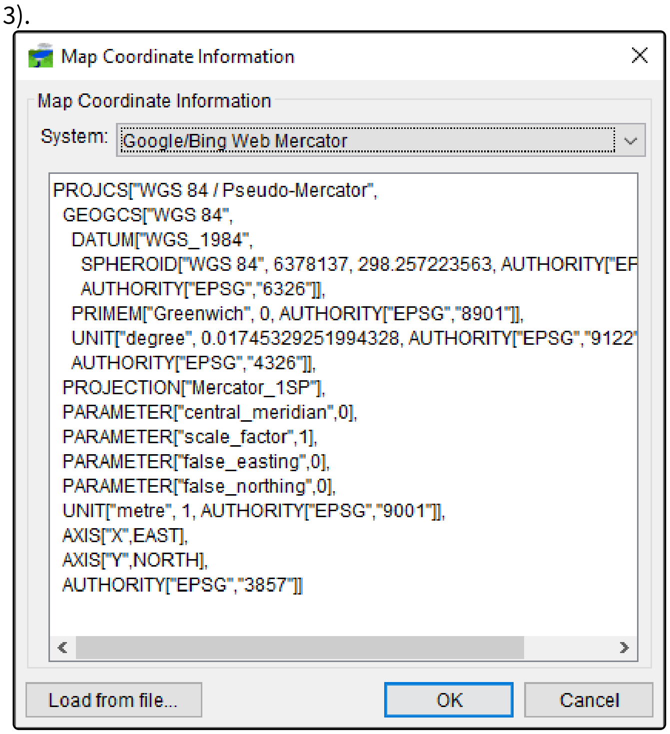


2. From the **Units** list, select the units for the X-Y coordinate system.
3. Click **OK**, and the **Map Coordinate Information** dialog will close, and you will be returned to the **Create New Watershed** dialog box. In the **Coordinate System** box (Figure 1), *X-Y* should be displayed.

26.2.2 Google/Bing Web Mercator Coordinate System

To set parameters for the Google/Bing Web Mercator coordinate system:

1. From the **Map Coordinate Information** dialog, from the **System** list, select **Google/Bing Web Mercator** (Figure 3).

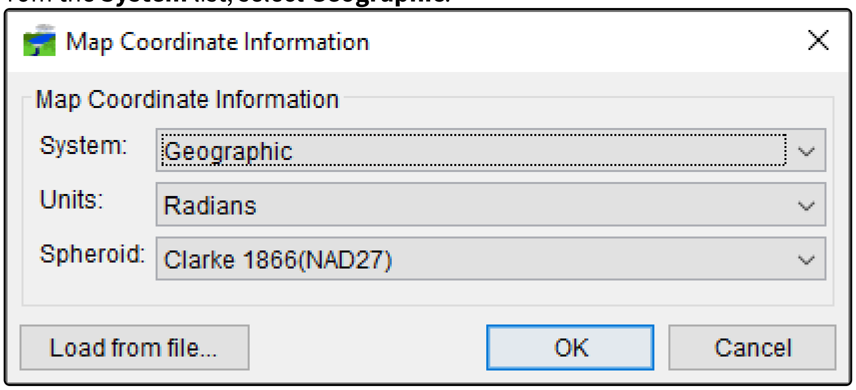


2. Information about the Google/Bing Web Mercator coordinate system is provided (Figure 3).
3. Click **OK**, and the **Map Coordinate Information** dialog will close, and you will be returned to the **Create New Watershed** dialog box. In the **Coordinate System** box (Figure 1), *Google/Bing Web Mercator* should be displayed.

26.2.3 Geographic Coordinate System

To set parameters for the Geographic coordinate system:

1. From the **Map Coordinate Information** dialog, from the **System** list, select **Geographic** (Figure 4).
2. From the **System** list, select **Geographic**.



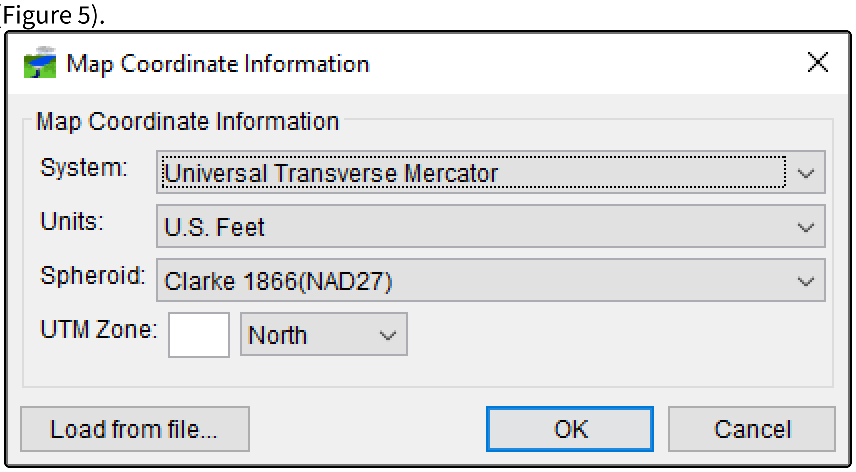
3. From the **Units** list, select the units for the Geographic coordinate System. The only units available should be **Radians, Degrees of Arc,** and **Seconds of Arc**.
4. From the **Spheroid** list, select a spheroid type for the Geographic coordinate system. (The **Sphere of Radius 6371200 meters** should only be used with the HRAP grid coordinate system.)
5. Click **OK**, and the **Map Coordinate Information** dialog will close, and you will be returned to the **Create New Watershed** dialog box. In the **Coordinate System** box (Figure 1), *Geographic* should be displayed.

26.2.4 Universal Transverse Mercator Coordinate System

The Universal Transverse Mercator (UTM) coordinate system is a projected coordinate system. UTM is used to define horizontal positions throughout the world by dividing the surface of the Earth into six-degree zones, with a central meridian in the center of each zone.

To set parameters for the Universal Transverse Mercator Coordinate System:

1. From the **Map Coordinate Information** dialog, from the **System** list, select **Universal Transverse Mercator** (Figure 5).



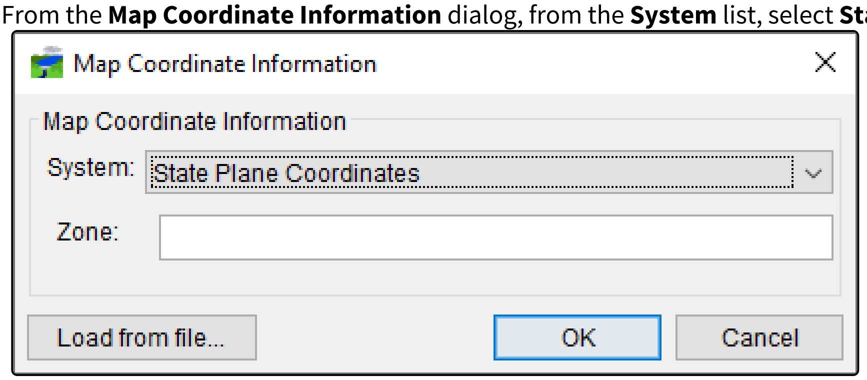
2. From the **System** list, select **Universal Transverse Mercator**.
3. From the **Units** list, select the units for the Universal Transverse Mercator coordinate system.
4. From the **Spheroid** list, select a spheroid type for the Universal Transverse Mercator coordinate system.
5. In the **UTM Zone** box, enter the UTM zone number. To use a UTM zone in the southern hemisphere, enter a negative value for the zone number.
6. Click **OK**, and the **Map Coordinate Information** dialog will close, and you will be returned to the **Create New Watershed** dialog box. In the **Coordinate System** box (Figure 1), *Universal Transverse Mercator* should be displayed.

26.2.5 State Plane Coordinates System

The State Plane Coordinates System (SPCS) was established in the 1930's and now covers all fifty states. Zones were established for each state using either the Lambert Conformal or Traverse Mercator projections. Units are generally in feet (NAD27) or meters (NAD83).

To set parameters for the State Plane Coordinates coordinate system:

1. From the **Map Coordinate Information** dialog, from the **System** list, select **State Plane Coordinates** (Figure 6).



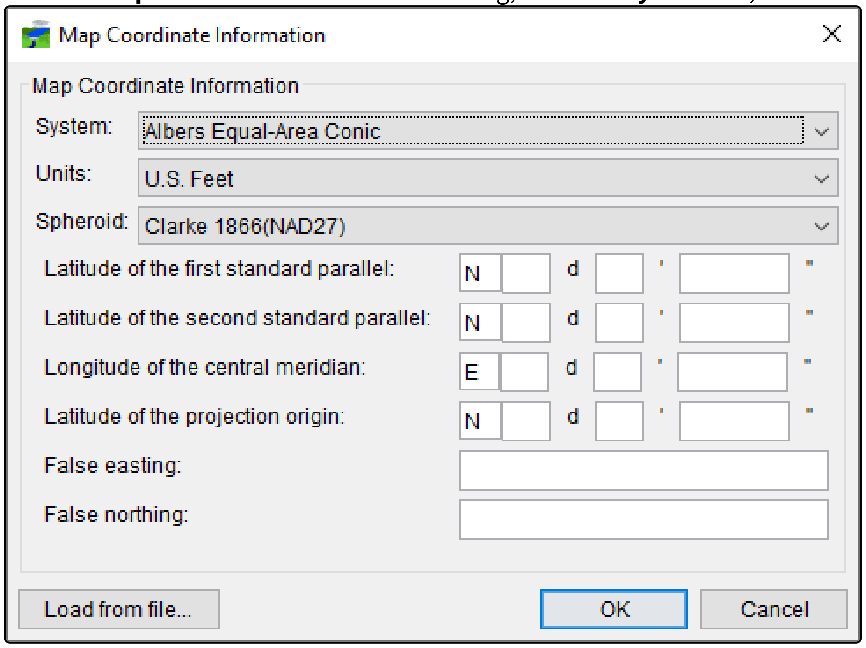
2. From the **System** list, select **State Plane Coordinates**.
3. In the **Zone** box, enter the FIPS code for the state plane zone.
4. Click **OK**, and the **Map Coordinate Information** dialog will close, and you will be returned to the **Create New Watershed** dialog box. In the **Coordinate System** box (Figure 1), *State Plane Coordinates* should be displayed.

26.2.6 Albers Equal-Area Conic Coordinate System

The Albers Equal-Area Conic projection is used by several federal government agencies for maps of the conterminous 48 states. HEC uses an Albers projection for the definition of the Standard Hydrologic Grid. The equal area property of this projection means that areas in the map are proportional to the corresponding areas on the earth's surface. Directions and shapes distort increasingly away from the central meridian and outside the two standard parallels.

To set parameters for the Albers Equal-Area Conic Coordinate System:

1. From the **Map Coordinate Information** dialog, from the **System** list, select **Albers Equal-Area Conic** (Figure 7).



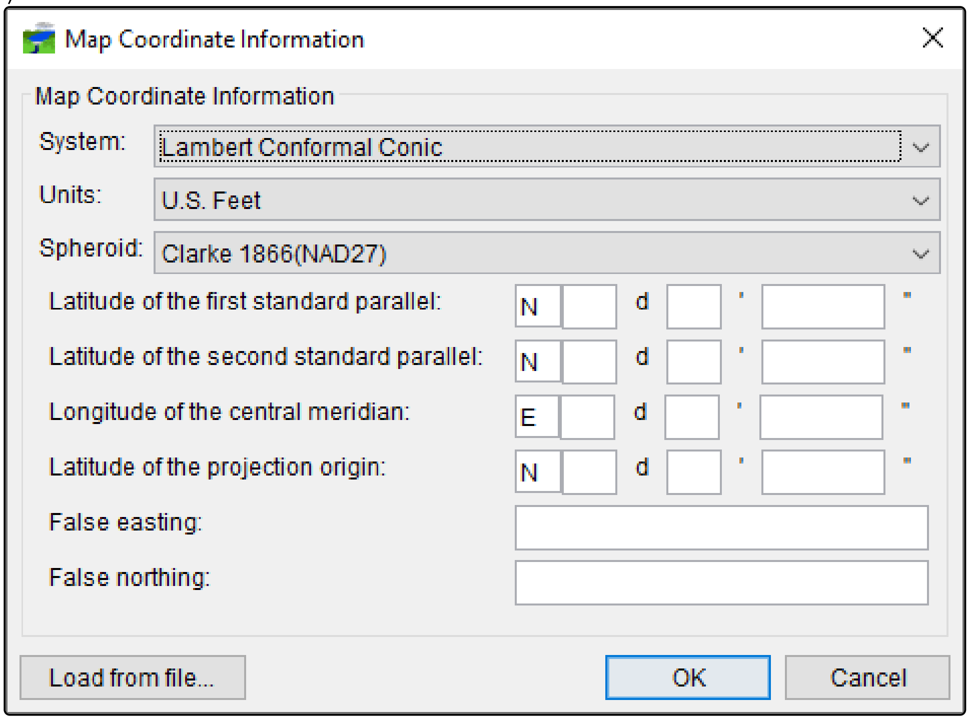
2. From the **System** list, select **Albers Equal-Area Conic**.
3. From the **Units** list, select the units for the Albers Equal-Area Conic coordinate system.
4. From the **Spheroid** list, select a spheroid type for the Albers Equal- Area Conic coordinate system.
5. The remaining items to be entered are the angular parameters that are required to fine-tune the projection. When specifying latitudes, you will enter **N** or **S**, and for longitudes, you will enter **E** or **W**. Use your tab key to toggle between the entries.
6. Click **OK**, and the **Map Coordinate Information** dialog will close, and you will be returned to the **Create New Watershed** dialog box. In the **Coordinate System** box (Figure 1), *Albers Equal-Area Conic* should be displayed.

26.2.7 Lambert Conformal Conic Coordinate System

The Lambert Conformal Conic projection is used extensively for mapping areas of the world with predominantly east-west orientation. It is like the Albers Equal-Conic projection, but the projection is not done in an equal-area.

To set parameters for the Lambert Conformal Conic coordinate system:

1. From the **Map Coordinate Information** dialog, from the **System** list, select **Lambert Conformal Conic** (Figure 8).



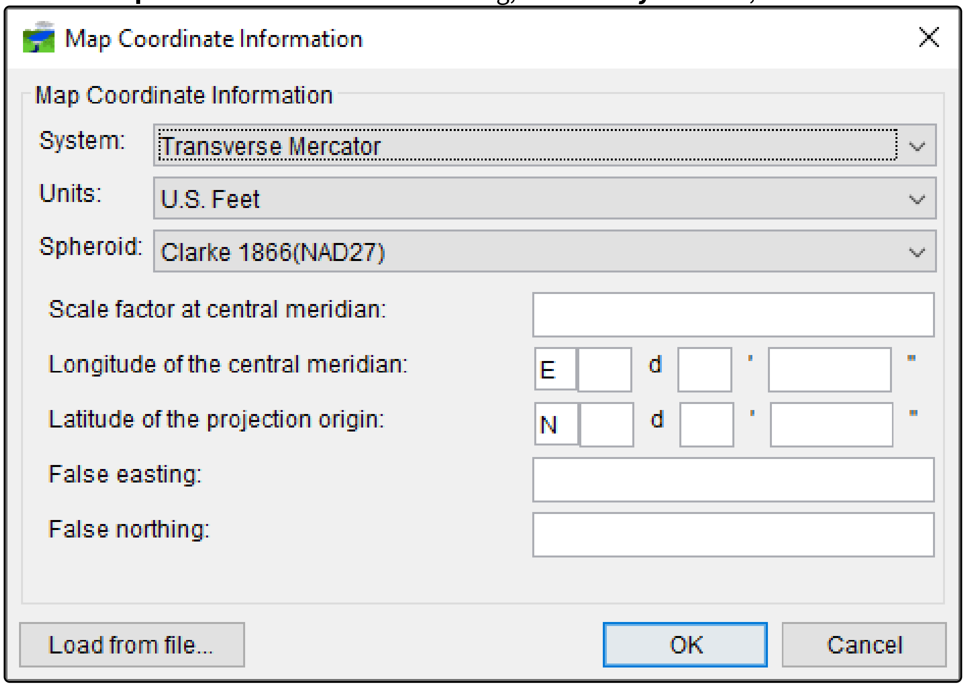
2. From the **System** list, select **Lambert Conformal Conic**.
3. From the **Units** list, select the units for the Lambert Conformal Conic coordinate system.
4. From the **Spheroid** list, select a spheroid type for the Lambert Conformal Conic coordinate system.
5. When specifying latitudes, you will enter **N** or **S**, and for longitudes, you will enter **E** or **W**. Use your tab key to toggle between the entries.
6. Click **OK**, and the **Map Coordinate Information** dialog will close, and you will be returned to the **Create New Watershed** dialog box. In the **Coordinate System** box (Figure 1), *Lambert Conformal Conic* should be displayed.

26.2.8 Transverse Mercator Coordinate System

The Transverse Mercator projection is where a sphere is projected onto a cylinder tangent to a central meridian. It is like the Lambert Conformal Conic project but is used to portray large areas in a north-south orientation. Many national grid systems are based on the Transverse Mercator projection.

To set parameters for the Transverse Mercator coordinate system:

1. From the **Map Coordinate Information** dialog, from the **System** list, select **Transverse Mercator** (Figure 9).



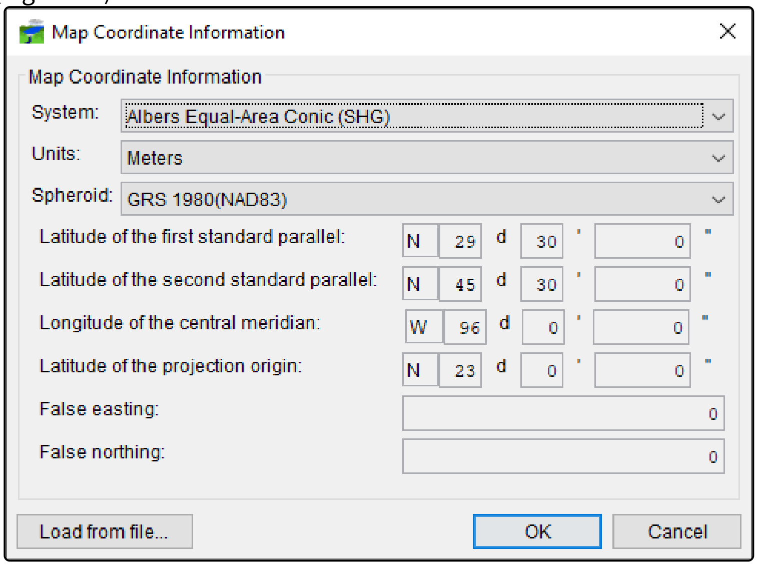
2. From the **System** list, select **Transverse Mercator**.
3. From the **Units** list, select the units for the Transverse Mercator coordinate system.
4. From the **Spheroid** list, select a spheroid type for the Transverse Mercator coordinate system.
5. Enter a scaling factor for central meridian, in the **Scale factor at central meridian** cell.
6. When specifying latitudes, you will enter **N** or **S**, and for longitudes, you will enter **E** or **W**. Use your tab key to toggle between the entries.
7. Click **OK**, and the **Map Coordinate Information** dialog will close, and you will be returned to the **Create New Watershed** dialog box. In the **Coordinate System** box (Figure 1), *Transverse Mercator* should be displayed.

26.2.9 Albers Equal-Area Conic (SHG) Coordinate System

This is a special case of the Albers Equal-Area Conic coordinate system. The projection parameters are set to match USGS 48-state maps, which are used for several common data sources including precipitation data on the Standard Hydrologic Grid. If you select this option for the coordinate system, all parameter fields will be filled in the editor. This coordinate system is also used by USACE's MMC, but with units of US Feet instead of meters. You can quickly set your watershed to MMC's coordinate system by selecting the SHG system and then switching the units to "US Feet".

To set parameters for the Albers Equal-Area Conic (SHG) coordinate system:

1. From the **Map Coordinate Information** dialog, from the **System** list, select **Albers Equal-Area Conic (SHG)** (Figure 10).



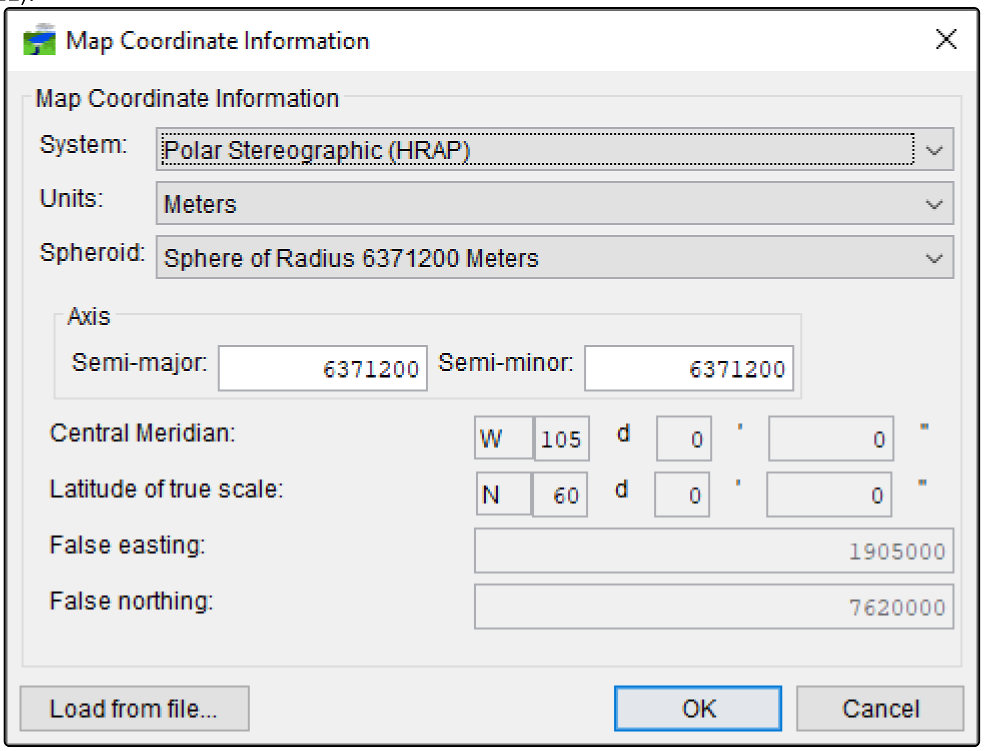
2. From the **System** list, select Albers Equal-Area Conic (SHG).
3. From the **Units** list, select the units for the Albers Equal-Area Conic (SHG) coordinate system.
4. From the **Spheroid** list (Figure 10), select a spheroid type for the Albers Equal-Area Conic (SHG) coordinate system.
5. The remaining parameters are informational and are not editable.
6. Click **OK**, and the **Map Coordinate Information** dialog will close, and you will be returned to the **Create New Watershed** dialog box. In the **Coordinate System** box (Figure 1), *Albers Equal-Area Conic (SHG)* should be displayed. Note that the (SHG) label is not part of the displayed coordinate system name, and that if you re-open the editor you will be able to edit all parameters of the projection.

26.2.10 Polar Stereographic (HRAP) Coordinate System

The Hydrologic Rainfall Analysis Project grid is used by the US National Weather Service for a variety of precipitation products. The row and column numbers in the grid are based on a Polar Stereographic projection of the northern hemisphere. The parameters are set to match NWS map coordinates used for radar grids, so do not change any parameter settings.

To use the parameters for the Polar Stereographic (HRAP) coordinate system:

1. From the **Map Coordinate Information** dialog, from the **System** list, select **Polar Stereographic (HRAP)** (Figure 11).

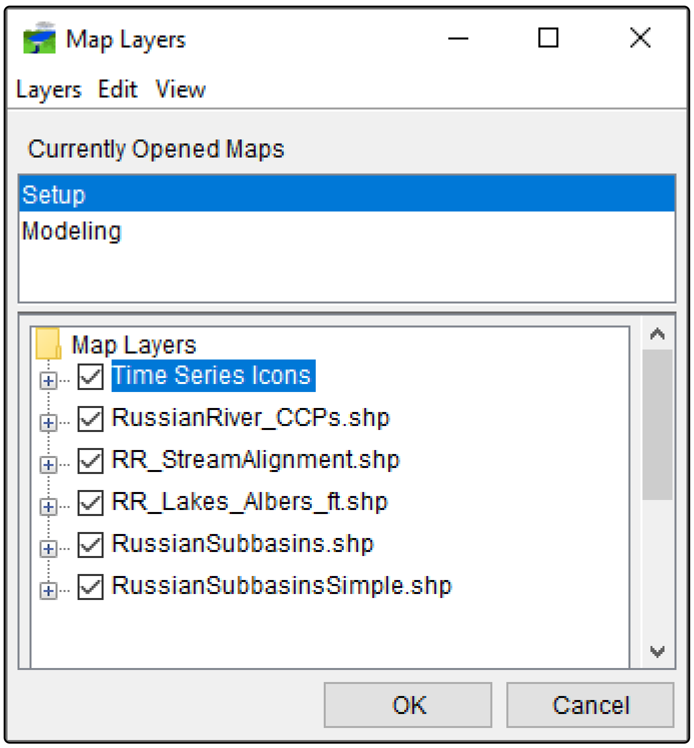


2. From the **System** list, select **Polar Stereographic (HRAP)**.
3. For Version 3.1.1, ignore the items in the Axis box; this will be removed from this dialog box in a later version of HEC-RTS.
4. The remaining parameters are informational and are not editable.
5. Click **OK**, and the **Map Coordinate Information** dialog will close, and you will be returned to the **Create New Watershed** dialog box. In the **Coordinate System** box (Figure 1), *Polar Stereographic (HRAP)* should be displayed.

27 Using Map Editors

There are twelve map layer formats supported by HEC-RTS, and the different map layers are described in Chapter 5 of this manual. In HEC-RTS you can configure several options for each type of map layer, except for AutoCAD® DXF files. Additional customization is in development for future versions of the software.

To access the **Map Layers** dialog, from the HEC-RTS main window, from the **Maps** menu, click **Map Layers**. The **Map Layers** dialog will open (Figure 1). From the **Layer Tree** either double-click on a map layer name, or right-click on a map layer name and click **Properties** from the shortcut menu. An editor specific to the type of map layer you selected will open.



147 Figure 1 Map Layers Dialog

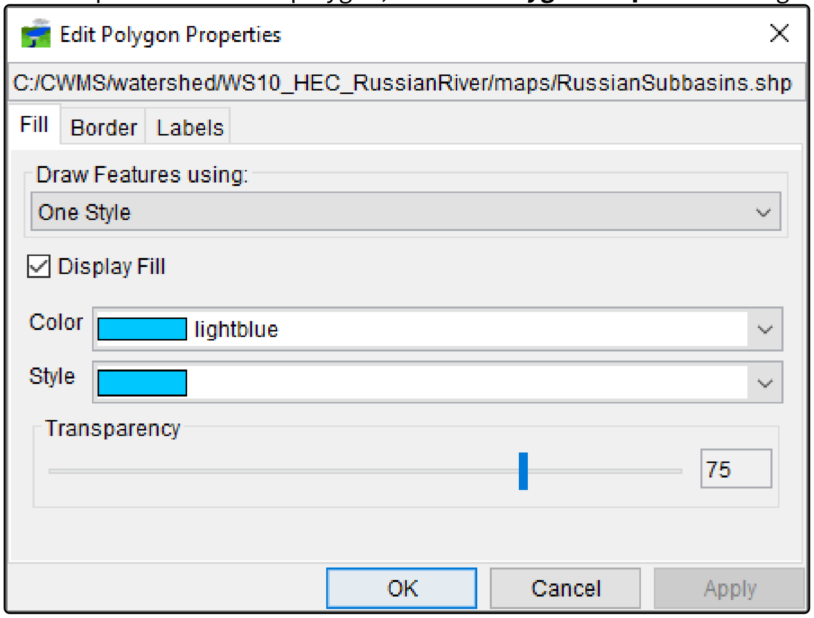
27.1 Arc Shapefiles (.shp)

hapefiles store non-topological geometry and attribute information for geo-referenced points, lines, or polygons. The format is native to ESRI's ArcGIS® software. Vector GIS layers can be exported from ArcGIS databases to shapefiles for display in HEC-RTS studies.

To edit the display properties of a line, polygon, or point shapefile:

1. From the **Layer Tree** (Map Layers Dialog), right-click on an Arc shapefile (**.shp) from the shortcut menu and click *Properties** or double-click on the Arc shapefile filename.

2. If the shapefile chosen is a polygon, the **Edit Polygon Properties** dialog will open (Figure 1).

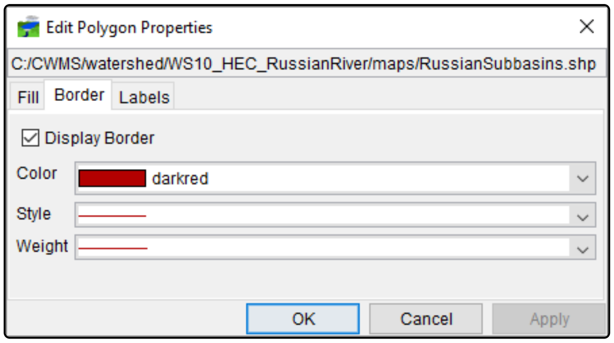


3. Across the top of the dialog is displayed the location and name of the shapefile.

To edit properties of an ArcGIS® shapefile:

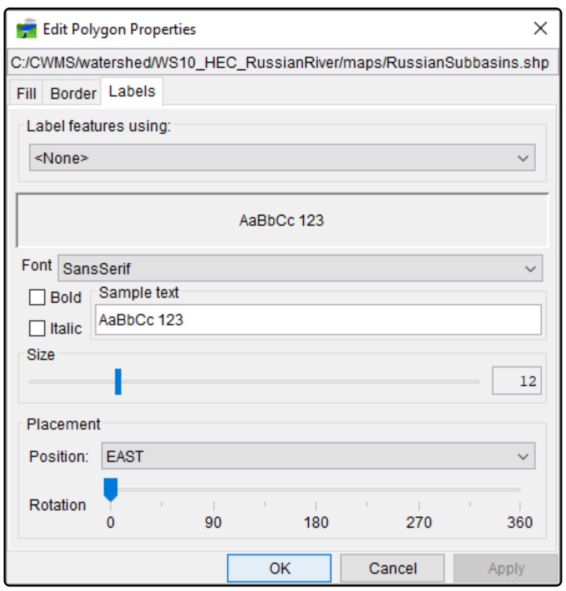
1. Click the **Fill** tab of the **Edit Polygon Properties** dialog (Figure 1).
2. The **Draw Features using** option allows you to select how the feature will be drawn, either using **One Fill** or the **Attribute Values**. If **Attribute Values** is selected, this option will associate the **Field for values** from the shapefile to a color and label. In Figure D.2, *AREA* has been selected, which displays the value of the *AREA* in the shapefile as blue to red gradations.
3. By default, the shapefile will display lines and areas (polygons). To turn off the fill, clear the **Display Fill** cell.
4. To change the style of line or polygon areas use the **Style** selection list
5. To classify the values in the **Field for values** check the **Use Gradations** cell and specify the number of classes. This will group the values in the **Field for values** into the range of classes specified.
6. The **Transparency** bar controls to what level the transparency of the shapefile will be displayed.
7. The properties of the **Color Chooser** are discussed in detail in Appendix E. From the color palette, select a color. If the feature is being drawn using an attribute value, select a color for the start and end value.
8. Click **Apply**, and the color for the selected field will appear in the display area. Click **OK**, and the **Edit Polygon Properties** dialog will close.
9. If you wish to save any changes, from the **File** menu, choose **Save Watershed**. Changes are saved to a file with the extension **.gdr*. For example, if the shapefile name is *subbasin.shp*, and changes have been made and saved, a file *subbasin.gdr* will be created.

To change the border color, style and weight of the ArcGIS® shapefile, click the **Border** tab of the **Edit Polygon Properties** dialog (Figure 2). You can change the border display setting in from the **Edit Polygon Properties** dialog.



148 Figure 2 Edit Polygon Properties Dialog - Border Tab

To format the labels of the ArcGIS® shapefile. click the **Label** tab of the **Edit Polygon Properties** dialog (Figure 3). If you wish to display labels associated with the shapefile, click on the **Label features using** drop down window and choose the attribute field which the labeling will be based on. Additionally, the **Font**, **Size**, and **Placement** of the label can be changed.



149 Figure 3 Edit Polygon Properties Dialog - Labels Tab

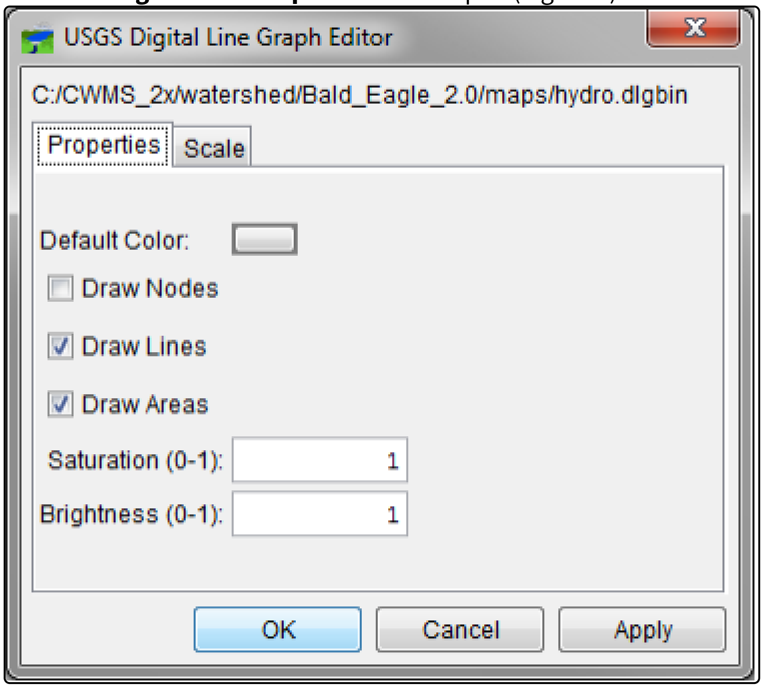
27.2 USGS Digital Line Graph (.dlg)

This layer type is a vector representation of the data. When HEC-RTS interacts with a *dlg* file it automatically creates a *dlgbin* file for use.

To edit a USGS Digital Line Graph file:

1. From the **Layer Tree** (**Map Layers** dialog), right-click on a USGS Digital Line Graph file. From the shortcut menu, click **Properties**, or double-click on the USGS Digital Line Graph filename.

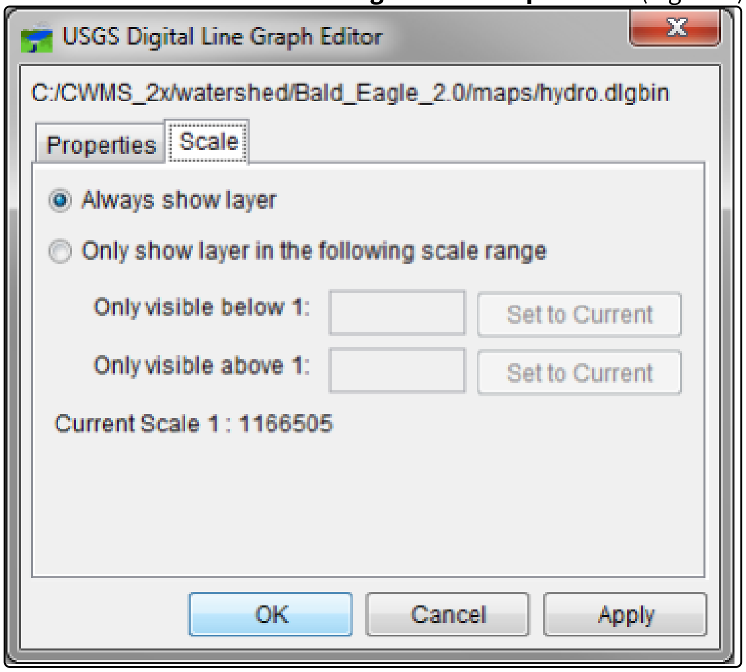
2. The **USGS Digital Line Graph Editor** will open (Figure 1).



3. Across the top of the dialog is displayed the location and name of the USGS Digital Line Graph file.
4. You can control the appearance of nodes, line, and areas by selecting and clearing **Draw Nodes**, **Draw Lines**, and **Draw Areas**, respectively (Figure 1).
5. You can also control the saturation and brightness level of the color you have selected by entering values in the **Saturation** or **Brightness** cells. Values entered for saturation or brightness must be between zero (0) and one (1).
6. If you wish to save any changes, from the HEC-RTS main window, from the **File** menu, click **Save Watershed**. The color for a USGS digital line graph file is stored in the watershed file (**.wtshd*).

To set scaling for a USGS Digital Line Graph:

1. Click the **Scale** tab of the **USGS Digital Line Graph Editor** (Figure 2).



2. Both zoom-in and zoom-out scale factors can be set. By default the digital line graph layer is set for **Always show layer**. To set visualization scales click **Only show layer in the following scale range**.
3. To set the scale 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**. If you wish to enter a value, from the **Only visible below 1:** cell enter the scale factor.
4. 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**. If you wish to enter a value, from the **Only visible above 1:** cell enter the scale factor.
5. Click **OK**, the **USGS Digital Line Graph Editor** will close (Figure 2), and the visualization scale is set for the digital line graph layer.
6. Click **OK**, the **Map Layers** dialog closes.

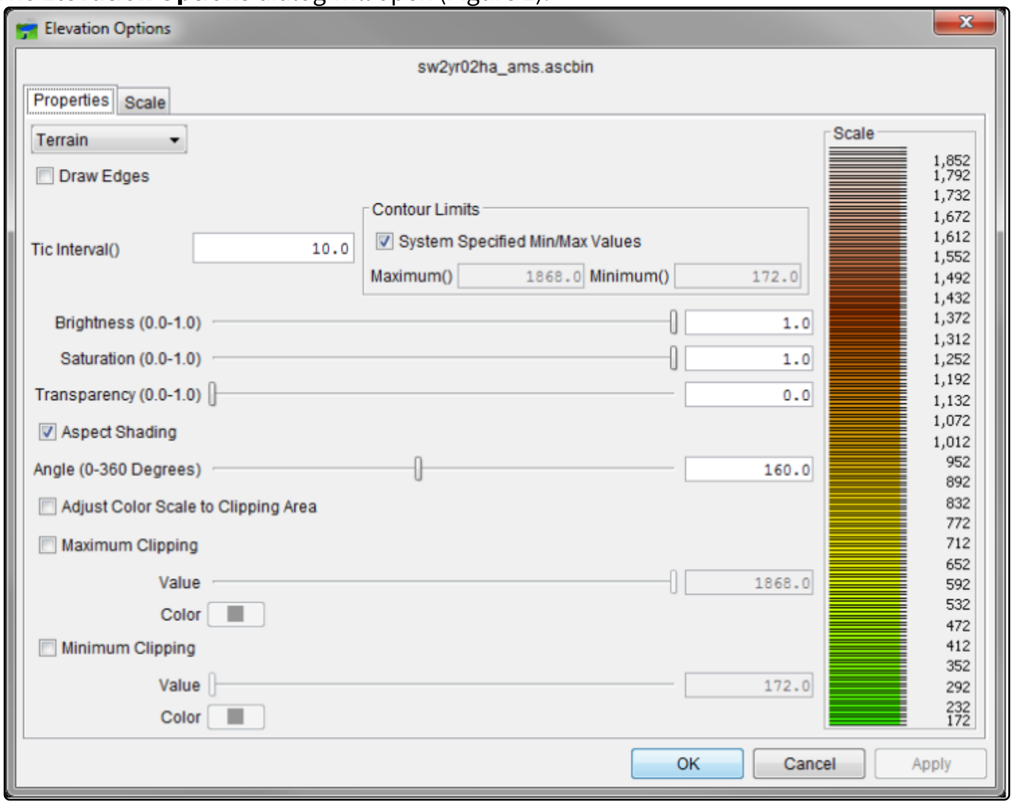
27.3 Elevation Options Dialog

For the other map layer formats – USGS DEM, ASCII NetTin, and ArcGIS® DEM; they all have the same editor, the **Elevation Options** dialog.

To edit an ASCII NetTin file:

1. From the **Layer Tree** (**Map Layers** dialog), right-click on an ASCII NetTin file. From the shortcut menu, click **Properties,** or double-click on the ASCII NetTin name.

2. The **Elevation Options** dialog will open (Figure 1).



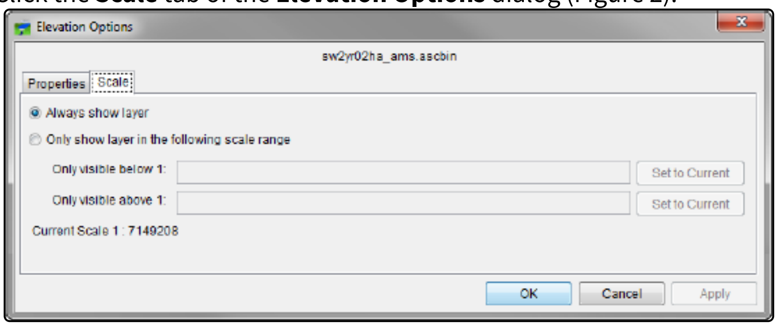
3. Across the top of the dialog, is displayed the name of the ASCII NetTin file.
4. Click the **Properties** tab of the **Elevation Options** dialog (Figure 1).
5. From the dropdown list, there are available color contour schemes – **Aspect Shading, Grayscale, Linear, Precipitation, Red-Green-Blue,** and **Terrain** (default).
6. The **Draw Edges** check box is specifically for ASCII NetTin files. If selected, the edges of the triangles the make up an ASCII NetTin file will be drawn.
7. You can set the contour tic interval, by entering a value in the **Tic Interval** cell. The program sets the maximum and minimum limits of the contours automatically. You can set your own maximum and minimum limits. Clear **System Specified Min/Max Values**, the **Maximum** and **Minimum** cells (Figure 1) are now available for you to enter values.
8. You can control the brightness (amount of white), saturation (amount of black), and transparency (level of opacity or alpha) for the colors. You can adjust the values by using the **Brightness**, **Saturation**, and **Transparency** slider bars or cells.
9. By default the **Aspect Shading** option is on, aspect shading is where you make the map layer appear in relief by placing an imaginary light source above the map and shading the contours. You can adjust the angle of the light source by using the **Angle** slider bar or the cell. If you do not want aspect shading, clear **Aspect Shading**.
10. Clipping provides you a way to highlight an area based on your color choices and values. For maximum clipping, you would fill the contour with the clip color from the maximum value of the map to the value entered for **Maximum Clipping**. An example of where you might use maximum clipping would be smog levels, cloud cover, or snow level at a particular elevation. By default, maximum clipping is turned off, to select click **Maximum Clipping**. Then either enter a maximum clipping value by using the slider bar or enter a value in the cell. You can also set the **Maximum Clipping** color. Click **Color**, the **Color Chooser** will open (see Using the Color Chooser for

more on the color chooser). From the color palette, select a color, click **OK**. The **Color Chooser** will close and the selected color will now appear on the **Color** button and at the top of the **Scale** (Figure 1).

11. Minimum clipping will fill the contour with the clip color from the minimum value of the map to the value entered for **Minimum Clipping**. An example of where you might use minimum clipping would be to see where a water level would be if it flooded to a particular elevation. By default, minimum clipping is turned off, to select click **Minimum Clipping**. Then either enter a minimum clipping value by using the slider bar or enter a value in the cell. You can also set the **Minimum Clipping** color. Click **Color**, the **Color Chooser** will open (see Using the Color Chooser for more on the color chooser). From the color palette, select a color, click **OK**. The **Color Chooser** will close and the selected color will now appear on the **Color** button and at the bottom of the **Scale** (Figure 1).

To set scaling for an ASCII NetTin file:

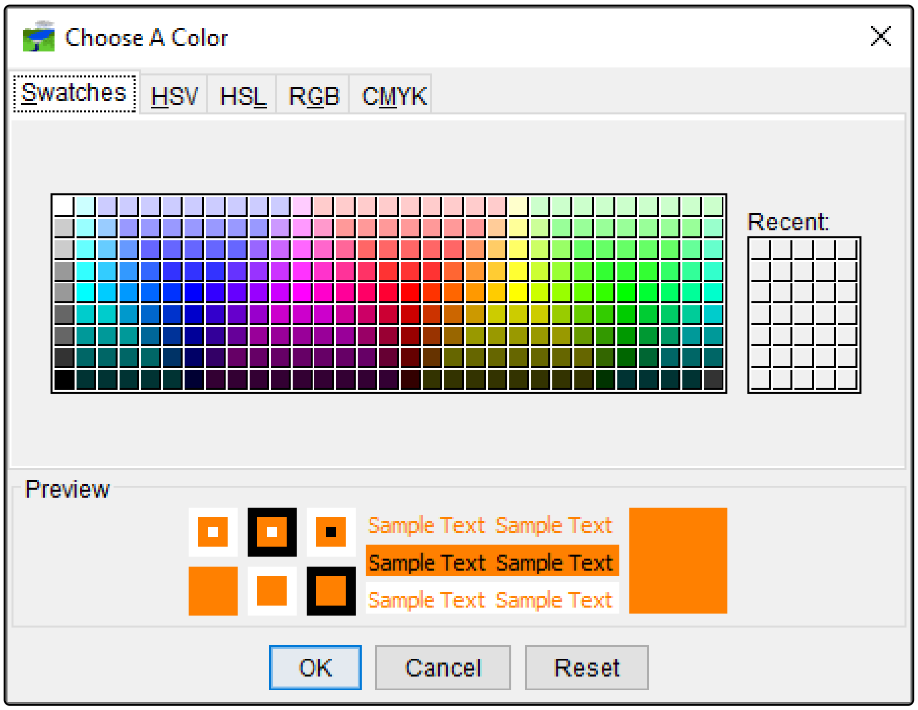
1. Click the **Scale** tab of the **Elevation Options** dialog (Figure 2).



2. Both zoom-in and zoom-out scale factors can be set. By default the ASCII NetTin layer is set for **Always show layer**. To set visualization scales click **Only show layer in the following scale range**.
3. To set the scale so that the ASCII NetTin layer becomes visible as you zoom-in, you can set the scale to the current scale by clicking **Set to Current**. If you wish to enter a value, from the **Only visible below 1:** cell enter the scale factor.
4. To set the scale so that the ASCII NetTin layer becomes visible as you zoom-out, you can set the scale to the current scale by clicking **Set to Current**. If you wish to enter a value, from the **Only visible above 1:** cell enter the scale factor.
5. Click **OK**, the **Elevation Options** dialog will close (Figure 2), and the visualization scale is set for the ASCII NetTin layer.
6. Click **OK**, and the **Map Layers** dialog closes.

28 Using the Color Chooser

The **Color Chooser** window (Figure 1) affords great flexibility when you need to select default colors for map layers, labels, and background colors in your watershed display. The **Color Chooser** window has five tabs: **Swatches**, **HSV**, **HSL**, **RGB**, and **CMYK** - offering five methods for choosing a color. For each method, the preview area allows you to see your choice before applying changes.



150 Figure 1 Color Chooser - Swatches Tab

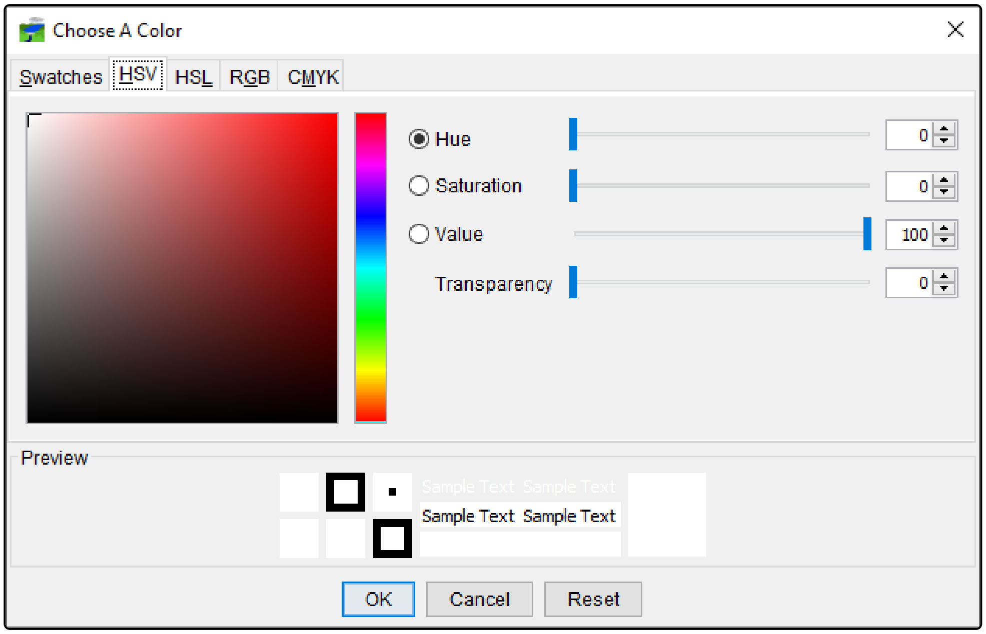
28.1 Swatches Tab

The **Swatches** tab provides a palette of pre-defined colors. From the palette, select a color. Once you have selected a color, the **Recent** field displays that color, as also does the **Preview** field. Click **OK**, and the **Color Chooser** dialog will close. Depending on where you accessed the **Color Chooser**, you will have to click either **Apply** or **OK** for the color change to appear in the display area.

For example, for a stream alignment, you would have accessed the **Color Chooser** from the **Stream Alignment Properties** dialog (Stream Alignment Layer). Once you have selected the color from the **Color Chooser** dialog and clicked **OK**, you will need to click **OK** or **Apply** from the **Stream Alignment Properties** dialog. Then you need to click **OK** from the **Map Layer** dialog for the color change to appear in the **Map Window**.

28.2 HSV Tab

HSV is the Hue, Saturation, and Value color model, replaces the RGB (Red, Green, and Blue) color model in graphics and paint software applications. The HSV representation models the way paints of different colors mix together, with the saturation dimension resembling various tints of brightly colored paint, and the value dimension resembling the mixture of those paints with varying amounts of black or white paint. From the **Color Chooser** window, select the **HSV** tab; and the **HSV** tab (Figure 1) becomes available.



151 Figure 1 Color Chooser - HSV Tab

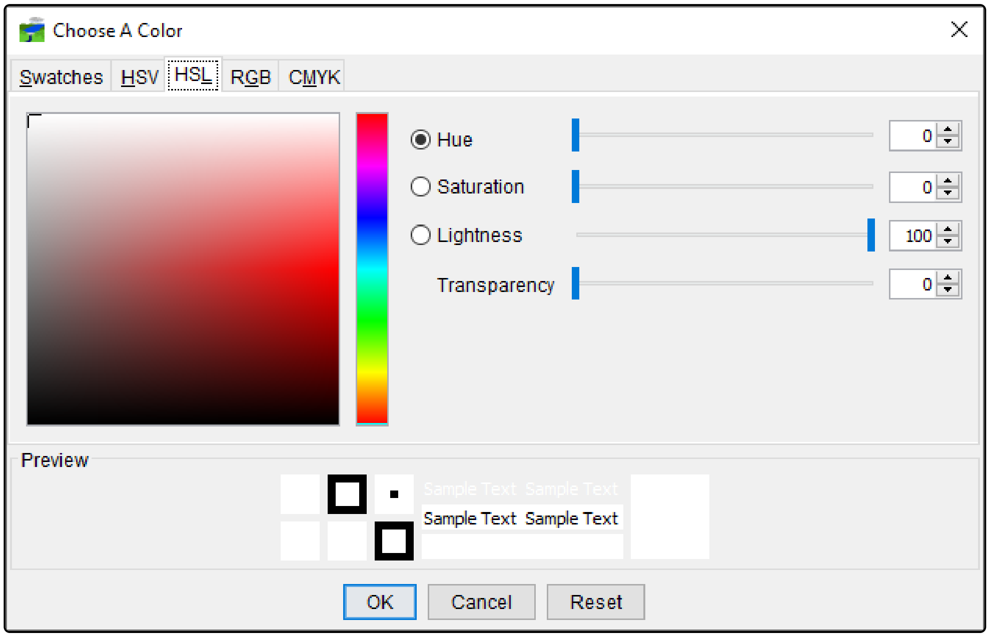
There are several ways to adjust the hue, saturation, value, and transparency of the colors. The slider bar, in conjunction with the **Hue**, **Saturation**, and **Value** options will change the selected color. Or you can directly enter a value for the **Hue**, **Saturation**, and **Value** cells. The following rules apply to the values you can enter for each cell:

- **Hue**: Hue is measured in a circle from zero (0) to 360 degrees as follows: 0 = red, 60 = yellow, 120 = green, 180 = cyan, 40 = blue, 300 = magenta.
- **Saturation**: Saturation is the amount of black, measured from zero (0) to one hundred (100) percent.
- **Value**: Is a measurement of the brightness of a color, measured from zero (0) to one hundred (100) percent. A value of zero (0) is considered black.

The final way to set a color on the **HSV** tab is from the color palette. Click and drag the crosshair symbol, which will automatically change the hue, saturation, and value items. Click **OK**, and the **Color Chooser** window will close. Depending on where you accessed the **Color Chooser** window, you will have to click either **Apply** or **OK** for the color change to appear in the display area.

28.3 HSL Tab

HSL is the Hue, Saturation, and Lightness color model, replaces the RGB (Red, Green, and Blue) color model in graphics and paint software applications. The **HSL** model attempts to resemble more perceptual color models such as the Natural Color System (NCS) or Munsell color system, placing fully saturated colors around a circle at a lightness value of .5, where a lightness value of zero (0) or one is fully black or white, respectively. From the **Color Chooser** window, select the **HSV** tab; and the **HSV** tab (Figure 1) becomes available.



152 FIgure 1 Color Chooser - HSL Tab

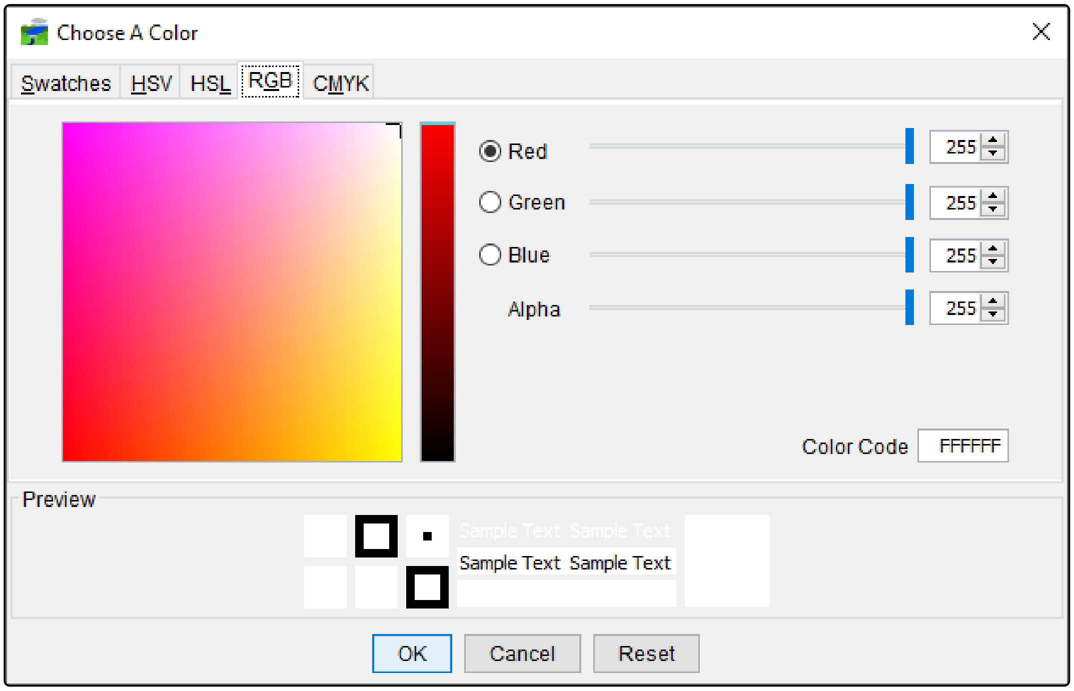
There are several ways to adjust the hue, saturation, lightness, and transparency of the colors. The slider bar, in conjunction with the **Hue**, **Saturation**, and **Lightness** options will change the selected color. Or you can enter directly a value for the **Hue**, **Saturation**, and **Lightness** cells. The following rules apply to the values you can enter for each cell:

- **Hue**: Hue is measured in a circle from zero (0) to 360 degrees as follows: 0 = red, 60 = yellow, 120 = green, 180 = cyan, 40 = blue, 300 = magenta.
- **Saturation**: Saturation is the amount of black, measured from zero (0) to one hundred (100) percent.
- **Lightness**: Is a measurement of the brightness of a color, measured from zero (0) to one hundred (100) percent. A value of zero (0) is considered black.

The final way to set a color on the **HSL** tab is from the color palette. Click and drag the crosshair symbol, which will automatically change the hue, saturation, and lightness items. Click **OK**, and the **Color Chooser** window will close. Depending on where you accessed the **Color Chooser** window, you will have to click either **Apply** or **OK** for the color change to appear in the display area.

28.4 RGB Tab

RGB is the Red, Green, and Blue color model, which allows you to set the red, green, and blue values of the colors you wish to produce. The **RGB** color model is an "additive" model. When 100 percent of each color is mixed together, it creates white light. From the **Color Chooser** window, select the **RGB** tab; and the **RGB** tab (Figure 1) becomes available.

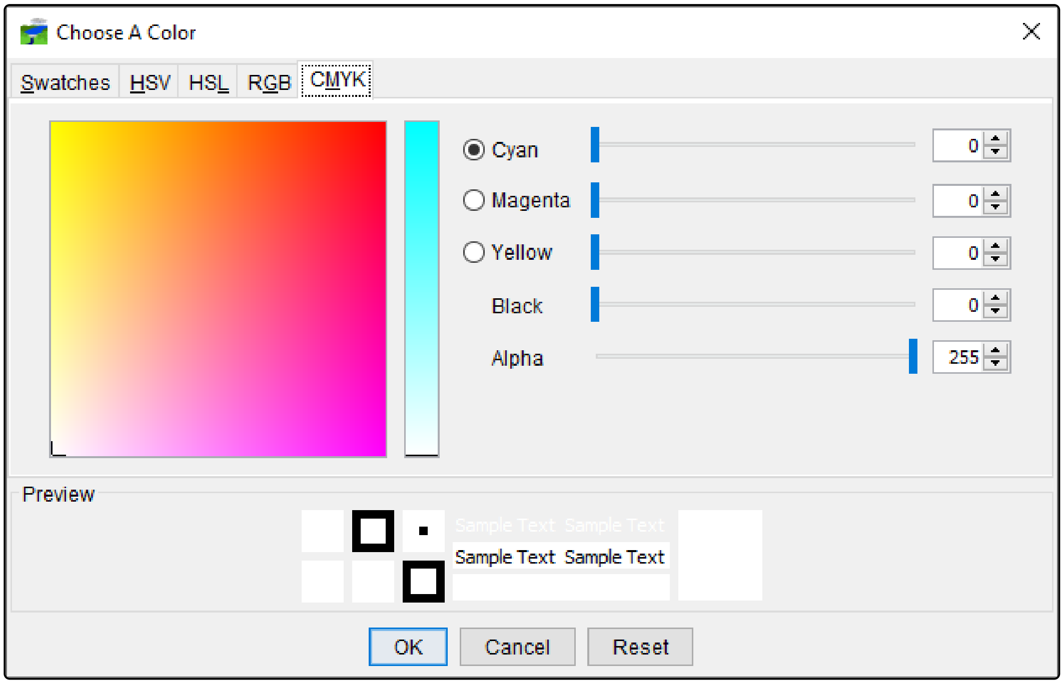


153 Figure 1 Color Chooser - RGB Tab

There are two ways to adjust the red, green, blue, and alpha value of colors. There is a slider bar for **Red**, **Green**, **Blue**, and **Alpha**. **Alpha** controls the opacity of the color. As you change the slider bar for each one, in the **Preview** field, the color you are producing is shown. Another way to affect the color is you can directly enter a value in the **Red**, **Green**, **Blue**, and **Alpha** cells. Click **OK**, and the **Color Chooser** window will close. Depending on where you accessed the **Color Chooser** window, you will have to click either **Apply** or **OK** for the color change to appear in the display area.

28.5 CMYK Tab

CMYK is the Cyan, Magenta, Yellow, and Black (key) color model, which are the four colors of ink used in the traditional method of printing hardcopies of images, call offset printing. The **CMYK** color model is a "subtractive" model. When 100 percent of each color is mixed together, it creates black light. From the **Color Chooser** window, select the **CMYK** tab; and the **CMYK** tab (Figure 1) becomes available.



154 Figure 1 Color Chooser - CMYK Tab

There are two ways to adjust the cyan, magenta, yellow, black, alpha value of colors. There is a slider bar for **Cyan**, **Magenta**, **Yellow**, **Black**, and **Alpha**. **Alpha** controls the opacity of the color. As you change the slider bar for each one, in the **Preview** field, the color you are producing is shown. Another way to affect the color is you can directly enter a value in the **Cyan**, **Magenta**, **Yellow**, **Black**, and **Alpha** cells. Click **OK**, and the **Color Chooser** window will close. Depending on where you accessed the **Color Chooser** window, you will have to click either **Apply** or **OK** for the color change to appear in the display area.

29 Using HEC-DSSVue

The HEC-DSSVue (Data Storage System Visual Utility Engine) software application is a graphical user interface for viewing, editing, and manipulating data in DSS files. With HEC-DSSVue, data may be plotted, tabulated, edited, and manipulated with over fifty mathematical functions. Along with these functions, HEC-DSSVue provides several utility functions, such as entering data sets into a database, renaming data set names, copying data sets to other database files, and deleting data sets.

In HEC-RTS, HEC-DSSVue will be used mainly as a viewing tool for data. This section will review some of the general functions in HEC-DSSVue (for further detail, see the HEC-DSSVue User's Manual).

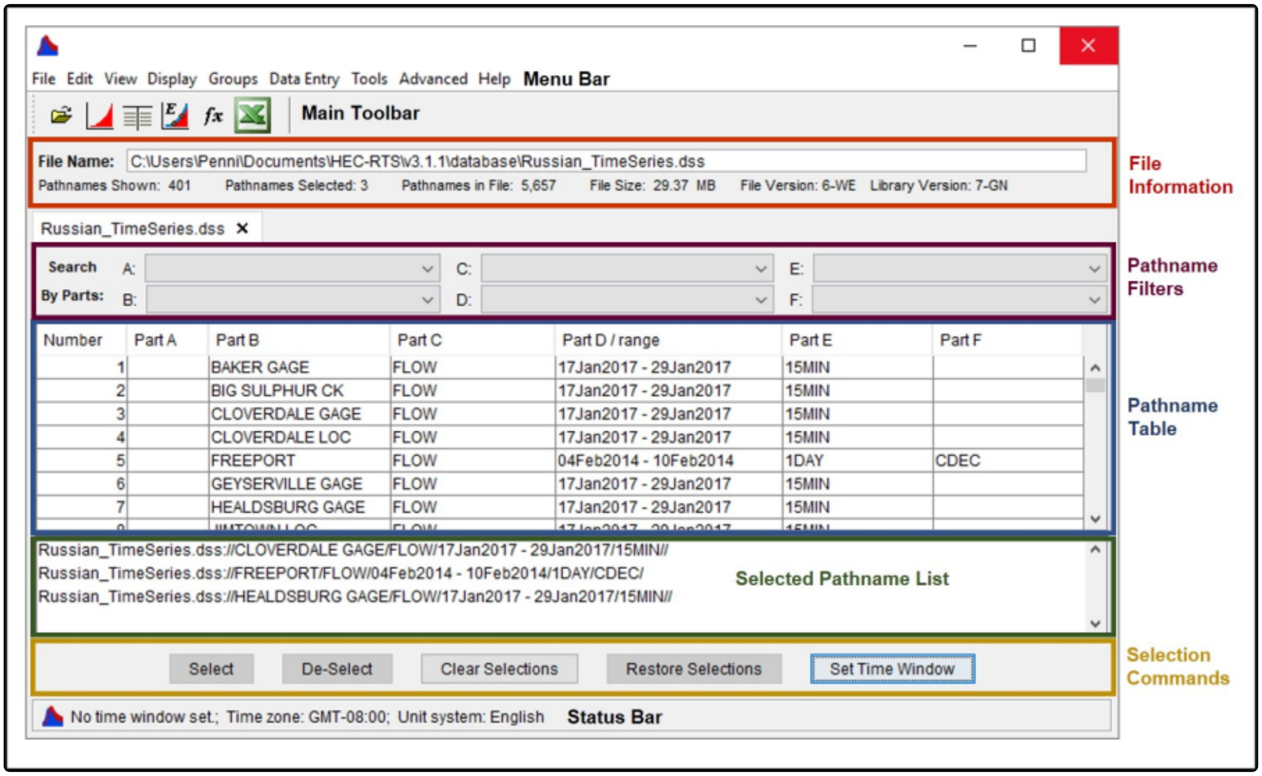
29.1 HEC Data Storage System (DSS) Concepts

DSS is a database system that was designed to efficiently store and retrieve scientific data that is typically sequential. Such data types include, but are not limited to, time series data, curve data, spatial-oriented gridded data, textual data, and others. The system was designed to make it easy for users and application programs to retrieve and store data.

DSS data is stored in blocks, or records, within a file, and each record is identified by a unique name called a *pathname*. The **pathname** is formulated in a way that makes it easy to group related data together. Software from HEC can read from and write to DSS files. This capability facilitates data exchange among the HEC software applications. For example, you could use HEC-ResSim to generate the time series data input for HEC-FIA.

29.2 Overview of HEC-DSSVue in HEC-RTS

From the HEC-RTS main window, from the **Tools** menu, click **HEC-DSSVue**, the **HEC-DSSVue** dialog will open (Figure 1). The **Menu Bar** contains the menus for HEC-DSSVue, following is a description of the available menus.



155 Figure 1 Figure F.1 HEC-DSSVue Dialog

- **File -** this menu is used for file management and contains the following: **New Vers 6, New Vers 7**, **Open**, **Close DSS File(s)**, **Print Catalog Preview**, **Print Catalog**, a list of most *recently opened files*, and **Close**.
- **Edit -** this menu allows you to edit data in a tabular format or graphically thru a graphical editor, and this menu also provides options for manipulating records in a DSS file: renaming records, deleting records, undeleting records, duplicating records, copying records to another DSS file, and merging of DSS files. Also, from this menu, you can select all the pathnames contained in the file. The menu contains the following commands: **Tabular Edit**, **Graphical Edit**, **Select All**, **Rename Records**, **Delete Records**, **Undelete**, **Duplicate**, **Copy To**, **Merge (copy)** ***dss filename*** **into**, and **Edit in Excel**.
- **View** - this menu provides options for viewing pathnames. Available commands are: **Pathname List**, **Pathname Parts**, **Condensed Catalog**, **Condensed – Group Collections**, **No Pathnames**, **Unsorted List**, **Search pathnames by string**, **Search pathnames by parts**, **Catalog using wild characters**, **Refresh Catalog**, **Unit System**, and **Time Zone**.
- **Display** - after you have selected a pathname, this menu gives you the ability to display the data for that pathname. Available commands are: **Plot**, **Plot Water Quality Profile**, **Plot Individual Data Sets**, **Tabulate**, **Display Data Options**, **Supplemental Information**, **Time Window**, **Charts**, **Tabulate in Excel**, **Set Times for Sync**, and **Plot Synched Data Sets**.
- **Groups** - this menu provides management and commands for groups of data. Available commands are: **Save Selected**, **Get**, **Get (add to list)**, **Plot**, **Plot Individual Sets**, **Tabulate**, **Math**, and **Manage**.
- **Data Entry** - this menu provides options for entering and adding data. Available commands are: **Manual Time Series**, **Manual Paired Data**, **Manual Text**, **Import**, and **Export**.
- **Tools** - this menu provides access tools for math operations, comparisons, and scripts. Available commands are: **Math Functions**, **Compare**, **Search for Value**, **Check File Integrity**, **Squeeze**, **Convert to DSS Version 7**, **Script Editor**, and **Script Selector**.

- **Advanced** - this menu provides options for manipulating records in a DSS file. Available commands are: **Catalog to File**, **Output**, **Debug**, and **Program Options**.
- **Help** - displays current version information about HEC-DSSVue.

The **Toolbar** (Figure 1) contains buttons that provide you with the capability to open a DSS file, plot the data, edit the data in a table, edit the data from a graphical editor, and apply math functions to the data. These buttons provide a shortcut to the respective commands in the **Display** menu.

 **Open File** - an **Open HEC-DSS File** browser will open, which allows you to select a DSS file.

 **Plot** - plots the data of the selected pathname(s).

 **Tabulate** - displays the data of the selected pathname(s) in a tabular form.

 **Graphical Edit** - displays the data of the selected pathname(s) in a graphical editor.

 **Math Functions** - allows the user to perform math functions on the data of the selected pathname(s) from the **Math Functions** dialog.

 **Tabulate in MS Excel** – any pathnames in the **Select Pathname List** (Figure 1) are displayed in Microsoft Excel®.

The **File Information** panel (Figure 1) displays the DSS file and path of the opened file, and statistics about the pathnames in the file and the size of the file or number of datasets in a DSS file.

The **Pathname Filters** (Figure 1) are used to sort pathnames by parts. There is a filter list for each individual part, A through F for the DSS records. The **Pathname Table** (Figure 1) displays the filtered dataset names contained in the DSS file. The pathnames are displayed by individual parts. The **Selected Pathname List** (Figure 1) displays the pathnames that you selected from the **Pathname Table**.

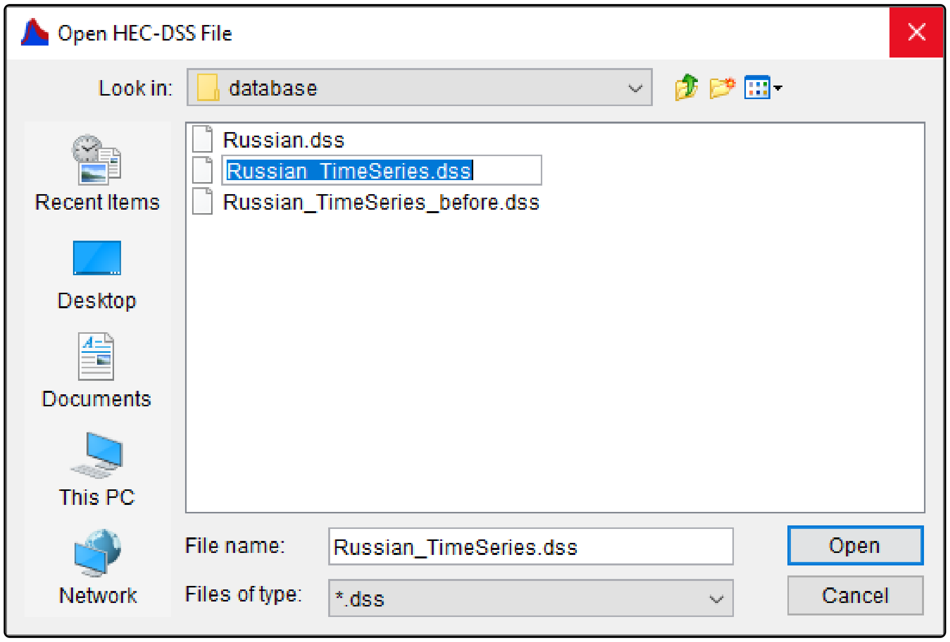
The **Selection Commands** (Figure 1) are available after you have selected pathnames you wish to view or manipulate. Available commands are **Select, De-Select, Clear Selections, Restore Selections**, and **Set Time Window**. These commands will be detailed later in this appendix.

When you have entered time window information, the **Status Bar** (Figure 1) will display the time window information. A time window is where you give a specific time frame for viewing or manipulating data.

29.3 Opening a DSS File

There are several ways to open a DSS file from HEC-DSSVue. One way is to type in the location and filename of the DSS file in the **File Name** field of the **HEC-DSSVue** main window (HEC-DSSVue Dialog). After entering the correct information, click **Enter**, the **Pathname Table** will display the pathnames of the selected DSS file, and the **File Information** (HEC-DSSVue Dialog) field will display statistics about the DSS file.

Another way to open a DSS file is from the **File** menu, click **Open**, the **Open HEC-DSS File** browser will open (Figure 1). Browse to the DSS file you wish to use, click on the filename. The filename will display in the **File name** box (Figure 1), click **Open**. The **Open HEC-DSS File** browser will close, and the pathnames associated with the DSS file will display in the **Pathname Table** (Figure F.1HEC-DSSVue Dialog).



156 Figure 1 Open HEC-DSS File Browser

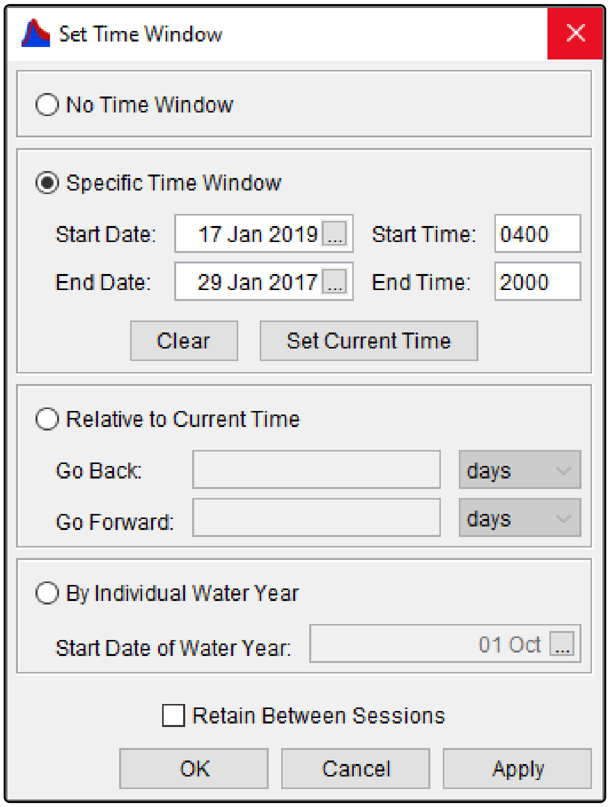
Another way to open a DSS file is from the **Main Toolbar**, click  , the **Open HEC-DSS File** browser will open (Figure 1). Browse to the DSS file you wish to use, click on the filename. The filename will display in the **File name** box (Figure 1), click **Open**. The **Open HEC-DSS File** browser will close, and the pathnames associated with the DSS file will display in the **Pathname Table** (HEC-DSSVue Dialog).

29.4 Set Time Window in HEC-DSSVue

To view the data for a certain block of time, you can set a time window:

1. Click **Set Time Window**, and the **Set Time Window** dialog opens (Figure 1). By default, the time window is set to **No Time Window**. You have the option of setting an exact time window or a time window that is relative to the

current time.



2. To enter a specific time frame, select **Specific Time Window**. Enter a **Start Date** and **Start Time**, in the format ddmmmyyyy (i.e., *17Jan2017*). Do the same for the **End Date** and **End Time**.
3. You can clear all the window fields by clicking **Clear**. To use the current date and time, click **Set Current Time** and all the time window fields will be filled. If needed, edit the ending date and time.
4. To set the time frame relative to the current time, select **Relative to Current Time**. Enter the number of time increments to **Go Back** or **Go Forward** from the current time. Then select the time step increment to **Go Back** and **Go Forward** from the current date in the drop-down lists. Your choices are **minutes, hours, days, months,** or **years**.
5. To set the time frame based on water year, select **By Individual Water Year** (Figure 1), and enter the **Start Date of Water Year** in the format dd mmm (i.e., *01 Oct*). Alternately, you can click to use a calendar tool for selecting the start date of the water year.
6. Click **OK**, the **Set Time Window** dialog closes (Figure 1). The time window is set, and it is displayed in the **Status Bar** of the HEC-DSSVue main window.

29.5 DSS Pathnames

Data is stored in blocks or records, within a file, and each record is identified by a unique name called a **pathname**. Each time that data is stored or retrieved from the file, its pathname must be given. The pathname is the key to the data's location in the database. A pathname is analyzed by DSS to determine an index number. This index determines where the dataset is stored within the database. The design ensures that very few disk accesses are made to retrieve or store datasets. One dataset is not related to another, so there is no need to update other areas of the database when a new dataset is stored.

DSS references records (datasets) by a pathname. The pathname of a DSS record can be three hundred and ninety-one (391) characters or less, and is, by convention, separated into **six parts**. The parts can be sixty-four (64) characters or less and are delimited by slashes in the pathname. The pathname parts are designated **A** through **F**. Table 1 provides an example naming convention for regular-interval time series:

A pathname may consist of up to 391 characters and is, by convention, separated into six parts, which may be up to sixty-four characters each. Pathnames are automatically translated into all upper-case characters. They are separated into six parts (delimited by slashes "/") labeled "A" through "F," as follows: /A/B/C/D/E/F/. Table 1 provides an example naming convention for regular-interval time series. A typical regular-interval time series might be: */RED RIVER/BEND MARINA/FLOW/01JAN1995/1DAY/OBS/*.

Table 1 DSS Pathname Parts

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

29.5.1 Viewing DSS Pathnames

The display of DSS pathnames in the **Pathname Table** (HEC-DSSVue Dialog) shows all the pathnames by their different parts. This display can be changed from the **View** menu. Available commands are **Pathname List, Pathname Parts, Condensed Catalog**, **Condensed – Group Collections**, **No Pathnames**, and **Unsorted List**.

The **Pathname List** command (Figure 1) displays the pathnames in the **Pathname Table**, while **No Pathnames** displays nothing. The **Pathname Parts** command (Figure 2) is the default view. **Condensed Catalog** is like the **Pathname Parts** command, but Part D displays a date span for each DSS pathname. The **Unsorted List** command displays the same as the **Pathname List** command, but you can automatically search pathnames by a string.

Number	Pathname
1	//BAKER GAGE/FLOW/01JAN2017/15MIN//
2	//BIG SULPHUR CK/FLOW/01JAN2017/15MIN//
3	//CLOVERDALE GAGE/FLOW/01JAN2017/15MIN//
4	//CLOVERDALE LOC/FLOW/01JAN2017/15MIN//
5	//FREEPORT/FLOW/01JAN2014/1DAY/CDEC/
6	//GEYSERVILLE GAGE/FLOW/01JAN2017/15MIN//
7	//HEALDSBURG GAGE/FLOW/01JAN2017/15MIN//

157 Figure 1 Pathname List

Number	Part A	Part B	Part C	Part D	Part E	Part F
1	RUSSIAN R	CLOVERDALE CA	DCP BATTERY VOLTAGE	06NOV2016	IR-DAY	USGS
2	RUSSIAN R	CLOVERDALE CA	DCP BATTERY VOLTAGE	07NOV2016	IR-DAY	USGS
3	RUSSIAN R	CLOVERDALE CA	DCP BATTERY VOLTAGE	08NOV2016	IR-DAY	USGS
4	RUSSIAN R	CLOVERDALE CA	DCP BATTERY VOLTAGE	09NOV2016	IR-DAY	USGS
5	RUSSIAN R	CLOVERDALE CA	DCP BATTERY VOLTAGE	10NOV2016	IR-DAY	USGS
6	RUSSIAN R	CLOVERDALE CA	DCP BATTERY VOLTAGE	11NOV2016	IR-DAY	USGS

158 Figure 2 Pathname Parts (default)

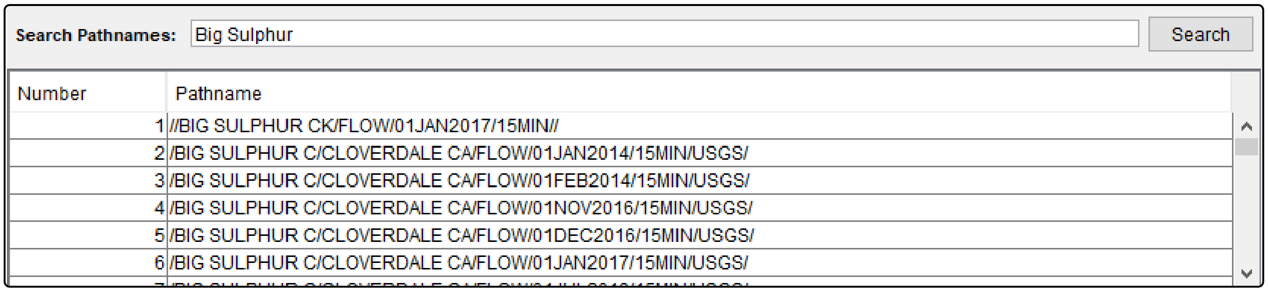
29.5.2 Search DSS Pathnames by Parts

Another way to change the display of the DSS pathnames is by searching pathnames by pathname parts. By default, you can search DSS pathnames by parts (**Search pathnames by part**). To search by pathname parts, from the **Pathname Filters** (Figure 3), decide which DSS pathname parts will be used. There is a **Filter List** for each individual pathname part. An example is shown in Figure 3, where DSS filters are used to see all the STAGE records that are were acquired from *USGS*. From the **Filter C** list, select *STAGE*, and from the **Filter F** list, select *USGS*. The **Pathname Table** displays only the pathnames that fit the criteria selected from the **Filter Lists**.

Search By Parts: A: B: C: STAGE D: E: F: USGS

Number	Part A	Part B	Part C	Part D	Part E	Part F
13	AUSTIN C	CAZADERO CA	STAGE	01AUG2019	15MIN	USGS
14	AUSTIN C	CAZADERO CA	STAGE	01SEP2019	15MIN	USGS
15	AUSTIN C	CAZADERO CA	STAGE	01OCT2019	15MIN	USGS
16	BIG SULPHUR C	CLOVERDALE CA	STAGE	01FEB2014	15MIN	USGS
17	BIG SULPHUR C	CLOVERDALE CA	STAGE	01NOV2016	15MIN	USGS
18	BIG SULPHUR C	CLOVERDALE CA	STAGE	01DEC2016	15MIN	USGS

Another way to search pathnames is by providing a "string for the search. From the **View** menu, click **Search pathnames by string**. The pathname filters are replaced with a **Search Pathnames** box (Figure 4), and a **Search** button. For example, in Figure 4, in the **Search Pathnames** box, *Big Sulphur* has been entered. Click **Search**, the **Pathname Table** displays all records that contain Big Sulphur. Further details for this method are described in the HEC-DSSVue User's Manual.



159 Figure 4 Search Pathnames by String

29.5.3 Selecting Pathnames

With the appropriate list of pathnames in the **Pathname Table** (HEC-DSSVue Dialog) select which pathnames to view or manipulate. To select a pathname, double-click on a pathname in the **Pathname Table**, and the selected pathname will appear in the **Selected Pathname List** (HEC-DSSVue Dialog). Alternatively, click on a pathname in the **Pathname Table**, and then click **Select**.

To select multiple pathnames, in the **Pathname Table**, click on a pathname the pathname will be highlighted. Find the next pathname to include in a group, hold down the **SHIFT** key and click. The block of pathnames will be highlighted. Click **Select**, and all the pathnames within that block will be added to the **Selected Pathname List**. Another way to select multiple pathnames is from the **Pathname Table**. Click and drag across the pathnames, release the mouse button and the block of pathnames will be highlighted. Click **Select**, and all the pathnames within that block will be added to the **Selected Pathname List**.

Individual pathnames can be selected to be in a group for selection. From the **Pathname Table**, select a pathname which will be highlighted. To select another not in a contiguous group, hold down the **CTRL** key and select the next pathname which will be highlighted. Continue to do this until all of the desired pathnames have been selected. Release the **CTRL** key and click **Select**. All the selected pathnames will be added to the **Selected Pathname List**.
You can select all pathnames displayed in the **Pathname Table**, from the **Edit** menu, click **Select All**. All of the pathnames available from the selected DSS file will now display in the **Selected Pathname List** (HEC-DSSVue Dialog).

29.5.4 Clearing Selected Pathnames

To clear a pathname from the **Selected Pathname List** (HEC-DSSVue Dialog), select a pathname and click **De-Select**. The pathname will be cleared from the **Selected Pathname List**. To clear all pathnames from the **Selected Pathname List**, click **Clear Selections**.

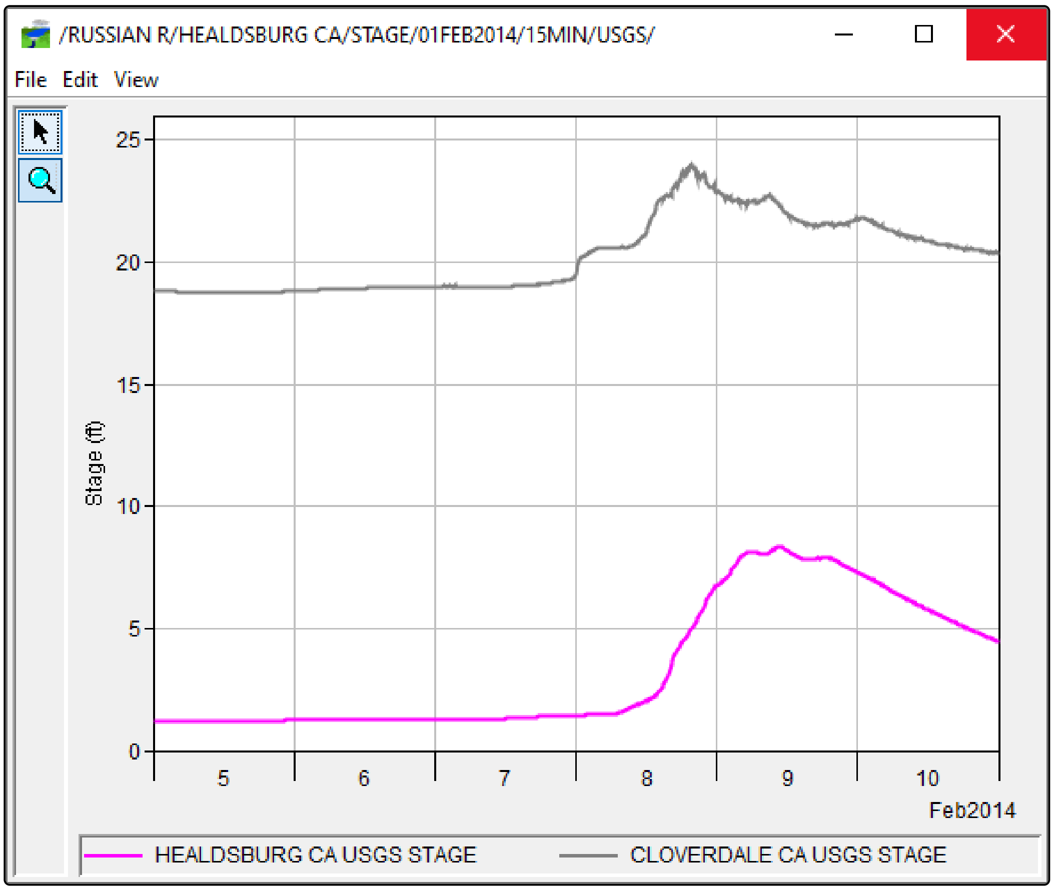
Groups of pathnames can be cleared by using the same concepts described in selecting pathnames by using the **SHIFT** and **CTRL** keys. Once pathnames have been cleared from the **Selected Pathname List**, the pathnames can be restored to the **Selected Pathname List** by clicking **Restore Selections**.

29.6 Visualizing Data in HEC-DSSVue

After a pathname or multiple pathnames have been selected, the data associated with the pathname can be visualized. This can be done through a graphical means - plots, or through a tabular form - tabulate. Further details on plots and tables are discussed in Printing and Copying HEC-RTS Data.

29.6.1 Plots

There are two ways to view plots from the HEC-DSSVue main window. From the **Display** menu, click **Plot**, a **Plot** window will open (Figure 1). The plot window is displaying the pathnames in the **Selected Pathname List** (HEC-DSSVue Dialog). Another way to display a plot, is from the **Main Toolbar** (HEC-DSSVue Dialog), click , a **Plot** window will open (Figure 1).



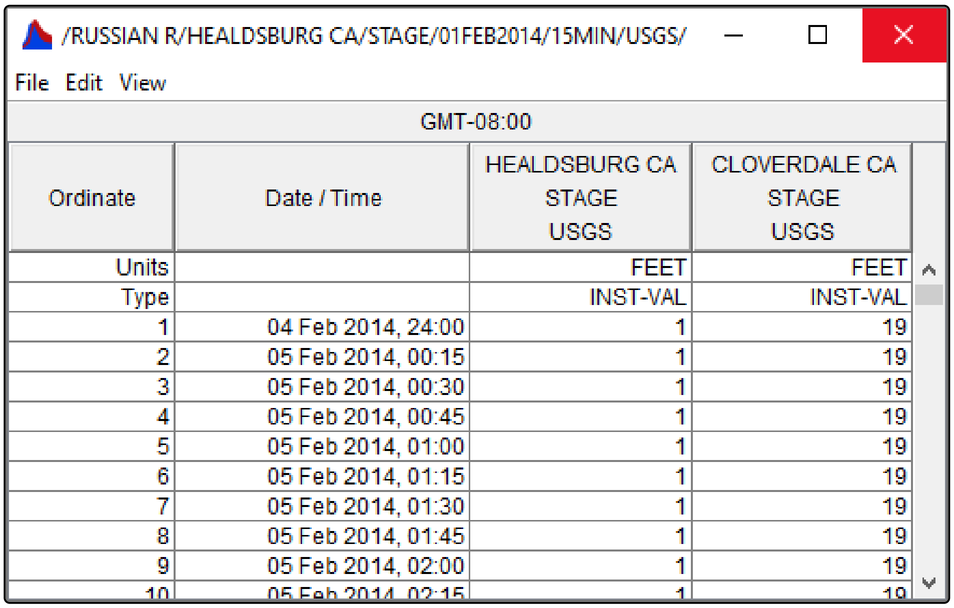
160 Figure 1 HEC-DSSVue Plot Window

The parameters for the plots are labeled and color-coded, date ranges are specified along one axis, and the units of measure are specified along the other axis. The **Plot** window (Figure 1) can be resized by grabbing the corner of the window and dragging it to the desired size.

Areas of the plot can also be zoomed in on. From the **Plot** window (Figure 1), click **(Zoom Tool**); find the area on the plot to zoom in on and click and drag over the area. The plot will refresh, and the area within the zoom rectangle will fill the plot window. To *zoom out*, with the **Zoom Tool** selected, right-click anywhere on the plot area.

29.6.2 Table

There are two ways to view tabular results from the HEC-DSSVue main window. From the **Display** menu, click **Tabulate**, a **Table** window will open (Figure 2). The table window is displaying the pathnames in the **Selected Pathname List** (HEC-DSSVue Dialog). Another way to display a tabular results, is from the **Main Toolbar** (HEC-DSSVue Dialog), click  , a **Table** window will open (Figure 2).



Ordinate	Date / Time	HEALDSBURG CA STAGE USGS	CLOVERDALE CA STAGE USGS
Units		FEET	FEET
Type		INST-VAL	INST-VAL
1	04 Feb 2014, 24:00	1	19
2	05 Feb 2014, 00:15	1	19
3	05 Feb 2014, 00:30	1	19
4	05 Feb 2014, 00:45	1	19
5	05 Feb 2014, 01:00	1	19
6	05 Feb 2014, 01:15	1	19
7	05 Feb 2014, 01:30	1	19
8	05 Feb 2014, 01:45	1	19
9	05 Feb 2014, 02:00	1	19
10	05 Feb 2014, 02:15	1	19

161 Figure 2 HEC-DSSVue Table Window

The **Tabule** window displays the ordinate (starting from the start date/time), the date and time stamp, and the values for the selected data sets. From the **Tabule** window, there are several options for displaying the tabular data (e.g., displaying commas in the number formatting, setting the precision of decimal places for your data).

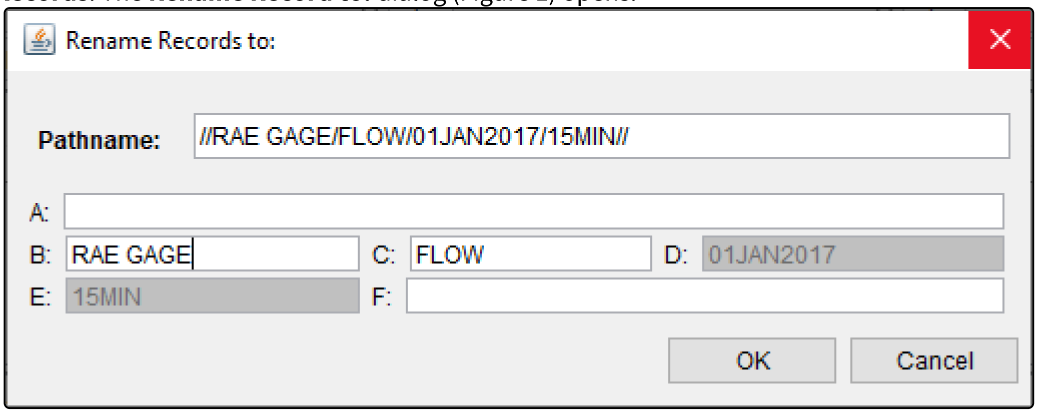
29.7 Data Management Functions

HEC-DSSVue provides several tools for manipulating DSS records. These tools provide the capability of creating DSS records - time series or paired data; renaming, duplicating, deleting, and undeleting records; copy records from one DSS file to another DSS file; merge two DSS files; perform maintenance; check the status of a DSS file; and, many more other data management functions. The following sections review a few of the available data management functions. For further details see the HEC-DSSVue User's Manual.

29.7.1 Renaming DSS Records

To change the pathname parts of a DSS pathname:

1. Select one or more pathnames, from the HEC-DSSVue main window, from the **Edit** menu, click **Rename Records**. The **Rename Record to:** dialog (Figure 1) opens.

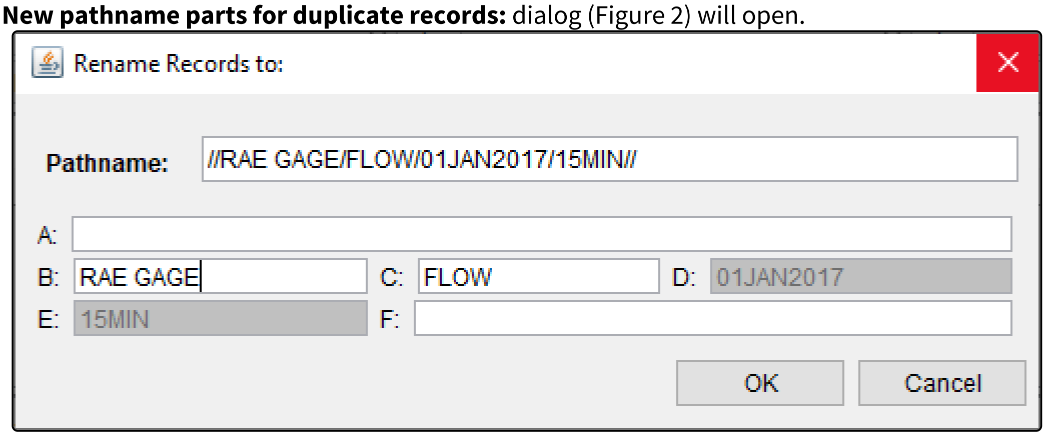


2. Each of the fields (**A** - **F**) represents the different pathname parts. Enter one or more of the new pathname parts in the appropriate field.
3. Click **OK**, and the **Rename Records to:** dialog closes (Figure 1). A message appears informing the user of the number of records renamed. The selected pathname is no longer displayed in the **Selected Pathname List** (HEC-DSSVue Dialog), and the pathname will display in the **Pathname Table** (HEC-DSSVue Dialog) with the new pathname parts.

29.7.2 Duplicating DSS Records

Duplicating a DSS record allows the user to make a copy of a record, but with a pathname part that has been changed. To duplicate records:

1. Select one or more pathnames. from the HEC-DSSVue main window, from the **Edit** menu, click **Duplicate**. The **New pathname parts for duplicate records:** dialog (Figure 2) will open.

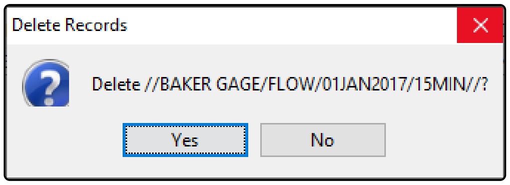


2. Each of the fields (**A** - **F**) represents the different pathname parts. Enter one or more of the new pathname parts in the appropriate field.
3. Click **OK**, and the **New pathname parts for duplicate records:** dialog closes (Figure 2). A message appears informing the user of the number of records that were duplicated. A new pathname is displayed in the **Pathname Table** (HEC-DSSVue Dialog).

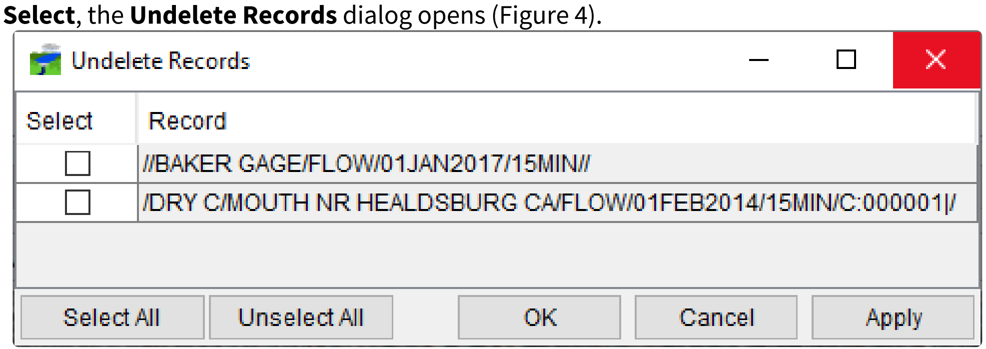
29.7.3 Deleting and Undeleting DSS Records

To delete DSS records:

1. Select one or more DSS pathnames, from the HEC-DSSVue main window, from the **Edit** menu, click **Delete Records**. A **Delete Records** window will open (Figure 3) asking you to confirm the deletion. Click **Yes**, the **Delete Records** window will close.



2. An **HEC-DSSVue** message window opens informing the user of the number of records that were deleted. Click **OK**, the **HEC-DSSVue** message window closes. The pathname is removed from the **Selected Pathname List** and the **Pathname List** (HEC-DSSVue Dialog).
3. If records were unintentionally deleted, before the next operation, from the **Edit** menu, point to **Undelete**, click **Select**, the **Undelete Records** dialog opens (Figure 4).



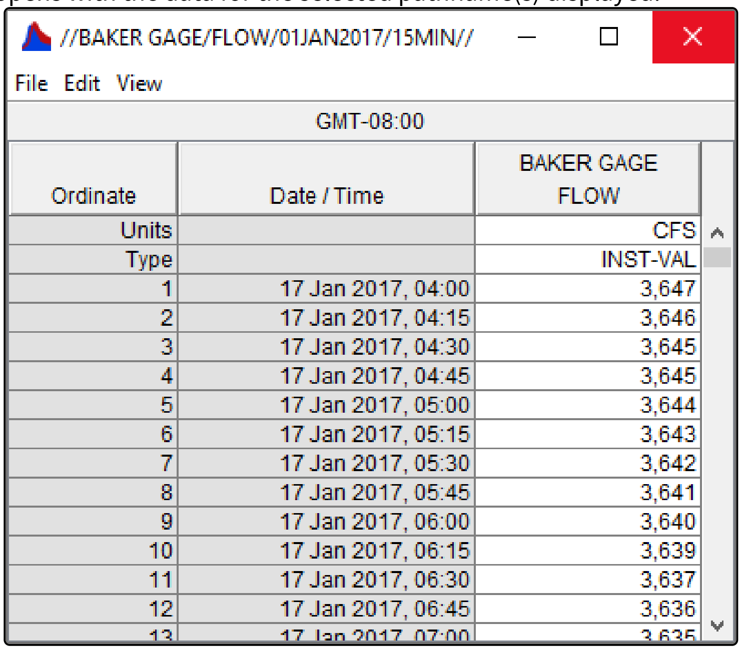
4. To select all the records displayed in the **Undelete Records** dialog, click **Select All**. To select individual records, under the **Select** column, click the checkbox that corresponds to the correct record.
5. To restore the selected records, either click **Apply** or **OK**. A message appears indicating that the records have been undeleted. Click **OK**, and the message closes, the **Undelete Records** dialog closes (Figure 4), and the records will be displayed in the **Selected Pathname List** and the **Pathname List** (HEC-DSSVue Dialog).
6. For further information on deleting and undeleting records in a DSS file, see the HEC-DSSVue User's Manual for further details.

29.8 Editing DSS Data

DSS data can be edited either in tabular form or from a graphical editor (with the appropriate permissions). To edit data in a tabular form:

1. Select one or more pathnames, the pathname(s) will display in the **Selected Pathname List** (HEC-DSSVue Dialog).

2. From the HEC-DSSVue main window, from the **Edit** menu, click **Tabular Edit**. A **Tabular Edit** dialog (Figure 1) opens with the data for the selected pathname(s) displayed.



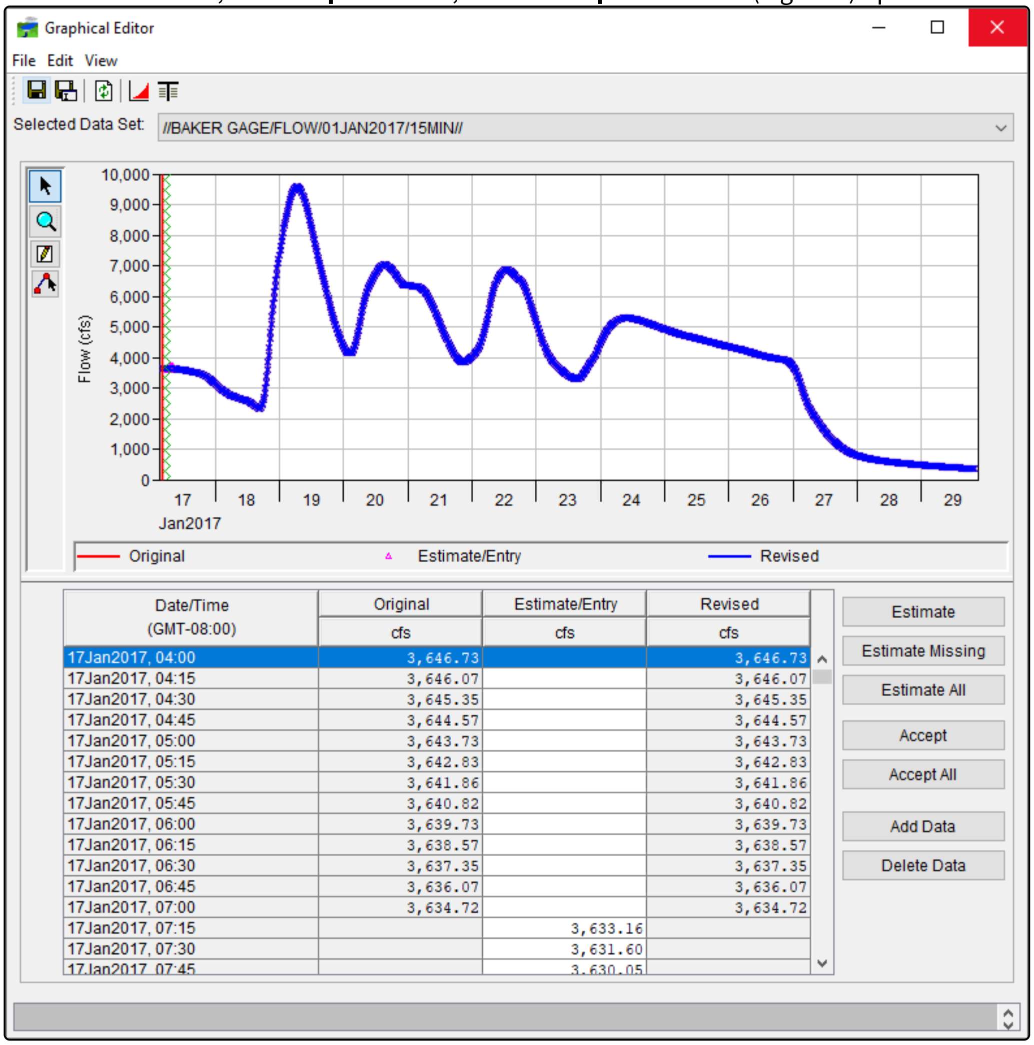
Ordinate	Date / Time	BAKER GAGE FLOW
Units		CFS
Type		INST-VAL
1	17 Jan 2017, 04:00	3,647
2	17 Jan 2017, 04:15	3,646
3	17 Jan 2017, 04:30	3,645
4	17 Jan 2017, 04:45	3,645
5	17 Jan 2017, 05:00	3,644
6	17 Jan 2017, 05:15	3,643
7	17 Jan 2017, 05:30	3,642
8	17 Jan 2017, 05:45	3,641
9	17 Jan 2017, 06:00	3,640
10	17 Jan 2017, 06:15	3,639
11	17 Jan 2017, 06:30	3,637
12	17 Jan 2017, 06:45	3,636
13	17 Jan 2017, 07:00	3,635

3. The columns in gray cannot be edited. The user can edit individual cells, insert (beginning of record), append (end of record), and delete rows (this command actual deletes the editable value of the selected row).
4. From the **File** menu, click **Save**. The **Saved Data Set** message appears, indicating that a record has been saved. Click **OK**, and the **Saved Data Set** message closes.
5. From the **File** menu, click **Close** and the **Tabular Edit** dialog (Figure 1) closes.
6. For further details on tabular editing see the HEC-DSSVue User's Manual.

To edit data using the graphical editor:

1. Select one or more pathnames, the pathname(s) will appear in the **Selected Pathname List** (HEC-DSSVue Dialog).

2. From the **Edit** menu, click **Graphical Edit**, and the **Graphical Editor** (Figure 2) opens.



3. The **Graphical Editor** allows the user to edit the data in a graphical mode. This editor is like the **Data Validation Editor** from the **Acquisition Module**. Review Data Validation Editor for similar functionality.
4. From the **File** menu, click **Save**, and the **Save Changes** message appears asking if the changes should be saved. Click **OK**, and the **Save Change**s message closes.
5. The **Saved Data Set** message opens indicating which record(s) was saved. Click **OK**, and the **Saved Data Set** message closes.
6. From the **File** menu, click **Close,** and the **Graphical Editor** (Figure 2) closes. For further details on graphical editing see the HEC-DSSVue User's Manual.

30 Scripting

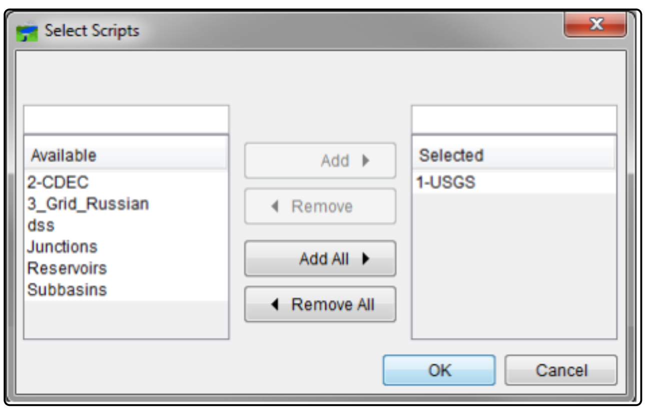
In HEC-RTS, scripts provide a way to automate the program for the purpose of executing a prescribed set of actions with a simple trigger. Scripting simplifies user operation, ensures repeatability and consistency of results, reduces time required to generate results, and permits schedules of an operation in the future. HEC-RTS uses the **Jython** scripting language. Jython is a special implementation of the **Python** scripting language that can interact with programs written in Java.

Some uses of scripting in HEC-RTS:

- Running an existing forecast with one or more alternatives with a single button click (replacing many mouse clicks with one).
- Creating a new forecast based on the current time and running it with one or more alternatives.
- Scheduling an action such as creating a new forecast to happen sometime in the future (possibly on a recurring basis).
- Work with a mixture of DSS files.

30.1 Scripts

Scripts can be created for all HEC-RTS modules or scripts can be specific to a HEC-RTS module. A user can schedule a script for execution, and you can check on the status of a script. This is accomplished from the **Script Selector**, which is accessed from the **Scripts** menu, click **Editor**, the **Scripts Editor** will open for the module that you are in (Figure 1). For example, in Figure 1, the **Script Editor** was opened in the Visualization module.



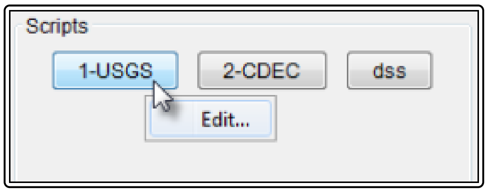
162 Figure 1 Select Scripts Dialog

30.1.1 Select Scripts

- **Available:** The Available column lists all the available scripts that can be added to the **Visualization Module**, Scripts box. Use the buttons **Add** or **Add All** to add script buttons to the watershed **Visualization Module**. The **Remove** or **Remove All** buttons will delete script buttons from the watershed **Visualization Module**.
- **Selected:** This column lists all script buttons that are or will be added to the **Visualization Module**, Scripts box.

30.1.2 Edit Existing Script Buttons

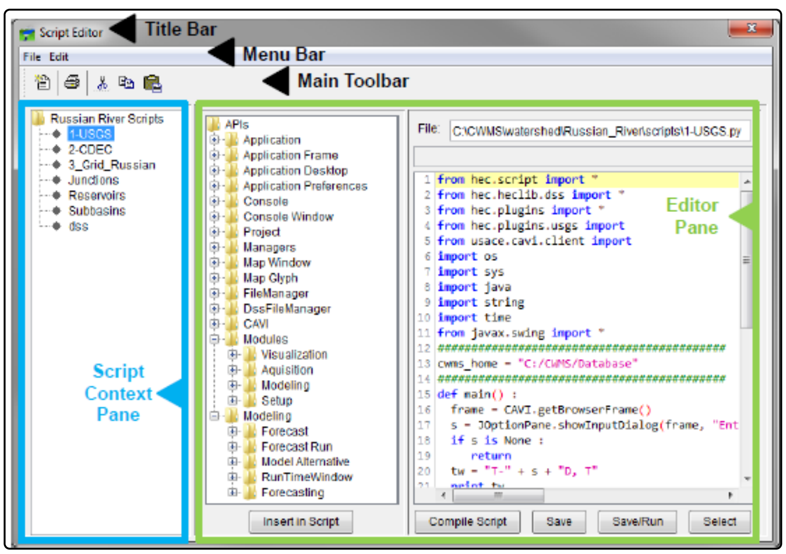
In the **Visualization Module**, after creating scripts and requesting that a button be created for that script, you can edit the script associated with the button, delete the script associated with the button, and schedule a job for that script. All tasks can be accomplished through the script button shortcut menu (Figure 2).



163 Figure 2 Script Button Shortcut Menu

30.2 Script Editor

You create, edit, and delete scripts using the **Script Editor**. From the **Tools** menu click **Script Editor,** and the **Script Editor** will open (Figure 1). Components of the **Script Editor** include the **Menu Bar**, the **Script Context Pane,** the **API Pane,** and the **Editor Pane**. The following sections describe these components.



164 Figure 1 Script Editor

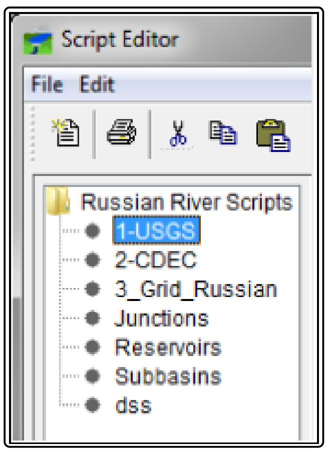
30.2.1 Menu Bar

An overview of the **Script Editor's** menu bar:

- **File:** From this menu, you create and import scripts, open, delete, print, save, and run scripts, or make a copy of the script. Available commands are **New, Open, Run, Delete, Save, Save As, Print,** and **Close**.
- **Edit:** From this menu, you can modify a script or search a script. Available commands are: **Cut, Copy, Paste, Select All,** and **Find**.

30.2.2 Script Context Pane

The **Script Context Pane** (Figure 2) of the **Script Editor** you can navigate to folders in the CWMS directory structure and access scripts. The tree structure allows you to navigate through the directory structure for scripts within CWMS. Scripts can be stored that are applicable to all CWMS modules or to individual CWMS modules. The name of the script is also the script name displayed on the associated script button in the **Script Selector**. To edit a script, double-click on the script name and the script opens in the **Editor Pane**.

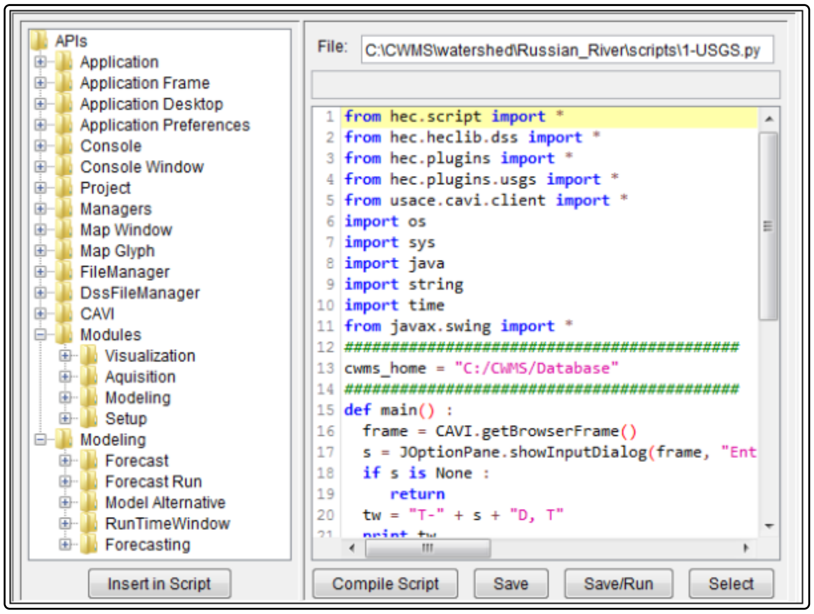


165 Figure 2 Script Context Pane

30.2.3 Editor Pane

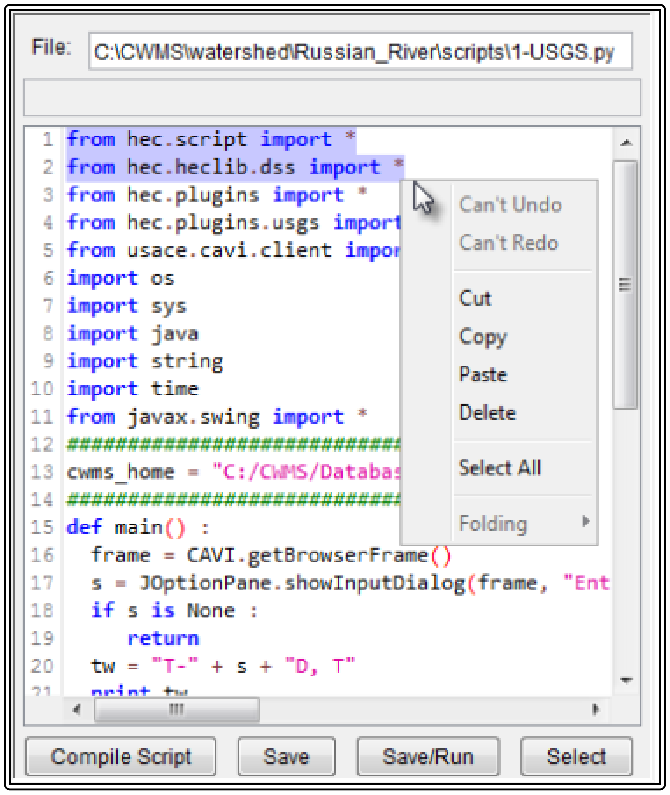
Scripts are modified in the **Editor Pane** (Figure 3) of the **Script Editor**. The name displayed on the associated script button in the **Script Selector** is the same as the name in the **Script Context Pane** and the ***.py* name (i.e. *1-USGS.py*) in the *File** box of the **Editor Pane**. The **File** box (Figure 3) also shows where the user can find the location of the associated file of the script. By default, a button for each script created will appear on the **Script Selector**. If the user wants an icon to appear on the script button, then from the **Icon** list, select an icon. If you do not want a button, you need to clear **Display Script to User**.

The **Test Arguments** field (Figure 3) allows you to initialize the values of variables used by the script. You can enter as many semicolon-separated variable assignments as your script requires (e.g., var1="abc"; var2=7.5) or leave the area blank.



166 Figure 3 Editor Pane

The **Description** field (Figure 3) allows you to enter a description of the script. The first line of your description serves as tooltip for the associated script button. The field below the **Description** field in the **Editor Pane** (Figure 3) is where you will enter the script text and serves as an editing window. This field will be referred to as the **Script Text** field. By right-clicking in the **Script Text** box, shortcut menu will appear (Figure 4) that contains the following commands:



167 Figure 4 Script Text Shortcut Menu

- **Cut:** Copies the currently selected script text to the system clipboard and removes it from the script.
- **Copy:** Copies the currently selected script test to the system clipboard and leaves it in the current script.
- **Paste:** Copies text from the system clipboard into the script at the current cursor location.
- **Select All:** Selects all the text in the script.

30.3 Create a Script

You create a script by selecting either the **General Scripts** directory or the **Watershed Directory** in the **Script Context Pane** of the **Script Editor**. From the **Script Editor**, from the **File** menu, click **New**. Enter the name of the script in the subsequent dialog.

30.4 Save a Script

In the **Script Context Pane**, select the script you wish to save. From the **File** menu, click **Save** or you can click on the **Save** button. You can also save and execute the script by simply clicking on the **Save/Run** button from the **Script Editor**. The **Editor Pane** will become grey and the name of the script will appear in the **Script Context Pane** in the appropriate place.

30.5 Delete a Script

In the **Script Context Pane**, select the script you wish to delete. From the **File** menu, click **Delete**. A **Confirm Deletion** message will appear asking you if really want to delete the selected script, click **Yes**. The confirmation message will close, and the script will no longer appear in the **Script Context Pane**.

30.6 Edit a Script

There are three ways to edit a script from the **Script Editor**. When editing is enabled, the **Editor Pane** will change from grey to white. From the **Script Context Pane**, double-click the script. From the shortcut menu, click **Edit Script**. From the **Script Editor**, click the **Edit** button.

Another way to edit a script is to select a script from the **Script Context Pane**, and from the **File** menu, click **Open**.

30.7 Running a Script

30.7.1 Manually Running a Script

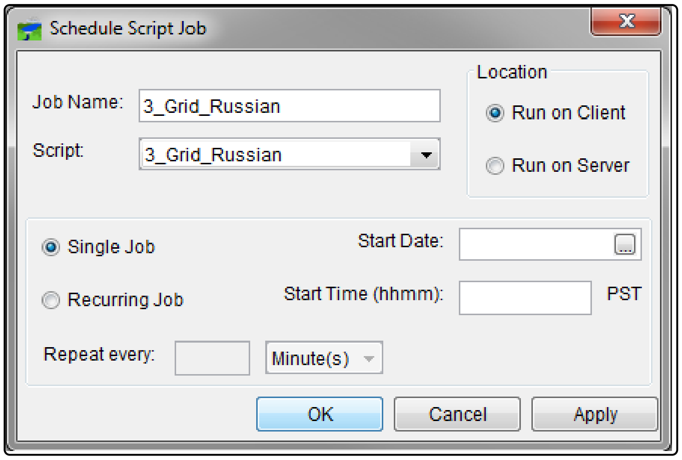
There are three ways to manually run a script: the first way is from the **Script Selector** (Scripts). From the **Script Selector**, select the script you wish to run, and click **Run**.

The second way is to manually run a script is from the Script Editor. From the **Script Editor**, from the **Script Context Pane**, select a script. From the **File** menu, click **Run**, and the script will execute.

The last way to trigger a script is to assign the script to a **Time Series Icon** and trigger the run through the icon's shortcut menu. See Time Series Icons for more details.

30.7.2 Automatically Running a Script

Running scripts automatically in CWMS is accomplished by scheduling script jobs. From the **Script Selector**, from the **Script** menu, click **Schedule Script Job**. The **Schedule Script Job** dialog will open (Figure 1). Another way to launch the **Schedule Script Job** dialog is from the **Script Selector** dialog: Right-click on a script button, and from the shortcut menu, click **Schedule Script Job**.



168 Figure 1 Schedule Script Job Dialog

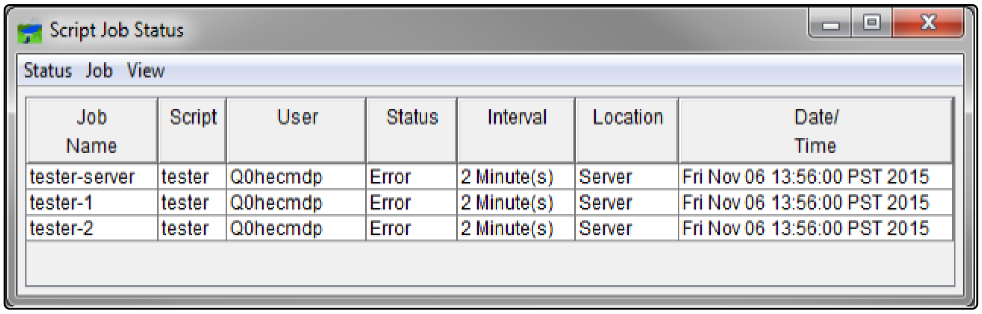
30.7.3 Schedule Script Job Dialog

To schedule a script, the CAVI client must be running and logged in as a valid CWMS user. For the script to run, that valid user must be logged in and running the CAVI. To setup a script to run automatically:

1. The **Job Name** field (Figure 1) contains the name of the job that will run the script. The job name contains the name of the script, and the start date and start time that you will enter. You can change this name, but it is recommended that you do not change this name.
2. From the **Script** name list (Figure 1), you can select the script you wish to run automatically.
3. In the **Location** box (Figure 1), you can schedule a script to run on either the server or the client.
4. Enter a starting date in the **Start Date** cell (i.e., DDMMMYYYY format), and a start time in the **Start Time** cell (Figure 1). The date and time reference are local time of the computer running the CAVI client (usually your workstation), as indicated the time zone ID next to the **Start Time** box.
5. The script that you are setting up can be run automatically one or be recurring. By default, the job is setup to run once - **Single Job** (Figure 1) is selected. If you want the job to be recurring, select **Recurring Job**, and enter the interval you wish the job to recur at in the **Recurrent Interva**l box, and from the list select **Minute(s), Hour(s)**, or **Day(s)**.
6. Once everything is setup, click **OK**, and a confirmation message will open telling you that the script has been scheduled. Click **OK**, and the confirmation message will close, and the **Schedule Script Job** dialog will close (Figure 1).

30.7.4 Checking the Status of a Scheduled Script

To check the status of an automatically schedule script, from the **Script Selector** dialog, from the **Script** menu, click **Status**; the **Script Job Status** dialog will open (Figure 2). The following gives an overview of what information is being provided to you:



169 Figure 2 Script Job Status Dialog

1. The **Job Name** column (Figure 2) displays the name assigned to the script job when it was initially scheduled.
2. The **Script** column (Figure 2) displays the name of the script that is executed with the scheduled job runs. A script can be used in more than one script job at a time.
3. The **User** column (Figure 2) displays the name of the valid CWMS user that was logged into the CAVI and scheduled the script.
4. The **Status** column (Figure 2) displays the status of the scheduler (not the script itself). This column will display three states:
   - **Pending:** The scheduler is waiting to start the script job for the first time.
   - **Successful:** The scheduler started the script job successfully.
   - **Error:** The scheduler did not start the script job successfully.
5. To see whether the job was a single execution or a recurring execution, the **Interval** column (Figure 2) displays this information.
6. From the **Location** column (Figure 2) you will know whether the job executed on the client or the server.
7. If the job is recurring, the **Date/Time** column (Figure 2) will display the date and time of the next scheduled execution.

30.7.5 Deleting a Scheduled Script

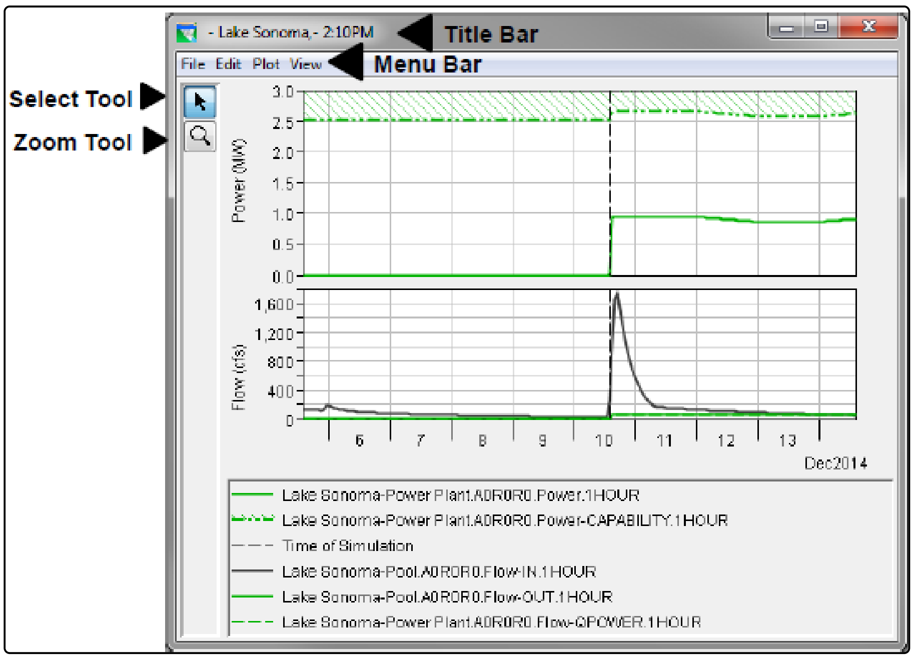
From the **Script Job Status** dialog box, from the table, select a schedule script. From the **Job** menu, click **Delete**; and a message will open asking you if you really want to delete this scheduled script. Click **Yes,** and the message will close, and the scheduled script will no longer appear in the table.

31 Plots

Plots are highly customizable and offer an array of information that will assist you with reviewing your data in HEC-RTS. This section covers some of the capabilities of plots within HEC-RTS. For more detail, see the HEC-DSSVue User's Manual.

31.1 Plot Window

The **Plot** window provides tools that facilitate the viewing of HEC-RTS data. The main plot window and the different components are illustrated in Figure 1. The **Title Bar** contains the name of the location that the data represents. For example, in Figure 1, the plot is illustrating the elevation and flow data at a reservoir location named *Savage*.



170 Figure 1 Plot Window

31.1.1 Menu Bar

Following is an overview of the **Menu Bar** (Figure 1) for the **Plot w**indow. The commands available from the menus will facilitate the viewing and customizing of plots.

- **File:** From this menu you can view the plotted data in a tabular format, save the plot as a graphics file, save a template, use a template, copy the plot to the clipboard, and print the plot. Available commands are: **Tabulate**, **Save As**, **Save Template**, **Apply Template**, **Save Specification**, **Copy to Clipboard**, **Print**, **Page Setup**, **Print Preview**, **Print Multiple**, and **Close**.
- **Edit:** From this menu you can customize the plot. Available commands are: **Plot Properties**, **Default Line Styles**, and **Default Plot Properties**.
- **Plot:** From this menu you can select variable to plot, open and save plot type. Available commands are: **Select Variables**, **Open Plot Type**, and **Save Plot Type**.

- **View:** From this menu you can have the plot window always on top of your desktop, and you can choose to use line styles. Available commands are: **Always On Top**, **Use Line Styles**, **Zoom to all**, **Legend Placement**, **Hide Legend**, **Refresh**, and **Live Display**.

31.1.2 Tools

The **Tools** appear in a toolbar on the left side of the **Plot** window (Figure 1). The tools change the appearance of the mouse, as well as the functionality of the mouse.

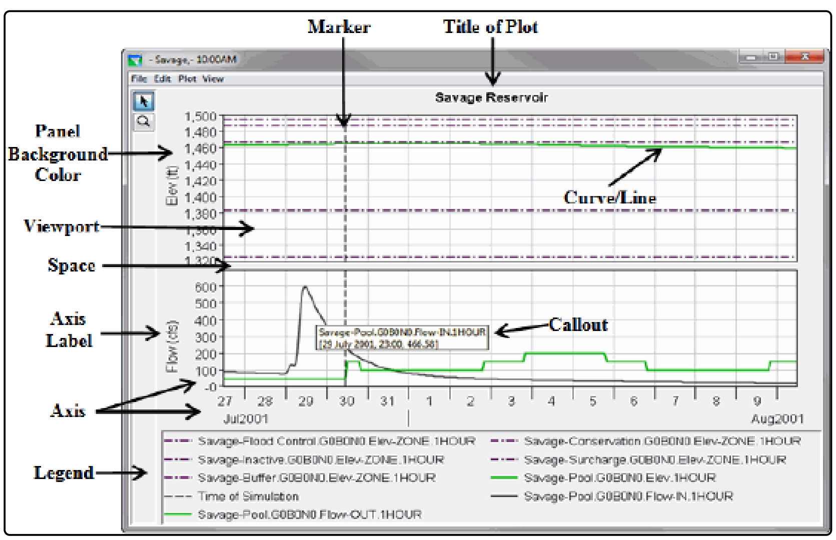
 **Pointer/Select Tool:** With the **Pointer/Select Tool**, you can access shortcut menus that allow you to customize features of your plots using the plot editing tools.

 **Zoom Tool:** The **Zoom Tool** allows you to view data closely at a specific time. To *zoom in*, hold the mouse button down and outline the area you want to enlarge. To *zoom out*, click the right mouse button (right-click). The zoom out is done by a factor of two and positions the clicked location at the center of the display area.

31.2 Customizing Plots

31.2.1 Overview

From the **Plot** window, several editors allow you to configure default properties for all plots, as well as customize individual plots. Figure 1 shows the features plot area that can be configured.



171 Figure 1 Elements of the Plot Window

- **Title of Plot:** You can add a title to all plots or to individual plots.
- **Marker:** You can add markers on the X- and Y-axes and customize the appearance of these markers.
- **Panel Background Color:** You can specify the background color of the plot window (light grey is the default) for all plots only.

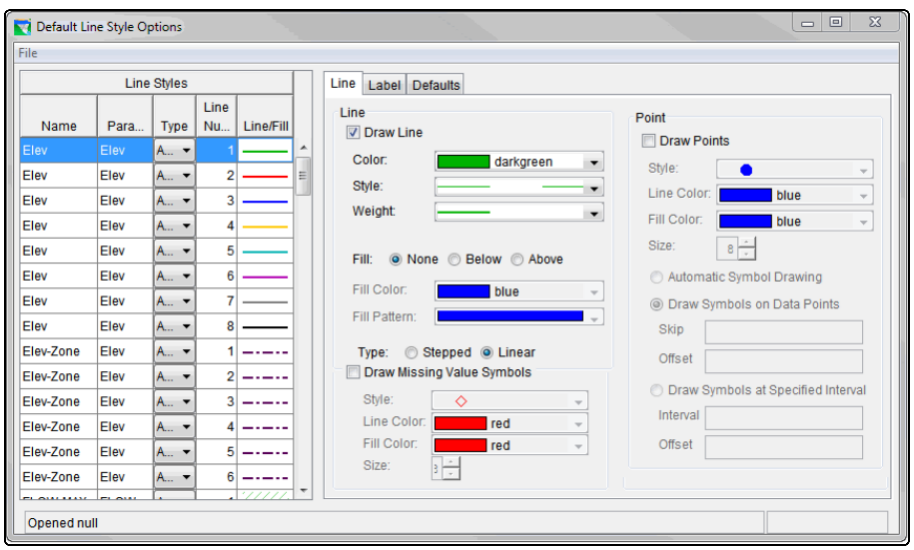
- **Curve/Line Properties:** You can choose the line and point styles, add labels, and specify symbols to indicate quality.
- **Label:** You can add borders and backgrounds to axis and legend labels.
- **Spacer:** You can specify the distance between viewports, between a viewport and the legend, and the width of side margins.
- **Viewport:** You can customize the border around the viewport, the background color and pattern, and the appearance of gridlines. You can also specify the number, size, and content of viewports.
- **Callout:** You can add descriptive callouts at specific points along a line.
- **Axis:** You can specify either a linear or log axis type, specify the axis scale, and customize tic marks.
- **Legend:** You can add titles to the plot legend and specify whether the legend appears below or to the right of the plot.

31.2.2 All Plots

You can customize all the plots for an entire watershed. From the **Plot** window, from the **Edit** menu, there are two commands - **Default Line Styles** and **Default Plot Properties**. These commands will allow you to customize all new plot windows that will be accessed for a watershed.

31.2.3 Default Line Style Options

The **Default Line Style Options** dialog (Figure 2) is for adjusting the curves (lines) in all of your plots.
It allows you to add curve (line) types, change the color and fill of the curves (lines), and add labels for the curves (lines) for all plots. For more detailed information on **Default Line Style Options** dialog, refer to the HEC-DSSVue User's Manual.

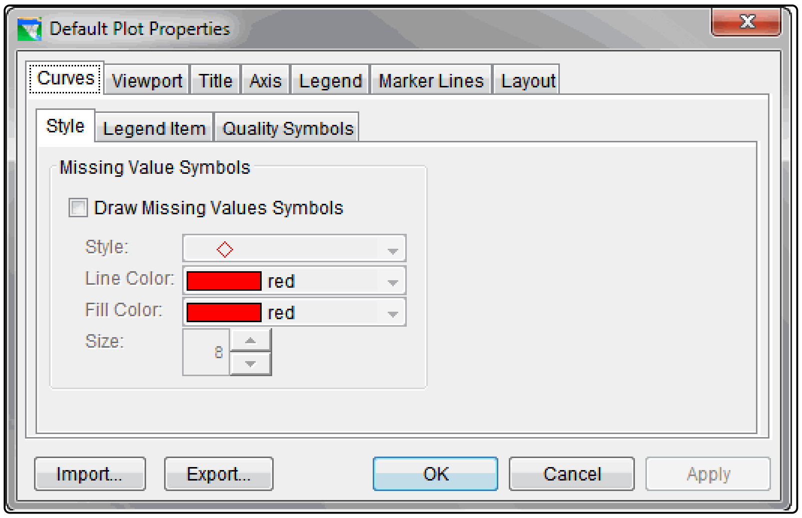


From the **Line** tab (Figure 2) you can set the color, style, and weight for your curves (lines) in all plots. If the curves have points, you can also set the style, color, and fill color for the points on your curves in all plots.
From the **Default Line Style Options** dialog (Figure 2) you can also add labels to the curves and adjust the position of

the label (**Label** tab) for all plots. You can manually adjust the maximum and minimum values for your X-axis, set viewport weight, and reverse the X-axis (**Defaults** tab) for certain curve types such as Precip.

31.2.4 Default Plot Properties

The **Default Plot Properties** dialog (Figure 3) allows you to configure the other properties of your plots. The tabs for this dialog allow you to make adjustment to the viewport, add a title for all plots, adjust grids, labels, axes, marker lines, and legend, change the color of the panel background, change the layout of the legend, and adjust the size of the spacer.

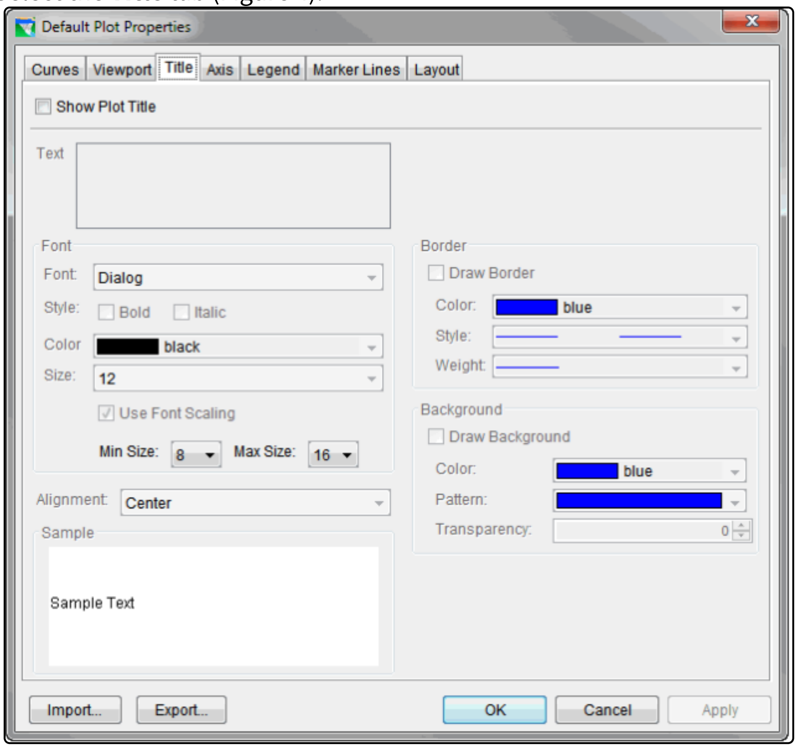


172 Figure 3 Default Plot Properties Dialog

To add a title for all plots:

1. Open a **Plot** window.
2. From the **Edit** menu, click **Default Plot Properties**.
3. The **Default Plot Properties** dialog (Figure 3) opens.

4. Select the **Title** tab (Figure 4).

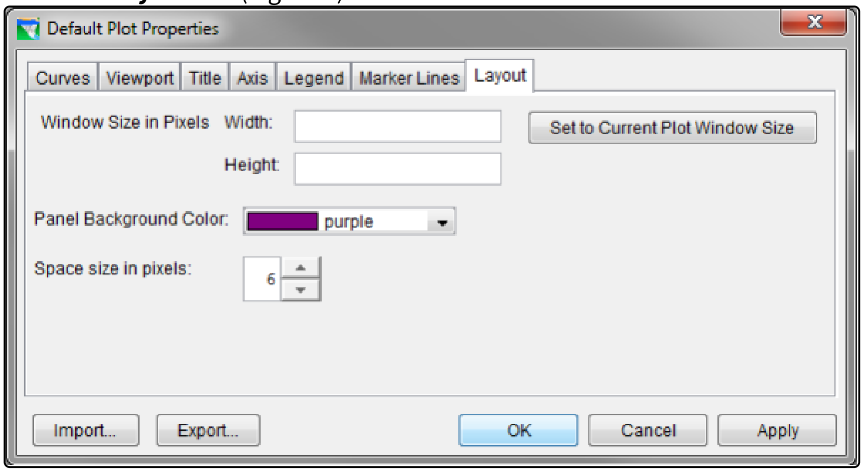


5. To add a title, click **Show Plot Title**. In the **Text** field enter a title that will appear on all of the plots in the watershed. You can align the title, adjust the font size, add a border with color, and add a color background to the title.
6. Click **Apply,** and the title will appear on all plots.
7. Click **OK**, and the **Default Plot Properties** dialog closes.

To change the panel background color for all plots:

1. Open a **Plot** window.
2. From the **Edit** menu, click **Default Plot Properties**.
3. The **Default Plot Properties** dialog (Figure 3) opens.

4. Select the **Layout** tab (Figure 5).



5. To change the panel background color of the plot, from the **Panel Background Color** list, select a color.
6. Click **Apply**, and the new color will appear in the background panels for all the plots.
7. Click **OK**, and the **Default Plot Properties** dialog closes.

31.2.5 Individual Plots

To customize individual plots that you are displaying (**Plot** window) you can use two dialogs, or you can use shortcut menus to make changes through individual editors. The dialogs include **Plot Properties** accessible from the **Edit** menu and **Select Plot Variables** accessible from the **Plot** menu.

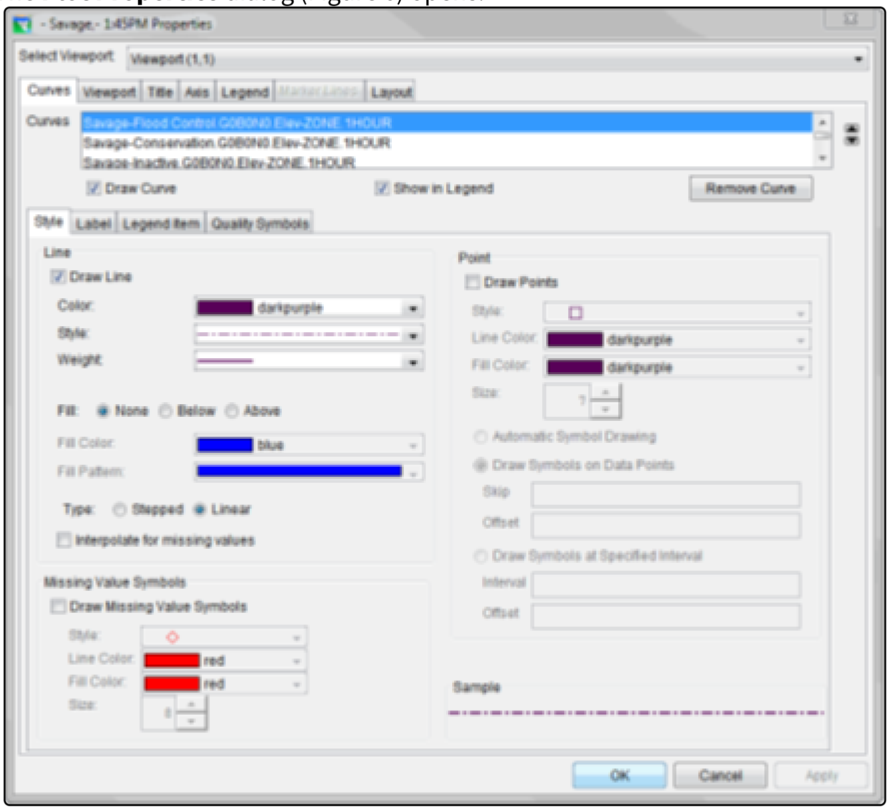
31.2.5.1 Individual Plot Properties

The **Plot Properties** dialog (Figure 6) allows you to configure multiple display properties of an individual plot. This includes the border and background patterns of the viewport; add a title to the plot; adjust the gridlines of the plot; edit the titles, tic marks, and scale of an axis; adjust the style, add points, and add quality symbols for curves; adjust the color, add labels, and position of marker lines; add a title, icon, and move the legend; change the color of the panel background; adjust the layout of the legend; and the size of the spacer.

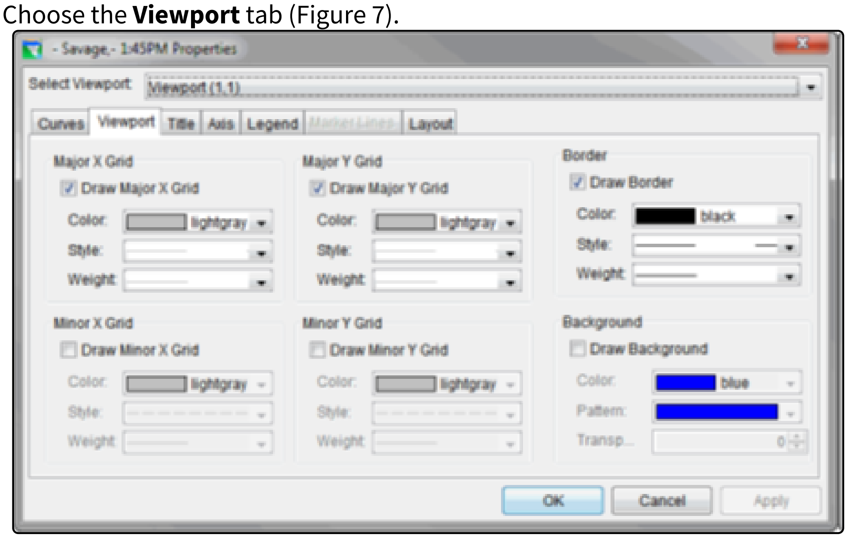
To change the gridlines of an individual plot:

1. Open a **Plot** window.
2. From the **Edit** menu, click **Plot Properties**.

3. The **Plot Properties** dialog (Figure 6) opens.



4. The **Title Bar** displays the name of the location that data is being plotted.
5. If your plot has multiple viewports, the **Select Viewport** list displays the viewport where your editing is taking place.
6. Choose the **Viewport** tab (Figure 7).



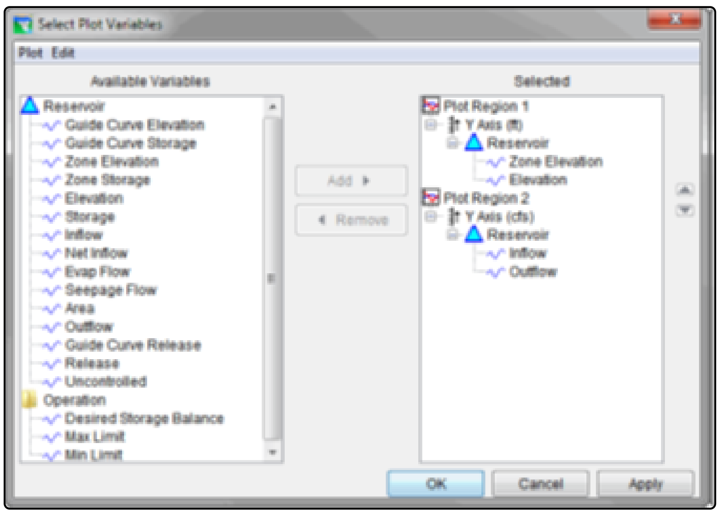
7. From the **Major X Grid** panel, click **Draw Major X Grid**. Select from the **Color**, **Style**, and **Weight** drop-down lists to modify the major X-axis gridline properties. Also, you can modify the **Minor X Grid** by clicking the **Draw Minor X Grid** and modifying the minor X-axis gridlines properties

8. From the **Major Y Grid** panel, click **Draw Major Y Grid**. Select from the **Color**, **Style**, and **Weight** drop-down lists to modify the major Y-axis gridline properties. Also, you can modify the **Minor Y Grid** by clicking the **Draw Minor Y Grid** and modifying the minor Y-axis gridlines properties.
9. Click **Apply**, and the color, style, and weight changes you made will reflect on the gridlines of your plot.
10. Click **OK**, and the **Plot Properties** dialog (Figure 7) closes.

31.2.5.2 Select Plot Variables

The **Select Plot Variables** dialog (Figure 8) allows you to customize the layout of an individual plot. You can add and remove axes, arrange the order of viewports, add and remove viewports, and set the weight of viewports. The plot components are displayed in a tree structure.

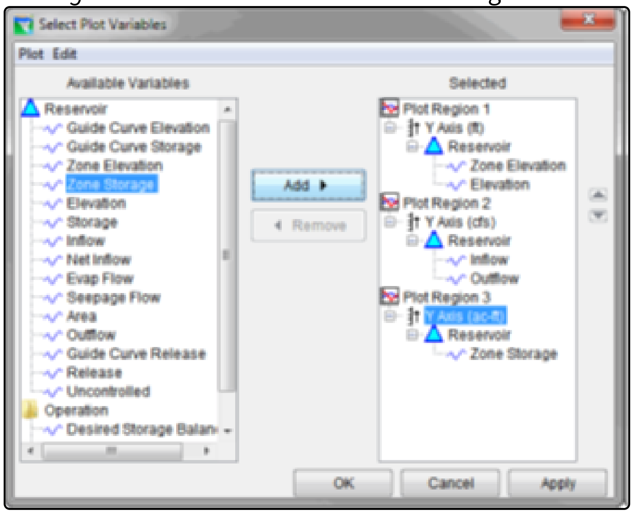
To add a viewport to an individual plot:



173 Figure 8 Select Plot Variables Dialog

1. Open a **Plot** window.
2. From the **Plot** menu, click **Select Variables**.
3. The **Select Plot Variables** dialog (Figure 8) opens.
4. From **Edit** menu, click **Add Plot Region**, and a new plot region (viewport) is added to the bottom of the **Selected**
5. You now need to add data from the **Available Variables** tree to your new plot region. Select a curve from **Available Variables** tree and click **Add**.

6. In this example (Figure 9), *Zone Storage* is added to the *Reservoir* tree from the **Available Variables** by selecting *Plot Region 3* in the **Selected** tree and clicking **Add**.



7. Click **Apply**, and the plot region displays the curve that was added.
8. Click **OK**, and the **Select Plot Variables** dialog (Figure 9) closes, and your plot will display the changes that were made.

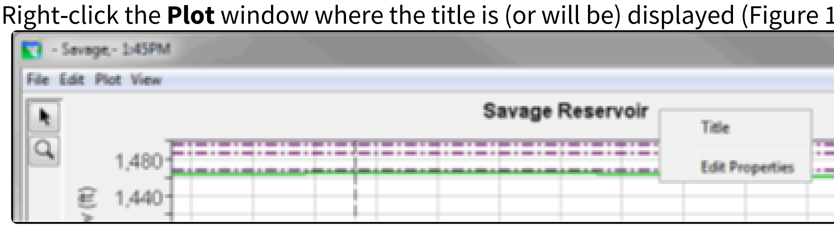
31.2.5.3 Shortcut Menus

When you want to edit a specific property of a plot without using the **Individual Plot Properties** dialog (Figure 6), you can use individual plot property dialogs instead. To access these dialogs, select the **Pointer/Select** tool, *right-click* the element of the **Plot** window (Figure 1) that you want to edit, and from the shortcut menu that appears, click **Edit Properties**.

31.2.5.4 Edit Title Properties

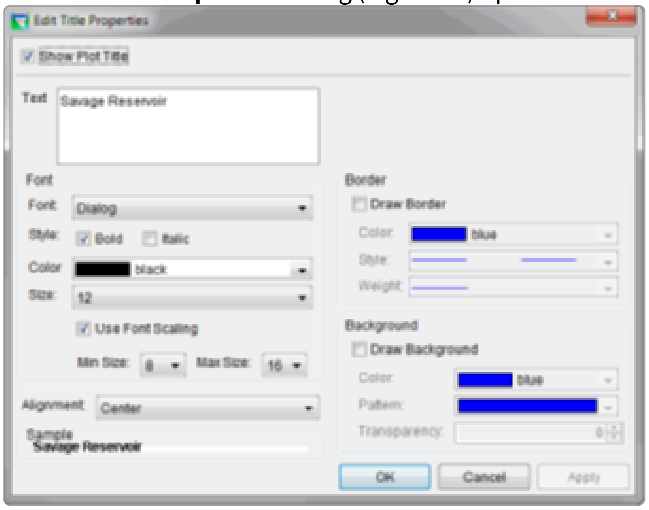
To add a title to an individual plot:

1. Right-click the **Plot** window where the title is (or will be) displayed (Figure 10).



2. From the shortcut menu, click **Edit Properties**.

3. The **Edit Title Properties** dialog (Figure 11) opens.

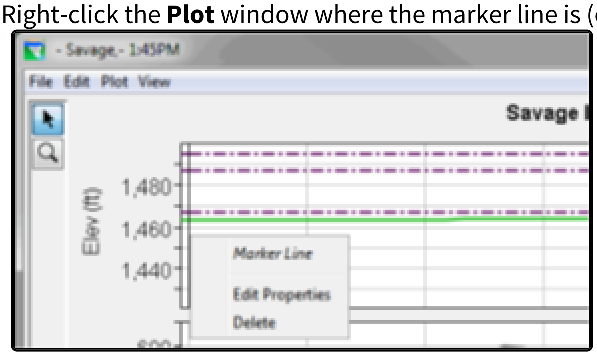


4. Check **Show Plot Title**, and in the **Text** field, enter the title for the individual plot. From this dialog you can also set the alignment, the size of the font, and add a border and background colors to the title.
5. Click **Apply**, and the title is displayed on the plot.
6. Click **OK**, and the **Edit Title Properties** dialog (Figure 11) closes.

31.2.5.5 Edit Marker Lines Properties

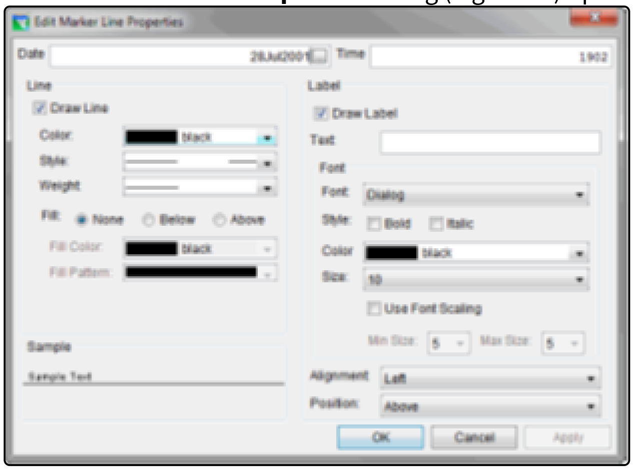
To edit the marker lines of an individual plot:

1. Right-click the **Plot** window where the marker line is (or will be) displayed (Figure 12).



2. From the shortcut menu, click **Edit Properties**.

3. The **Edit Marker Line Properties** dialog (Figure 13) opens.

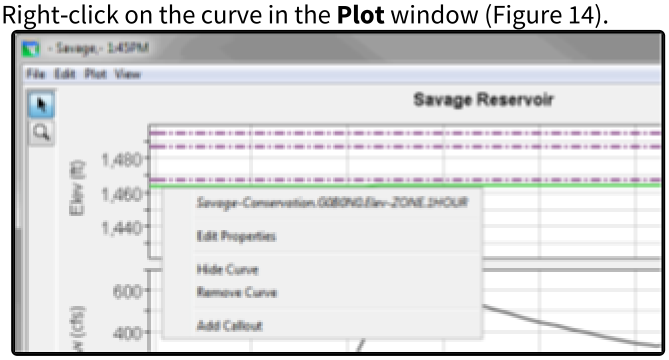


4. To change the style of the marker line, you need to click the **Draw Line** You can change the color, style, and weight of the marker line. You can also add fill if you wish.
5. If you want to add a label to the marker line, click the **Draw Label** You can add a label, the alignment of the label, and the placement of the label along the marker line.
6. If your plot has more than one viewport, you will have to set marker line properties for each viewport.
7. Click **Apply**, and the changes to the marker line are displayed on the plot.
8. Click **OK**, and the **Edit Marker Line Properties** dialog (Figure 13) closes.

31.2.5.6 Edit Curve Properties

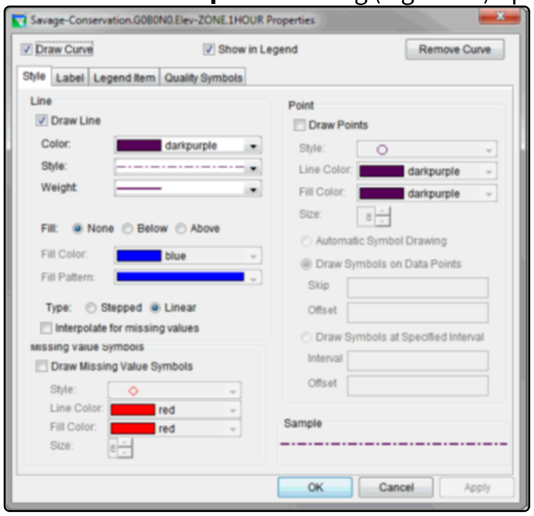
To edit the curves of an individual plot:

1. Right-click on the curve in the **Plot** window (Figure 14).



2. From the shortcut menu, click **Edit Properties**.

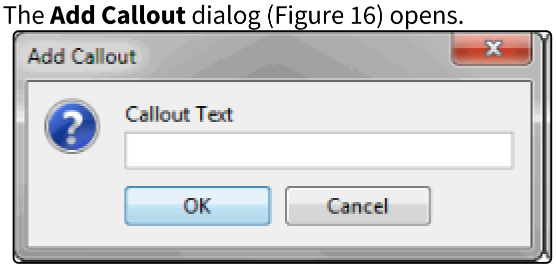
3. The **Edit Curve Properties** dialog (Figure 15) opens.



4. To change the style of the curve, select the **Style** You can change the color, style, and weight of the curve. You can also add fill below or above the curve if you wish.
5. If you want to display the quality of the data that the curve represents, select the **Quality Symbols** tab, and click the **Draw Quality Symbols** Then, set the symbol color and size for each type of quality.
6. If your plot has more than one curve, you will need to edit properties for each curve individually.
7. If your plot has more than one viewport, you will have to set curve properties for each viewport.
8. Click **Apply**, and the changes to the marker line are displayed on the plot.
9. Click **OK**, and the **Edit Curve Properties** dialog (Figure 15) closes.

To add a callout to a curve of an individual plot:

1. Right-click the **Plot** window where a curve is displayed (Figure 14).
2. From the shortcut menu, click **Add Callout**.
3. The **Add Callout** dialog (Figure 16) opens.

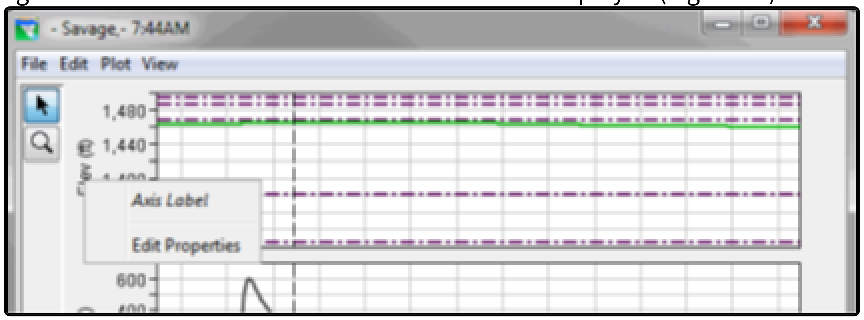


4. Enter the callout you want in the **Callout Text**
5. Click **Apply**, and the changes to the marker line are displayed on the plot.
6. Click **OK**, the new callout is displayed on the plot, and the **Add Callout Dialog** (Figure 16) closes.

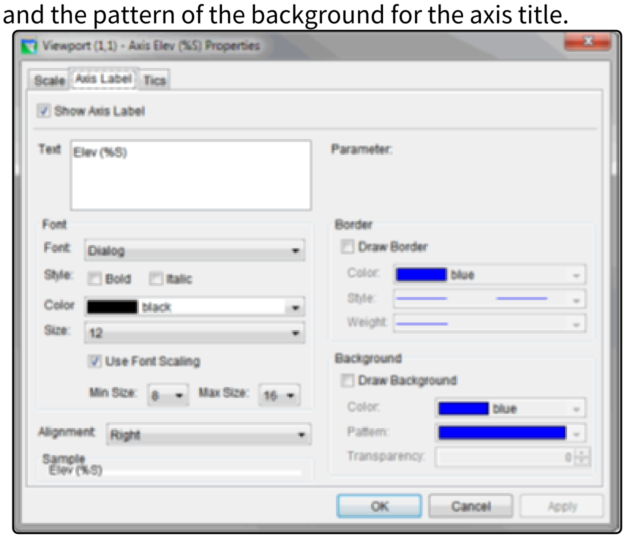
31.2.5.7 Edit Axis Title Properties

To edit the appearance of axis titles for an individual plot:

1. Right-click the **Plot** window where the axis title is displayed (Figure 17).



2. From the shortcut menu, click **Edit Properties**.
3. The **Edit Axis Properties** dialog (Figure 18) opens.
4. To give an axis title a border, click the **Axis Label** In the **Border** panel, click **Draw Border**. You can set the color, style, and weight of the border around the axis title.
5. To give an axis title a background color, in the **Background** panel, click **Draw Background**. You can set the color and the pattern of the background for the axis title.

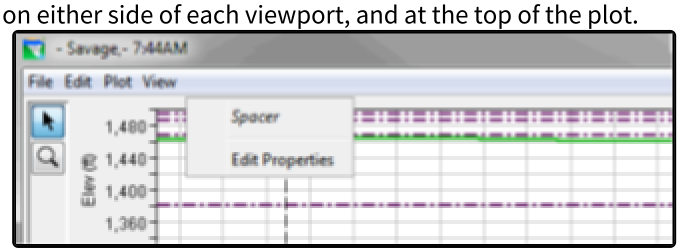


6. Click **Apply**, and the axis title is displayed with your changes.
7. Click **OK**, and the **Edit Axis Properties** dialog (Figure 18) closes.

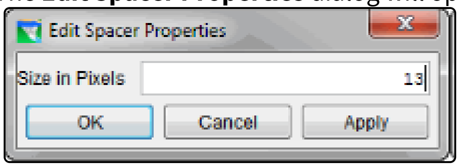
31.2.5.8 Edit Spacer Properties

To edit the spacer properties for an individual plot:

1. Right-click the **Plot** window where the spacer is (Figure 19) located on the plot. Spacers are between viewports, on either side of each viewport, and at the top of the plot.



2. From the shortcut menu, click **Edit Properties**.
3. The **Edit Spacer Properties** dialog will open (Figure 20).

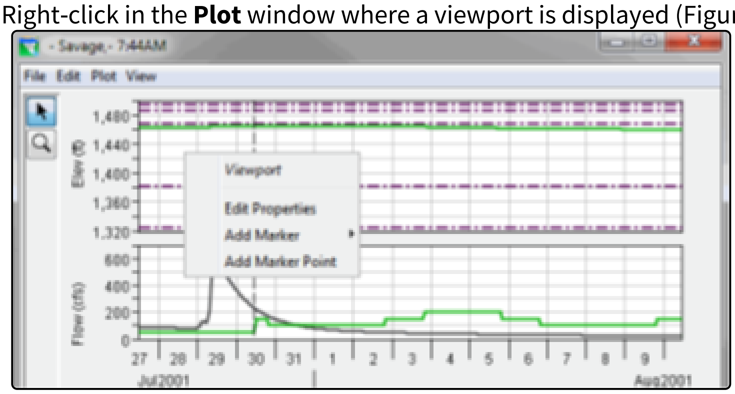


4. Enter the size of the spacer in pixels in the **Size in Pixels**
5. Click **Apply**, and the changes you made to the spacer will appear on the plot.
6. Click **OK**, and the **Edit Spacer Properties** dialog (Figure 20) will close.

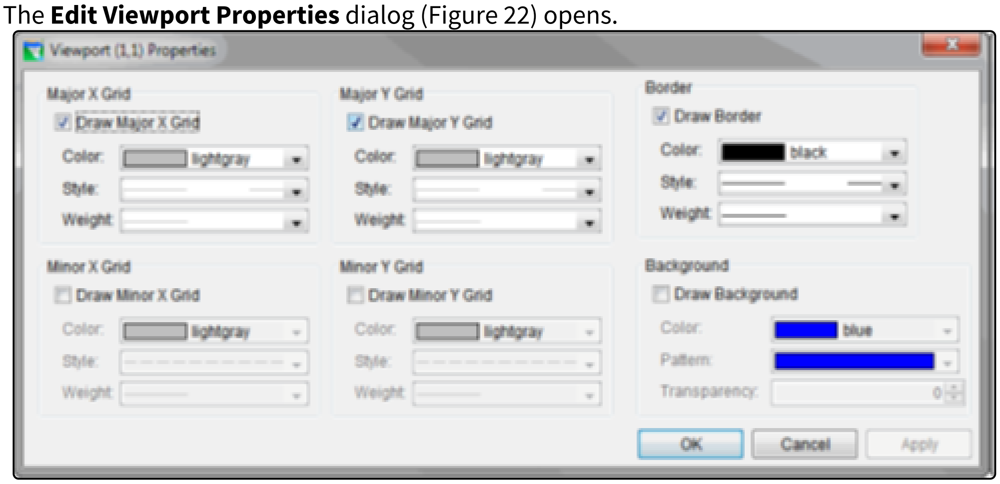
31.2.5.9 Edit Viewport Properties

To edit the viewports of an individual plot:

1. Right-click in the **Plot** window where a viewport is displayed (Figure 21).



2. From the shortcut menu, click **Edit Properties**.
3. The **Edit Viewport Properties** dialog (Figure 22) opens.



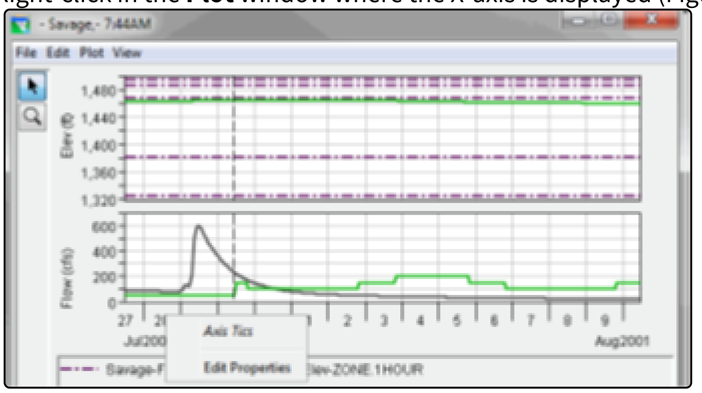
4. To change the viewport border's color, pattern or transparency click **Draw Border** under the **Border**
5. To change the viewport background's color, pattern or transparency click **Draw Background** under the **Background**
6. If you want to change the color, style, or weight of the gridlines of the viewport, click the **Draw Major X Grid**, **Draw Major Y Grid**, **Draw Minor X Grid**, or **Minor Y Grid** checkboxes under the grid section.

7. If your plot has more than one viewport, you will have to set curve properties for each viewport.
8. Click **Apply**, and the changes to the viewport are displayed on the plot.
9. Click **OK**, and the **Edit Viewport Properties** dialog (Figure 22) closes.

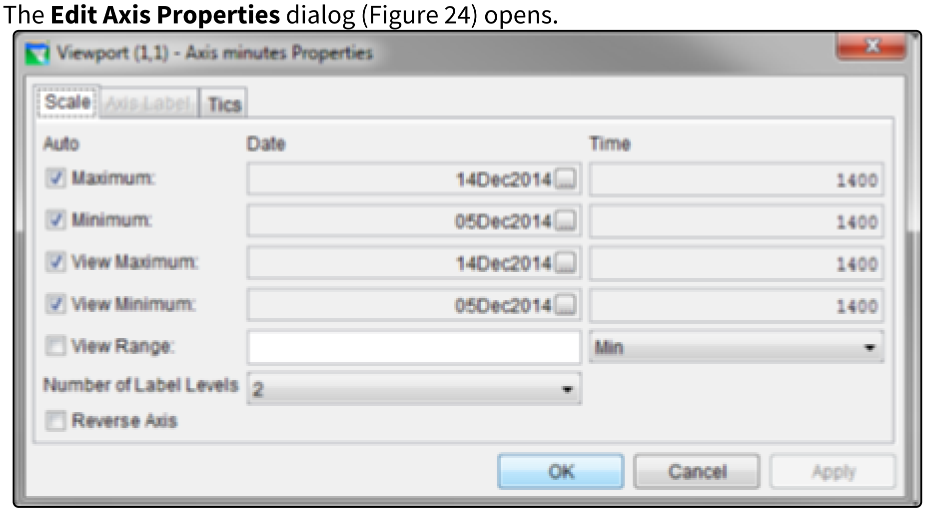
31.2.5.10 Edit Axis Tics Properties

To edit the X-axis scale of an individual plot:

1. Right-click in the **Plot** window where the X-axis is displayed (Figure 23).



2. From the shortcut menu, click **Edit Properties**.
3. The **Edit Axis Properties** dialog (Figure 24) opens.

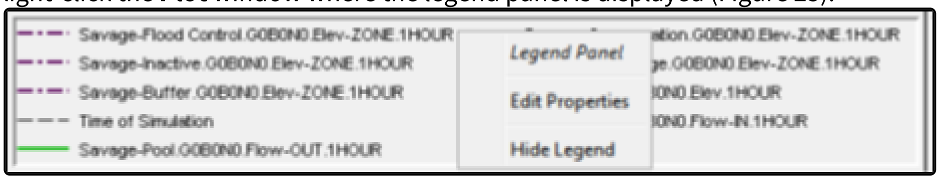


4. Select the **Scale** In the select table you can choose to automatically set the maximum, minimum, view maximum, view minimum, and view range. Also, you can set the number of label levels and reverse the X-axis.
5. To change the tic marks of the X-axis, select the **Tics** You can add minor tic marks, do minor tic marks only, and change the color of the tic mark and tic mark label.
6. If your plot has more than one viewport, you will have to set axis tics properties for each viewport.
7. Click **Apply**, and the changes to the X-axis are displayed on the plot.
8. Click **OK**, and the **Edit Axis Properties** dialog (Figure 24) closes.

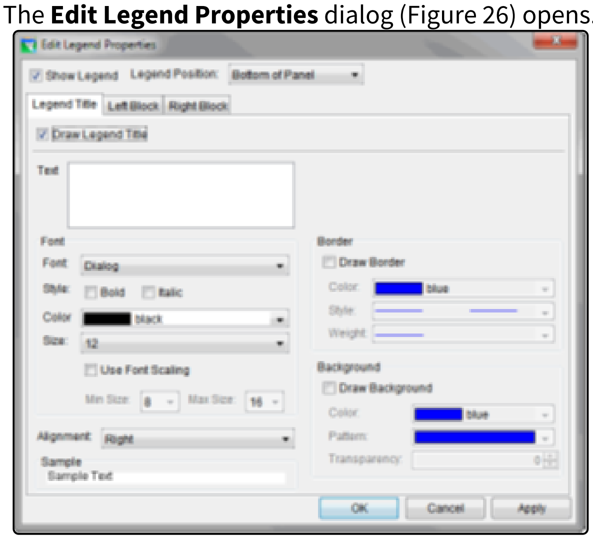
31.2.5.11 Edit Legend Panel Properties

To edit the legend panel of an individual plot:

1. Right-click the **Plot** window where the legend panel is displayed (Figure 25).



2. From the shortcut menu, click **Edit Properties**.
3. The **Edit Legend Properties** dialog (Figure 26) opens.



4. You can give the legend panel a title by checking the **Draw Legend Title** and entering information in the **Text**
5. You can also add icon information to the left and right areas of the legend bock from the **Left Block** and **Right Block** For example, if you want a description to display on the left side of the legend panel, click the **Left Block** tab and enter the description in the **Text** field.
6. To add an icon, you need to enter the path and filename of icon to be display in the **Icon** panel for either the left or right sides of the **Legend**
7. Click **Apply**, and the changes to the legend panel are displayed on the plot.
8. Click **OK**, and the **Edit Legend Properties** dialog (Figure 26) closes.

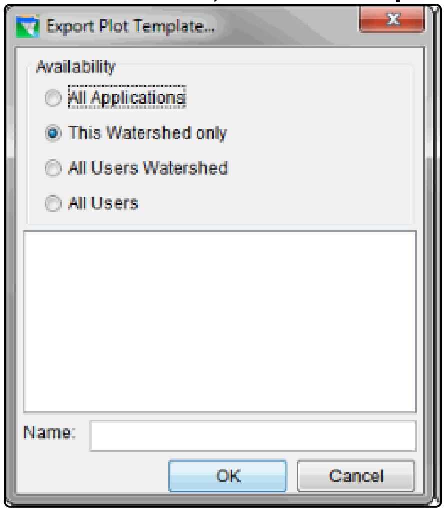
31.3 Exporting and Importing Templates

After you have customized a plot, you can save its settings as a template for use in other plots. Generally, you would use templates when scripting plots. For example, every day you might want to generate a plot of flow, stage, and precipitation via a script, and then apply a template that has all of the correct formatting, such as viewport placement, size, line colors, and fills.

To create a template from a plot:

1. Open a **Plot** window.

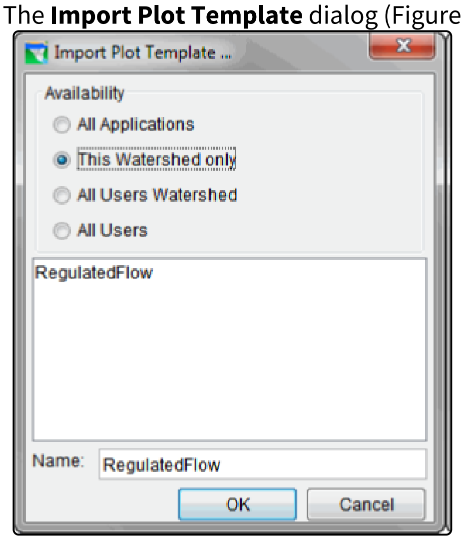
2. From the **File** menu, click **Save Template**, and the **Export Plot Template** dialog will open (Figure 1).



3. Specify whether you want the template to be available for **All Applications**, **This Watershed only**, **All Users Watershed**, or **All Users**. Then enter a name for the template in the **Name**
4. Click **OK**, the **Save Template** dialog will close, and the template will be saved.

To apply (import) a template you have previously created to another plot that is currently open:

1. Open a **Plot** window.
2. From the **File** menu, click **Apply Template**.
3. The **Import Plot Template** dialog (Figure 2) opens.



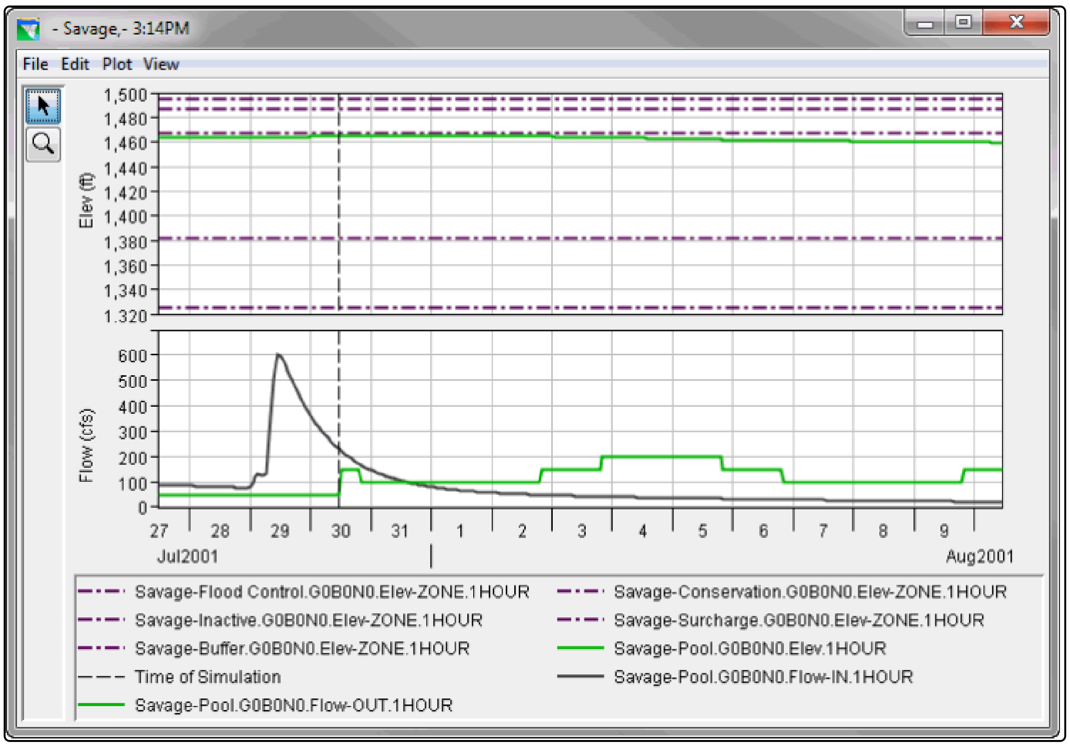
4. Specify whether you want the template to be available for **All Applications** or **This Watershed only**.
5. Choose the template you want from the list.
6. When you select a template, its name will display in the **Name**
7. Click **OK** to apply the template to the current plot, and the **Import Plot Template** dialog closes.

32 Printing and Copying HEC-RTS Data

Plots and tables in the Visualization, Acquisition, and Modeling modules offer detailed views of data and model results that you can print or copy and paste into other applications. In addition, you can print reports generated by the individual models.

32.1 Working with Plots

Plots provide a graphical representation of your data. From the **Plot** window (Figure 1) you can view the plot, as well as print, save as a graphics file, and save the properties of the plot to a template to be used by other plots. Following is an overview of the menu bar of the **Plot** window. The commands available from the menus are:



174 Figure 1 Plot Window

- **File:** From this menu you can view the plotted data in a tabular format, save the plot as a graphics file, save a template, use a template, and print the plot. Available commands are: **Tabulate**, **Save As**, **Save Template**, **Apply Template**, **Save Specification**, **Print**, **Page Setup**, **Print Preview**, **Print Multiple**, and **Close**.
- **Edit:** From this menu you can customize the plot. Available commands are: **Plot Properties**, **Default Line Styles**, and **Default Plot Properties**.
- **Plot:** From this menu you can select variable to plot, open and save plot type. Available commands are: **Select Variables**, **Open Plot Type**, and **Save Plot Type**.
- **View:** From this menu you can have the plot window always on top of your desktop, and you can choose to use line styles. Available commands are: **Always On Top**, **Use Line Styles**, **Zoom to all**, **Legend Placement**, **Hide Legend**, **Refresh**, and **Live Display**.

The **Tools** appear in a toolbar on the left side of the **Plot** window (Figure 1). The tools change the appearance of the mouse, as well as the functionality of the mouse:

 **Pointer/Select Tool.** With the **Pointer/Select Tool**, you can access shortcut menus that allow you to customize features of your plots using the plot editing tools.

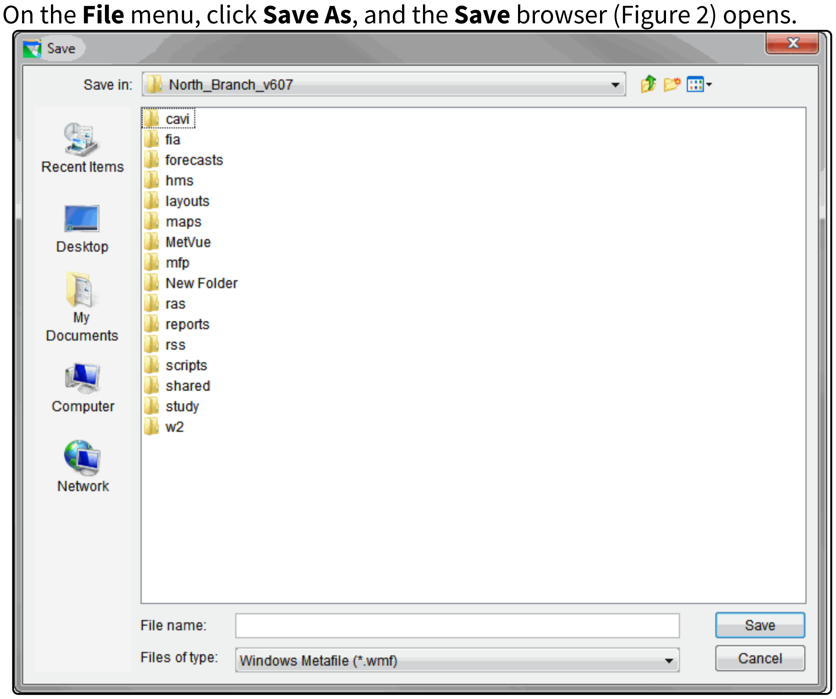
 **Zoom Tool.** The **Zoom Tool** allows you to view data closely at a specific time. To *zoom in*, hold the mouse button down and outline the area you want to enlarge. To *zoom out*, click the right mouse button (right-click). The zoom out is done by a factor of two and positions the clicked location at the center of the display area.

32.1.1 Save As

The **Save As** command allows you to save your plot to a graphics file. HEC-RTS has four possible formats: **Windows Metafile** (***.wmf*), ***Postscript** (**.ps*), ***JPEG** (***.jpg***, *.jpeg*), and **Portable Network Graphics** (**.png*).

To create a graphics file:

1. On the **File** menu, click **Save As**, and the **Save** browser (Figure 2) opens.



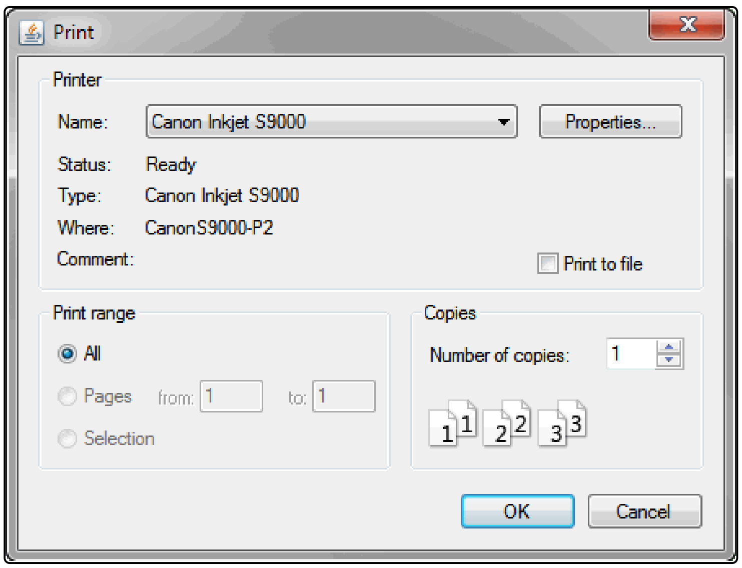
2. Select the location you will save your file from the **Save in** drop-down list.
3. Enter a name for the file in the **File name** field.
4. From the **Files of type** list, select the type of graphics file you wish to create.
5. Click **Save**, and the **Save** browser closes, and your plot is saved as the selected graphics file type.

32.1.2 Print Commands

The **Plot** window (Figure 1) provides the user with several different commands to setup the printing process and to print.

32.1.3 Print

The **Print** command opens a standard system **Print** dialog (Figure 3). From the **Print** dialog you can select the printer, choose to print to a file, and the number of copies to print. Click **OK**, and the **Print** dialog closes and the plot will print to the selected printer.

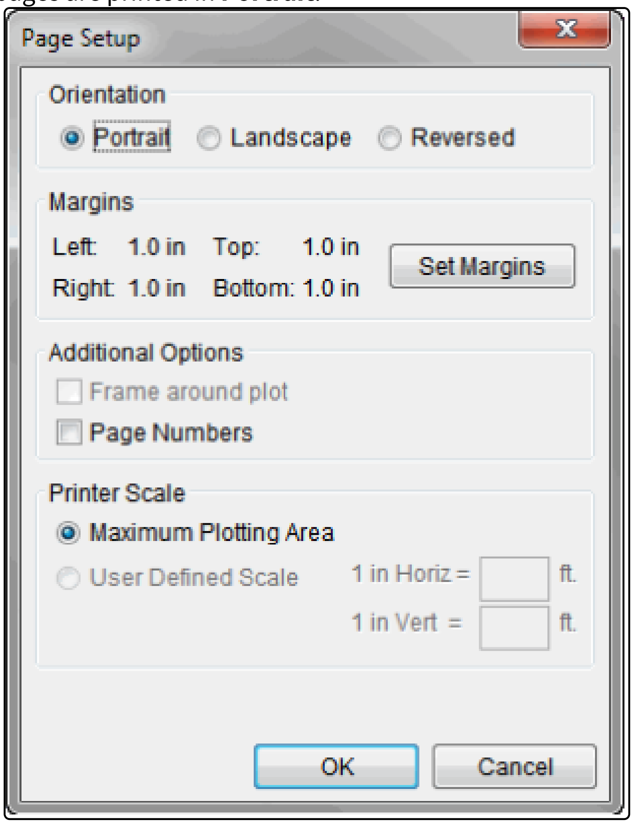


175 Figure 3 Print Dialog

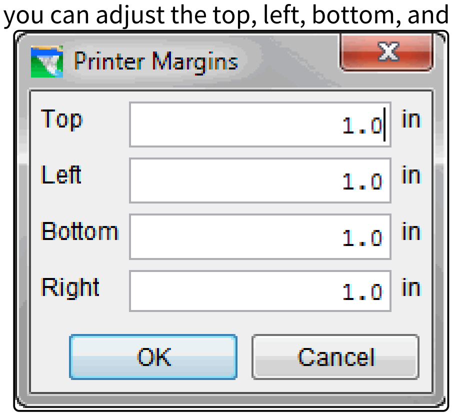
32.1.4 Page Setup

The **Page Setup** command opens the **Page Setup** dialog (Figure 4). When using this **Page Setup** dialog, you can setup the format of the printed page. Following is a list of the available options:

- **Orientation:** To set the orientation, select **Portrait** or **Landscape** from the **Orientation** panel. By default, the pages are printed in **Portrait**.



- **Margins:** The default margin settings are displayed in the Margins panel of the **Page Setup** dialog. To set the margins, click **Set Margins**, and the **Printer Margins** dialog (Figure 5) opens. From the **Printer Margins** dialog, you can adjust the top, left, bottom, and right margins.

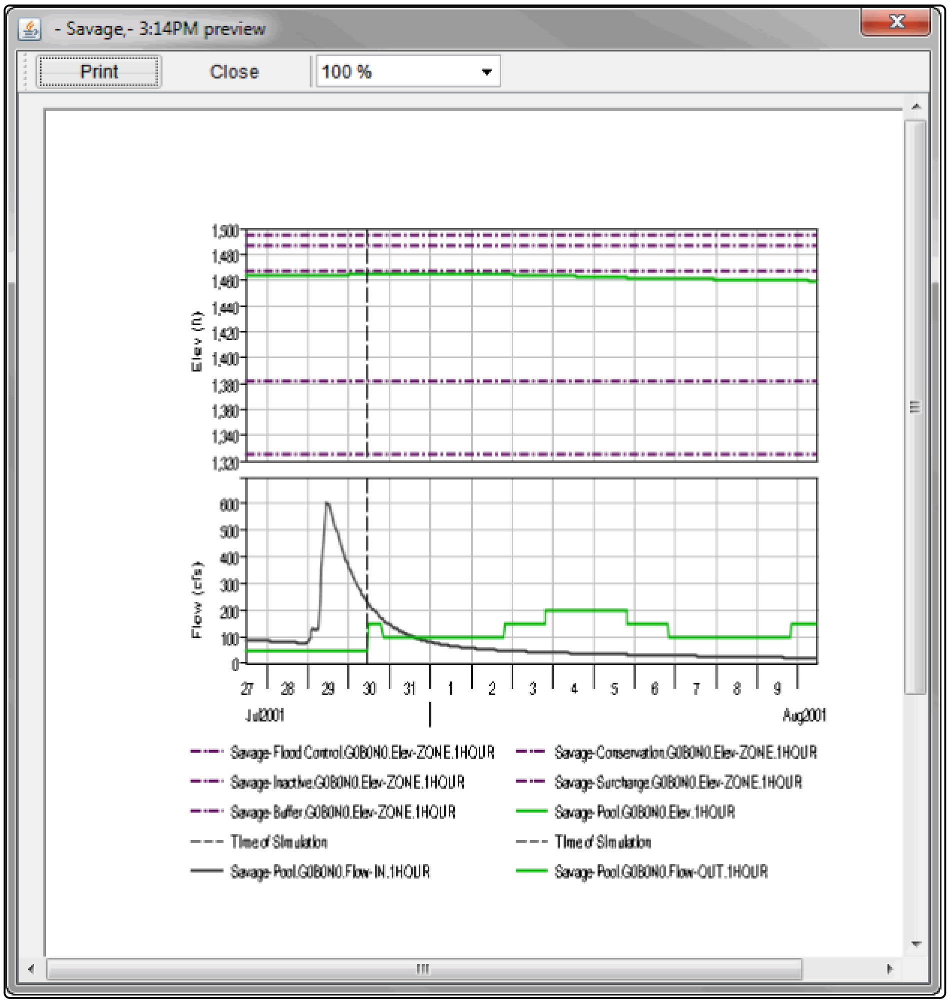


176 Figure 5 Printer Margins Dialog

- **Additional Options:** If you would like page numbers included on your printed page, select the **Page Numbers** checkbox (Figure 4).
- **Printer Scale:** The default option is **Maximum Plotting Area**.

32.1.5 Print Preview

The **Print Preview** command opens the **Print Preview** dialog (Figure 6). From the **Print Preview** dialog, you can view the plot, adjust the scaling of the plot, and print the plot. The **Print** command automatically prints the plot to the default system printer.

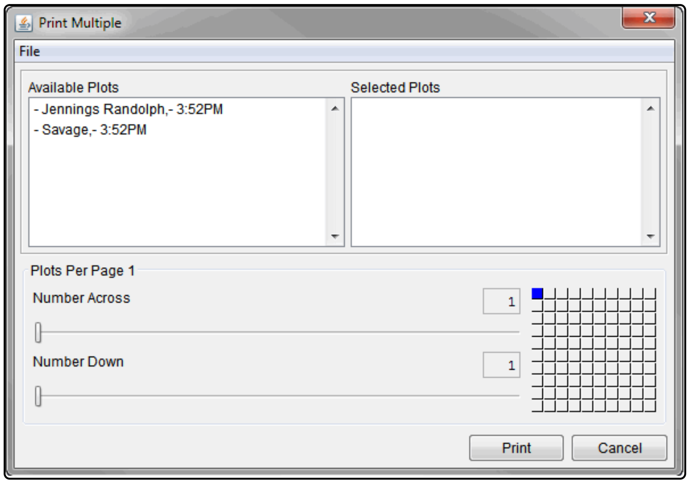


177 Figure 6 Print Preview Dialog - Plots

32.1.6 Print Multiple

The **Print Multiple** command opens the **Print Multiple** dialog (Figure 7). To be able to print multiples plots you must have opened multiple individual plot windows. The opened plots are listed in the **Available Plots** list of the **Print Multiple** dialog.

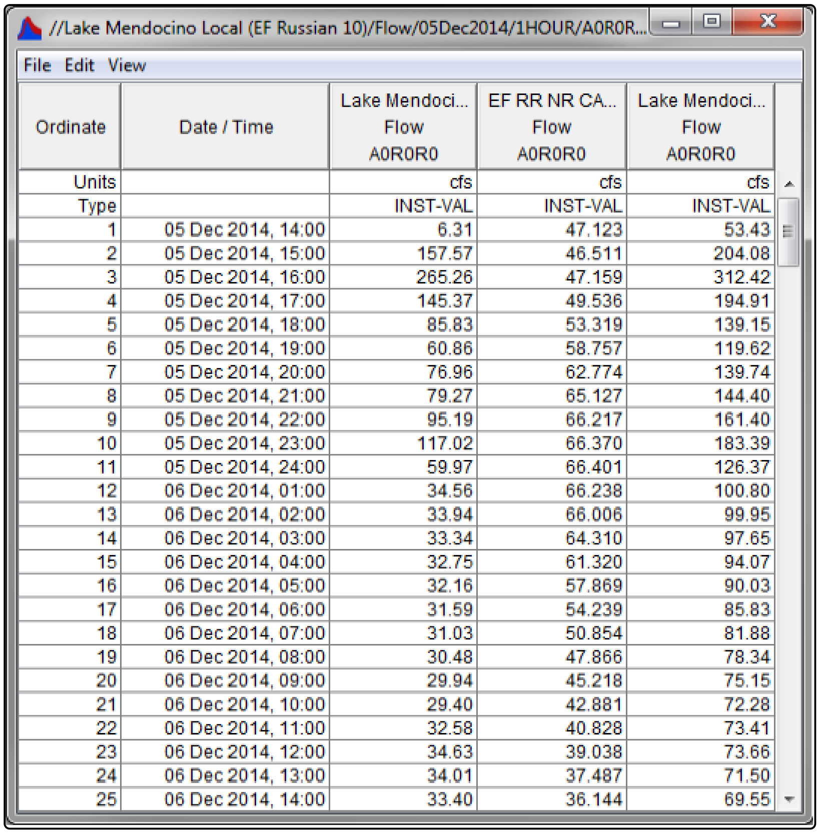
To select plots to be printed on one page, double-click on a plot name in the **Available Plots** list. The plot name will move to the **Selected Plots** list. You can adjust the way the plots are arranged on a page, horizontally or vertically. Use the **Number Across** or **Number Down** sliders. The grid to the right of the sliders reflects your choices. From the **File** menu, the **Page Setup** and **Print Preview** commands are available.



178 Figure 7 Print Multiple Dialog

32.2 Tables

Tables provide a tabular format of your data. From the **Tabulate** window (Figure 1) you can view the data and results, as well as print the table, copy the selected contents of the **Tabulate** window to the clipboard, adjust the display of the data, and save the data to an ASCII file. Following is an overview of the menu bar of the **Tabulate** window.



Ordinate	Date / Time	Lake Mendoci... Flow A0R0R0	EF RR NR CA... Flow A0R0R0	Lake Mendoci... Flow A0R0R0
Units		cfs	cfs	cfs
Type		INST-VAL	INST-VAL	INST-VAL
1	05 Dec 2014, 14:00	6.31	47.123	53.43
2	05 Dec 2014, 15:00	157.57	46.511	204.08
3	05 Dec 2014, 16:00	265.26	47.159	312.42
4	05 Dec 2014, 17:00	145.37	49.536	194.91
5	05 Dec 2014, 18:00	85.83	53.319	139.15
6	05 Dec 2014, 19:00	60.86	58.757	119.62
7	05 Dec 2014, 20:00	76.96	62.774	139.74
8	05 Dec 2014, 21:00	79.27	65.127	144.40
9	05 Dec 2014, 22:00	95.19	66.217	161.40
10	05 Dec 2014, 23:00	117.02	66.370	183.39
11	05 Dec 2014, 24:00	59.97	66.401	126.37
12	06 Dec 2014, 01:00	34.56	66.238	100.80
13	06 Dec 2014, 02:00	33.94	66.006	99.95
14	06 Dec 2014, 03:00	33.34	64.310	97.65
15	06 Dec 2014, 04:00	32.75	61.320	94.07
16	06 Dec 2014, 05:00	32.16	57.869	90.03
17	06 Dec 2014, 06:00	31.59	54.239	85.83
18	06 Dec 2014, 07:00	31.03	50.854	81.88
19	06 Dec 2014, 08:00	30.48	47.866	78.34
20	06 Dec 2014, 09:00	29.94	45.218	75.15
21	06 Dec 2014, 10:00	29.40	42.881	72.28
22	06 Dec 2014, 11:00	32.58	40.828	73.41
23	06 Dec 2014, 12:00	34.63	39.038	73.66
24	06 Dec 2014, 13:00	34.01	37.487	71.50
25	06 Dec 2014, 14:00	33.40	36.144	69.55

179 Figure 1 Tabulate Window

The commands available from the menus are:

- **File:** From this menu, you can print the data in a tabular format and view a plot of the data. Available commands are: **Save**, **Save As**, **Print**, **Print Preview**, **Export**, **Plot**, and **Close**.
- **Edit:** From this menu, select data to be copied to the clipboard. Available commands are: **Allow Editing**, **Cut**, **Copy**, **Paste**, **Select All**, **Find**, and **Compare Data Sets**.
- **View:** From this menu, you can control the appearance of the data displayed in the **Tabulate** window. Available commands are: **Commas**, **Reverse Order**, **Date and Time Separately**, **Date with 4 Digit Years**, **Show Decimal Places** (up to 6 decimal places), and **Show Missing As** (blank, 901.00, M, -M).

32.2.1 Displaying Data

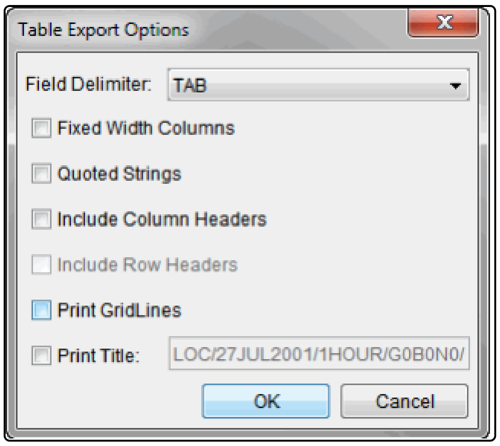
By default, data is displayed with commas and two decimal places. If you want to change the way the data is displayed, from the **View** menu there are several options. If you do not want commas displayed, from the **View** menu, click **Commas**.

For time series data the date and time are displayed in the following default format: **ddmmmyy hhmm**. You can change the way the date and time are displayed. From the **View** menu, click **Date and Time Separately**; and the date and time are now displayed in two separate columns in the table. Another way is from the **View** menu, click **Date with 4 Digit Years**; and the date displays a four-digit year instead of a two-digit year.

Numbers by default are displayed with two decimal places. If you want to see more or a smaller number of decimal places, from the **View** menu, point to **Decimal Places**, and click on the number (up to 6 decimal places) of decimal places you would like displayed.

32.2.2 ASCII File

You can save the table of data to an ASCII file. Right-click on the table, and from the shortcut menu, click **Export**. The **Table Export Options** dialog will open (Figure 2). From this dialog, you can set certain options for the ASCII file.



180 Figure 2 Table Export Options Dialog

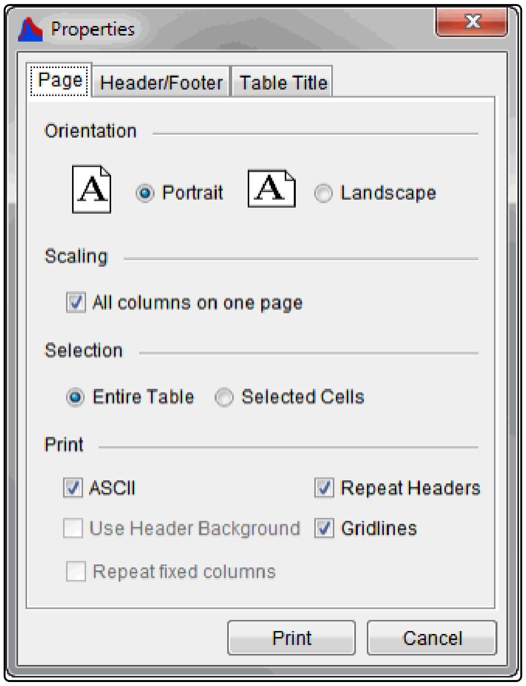
You can set the delimiter for the file from the **Field Delimiter** drop-down list. The field delimiter choices are **TAB** (default), **SPACE**, **COMMA COLON**, and **SEMI-COLON**. You also make choices on whether the file contains fixed width columns, strings are set in quotes, whether to include column headers in the file, include gridlines in the file, and whether to include the a title in the file.

When you have all of your options set, click **OK**, and a **Save** browser opens. Find the location where you want the file saved, and provide a filename. Click **Save**, and the file is created without a filename extension. Since this is an **ASCII** file, you might want to give the file an extension of **.txt**.

32.2.3 Printing

The **Print** command opens a standard system **Print** dialog. From the **Print** dialog you can select the printer, choose to print to a file, and the number of copies to print. Click **OK**, and the **Print** dialog closes, and the table will print to the selected printer.

Another way to print is to right-click on the table. From the shortcut menu click **Print,** and the **Properties** dialog (Figure 3) opens. From this **Properties** dialog, you can set up the format of the table for output. From the **Page** tab, you can set the orientation; fit the table to a page, and decide whether to include the entire table or only a selection.



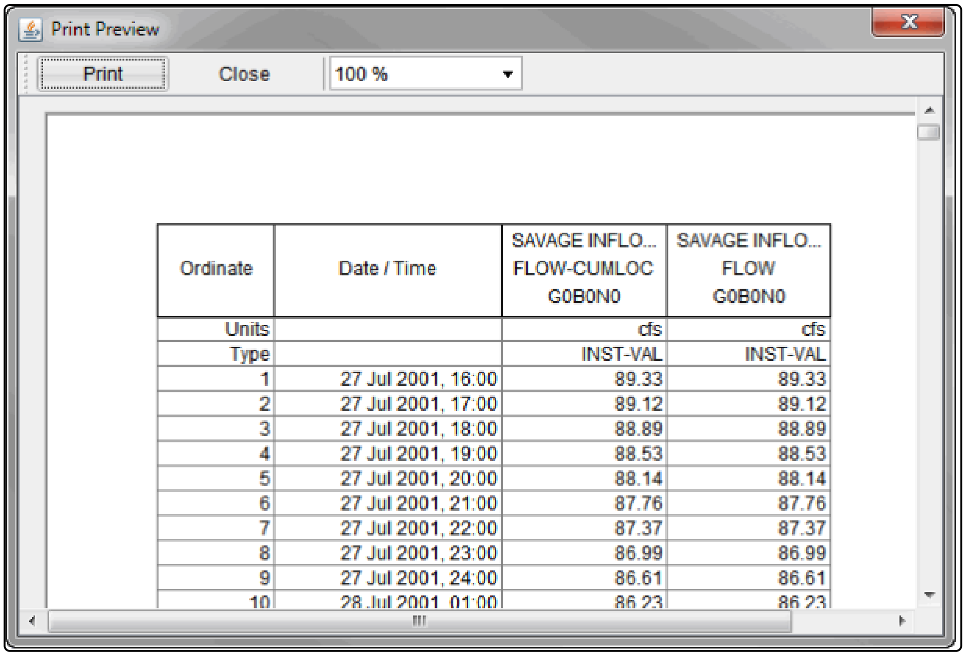
181 Figure 3 Properties Dialog

In addition, you can also choose to print the table in ASCII, add headers, add background color to headers (non-ASCII option), and add gridlines.

If you want to add header or footer information to the table report, click the **Header/Footer** tab. From the **Header/ Footer** tab enter header or footer information.

The title of the table report defaults to the pathname tag associated with the data. If you want to change this title, click the **Table Title** tab. From the **Table Title** tab, enter or change the title, and decide if you want the title to print on each page of the report. When you have everything set click **Print**, and the **Print** dialog opens.

To preview your table report, right-click on the table, and from the shortcut menu, click **Print Preview**. The **Properties** dialog (Figure 3) opens. Setup your options for the table report, click **Print**, and the **Print Preview** dialog (Figure 4) opens.



Ordinate	Date / Time	SAVAGE INFLO... FLOW-CUMLOC G0B0N0	SAVAGE INFLO... FLOW G0B0N0
Units		cfs	cfs
Type		INST-VAL	INST-VAL
1	27 Jul 2001, 16:00	89.33	89.33
2	27 Jul 2001, 17:00	89.12	89.12
3	27 Jul 2001, 18:00	88.89	88.89
4	27 Jul 2001, 19:00	88.53	88.53
5	27 Jul 2001, 20:00	88.14	88.14
6	27 Jul 2001, 21:00	87.76	87.76
7	27 Jul 2001, 22:00	87.37	87.37
8	27 Jul 2001, 23:00	86.99	86.99
9	27 Jul 2001, 24:00	86.61	86.61
10	28 Jul 2001, 01:00	86.23	86.23

182 Figure 4 Print Preview Dialog

From this **Print Preview** dialog, you can view the table report, adjust the scaling of the report, and print the report. The **Print** command automatically prints the report.

32.3 Printing Reports from HEC-RTS

32.3.1 Setup Module

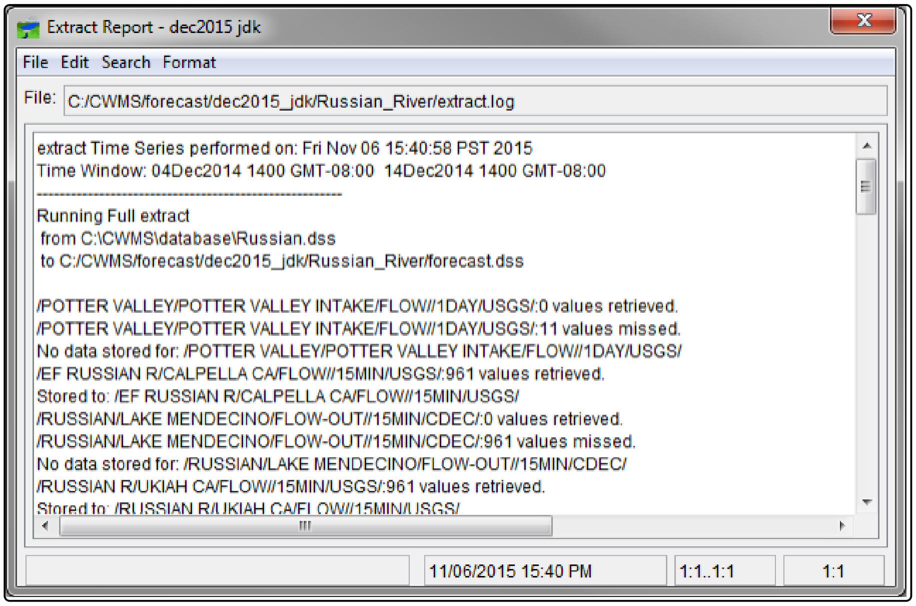
In the **Setup Module**, from the **Reports** menu there are several reports that are available that display information about the watershed elements. To print these reports, from the **File** menu of the report window, click **Print**, and the **Properties dialog** opens. See Tables for further details on the printing these reports.

32.3.2 Modeling Module

In the **Modeling Module**, from the **Reports** tab there are several reports that provide information by model about the active forecast run. Modeling Module and HEC-RTS Forecasts describe the available reports in detail.

32.3.3 Results

When a forecast run is selected, you can view the **Extract** and **Post Reports** choosing them from the **Forecast** menu **Log** secondary menu options. You can open and print the extract report (Figure 1). Choose **Print** from the **File** menu. See Working with Plots for further details about printing.



183 Figure 1 Extract Report

32.3.4 Forecast Run

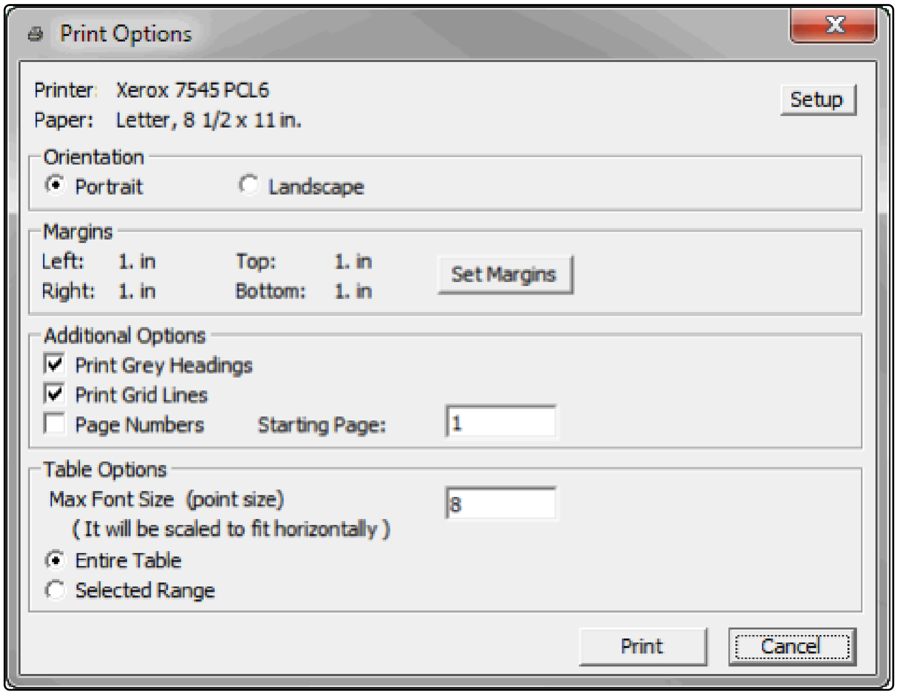
When a forecast run is selected you can choose to view the log file from the **Reports** tab in the **Modeling Module**. Click **Extract Log** and a log extract report opens displaying the report (Figure 1). You can print this report by clicking **Print** from the **File** menu. See Working with Plots for further details about printing.

32.3.5 HEC-ResSim

All of the **HEC-ResSim** reports are printable, and you can export the reports to an ASCII text file. To print HEC-ResSim reports, from the **File** menu of a HEC-ResSim report, click **Print**, and the **Properties dialog** opens. See Tables for further details on printing these reports.

32.3.6 HEC-RAS

The **HEC-RAS** reports can be copied to the clipboard and are printable. To copy to the clipboard, choose **Copy Plot to Clipboard** or **Copy Values to Clipboard** from the **File** menu. To print HEC-RAS reports, from the **File** menu of a HEC-RAS report, click **Print**, and the **Print Options** dialog (Figure 2) opens.

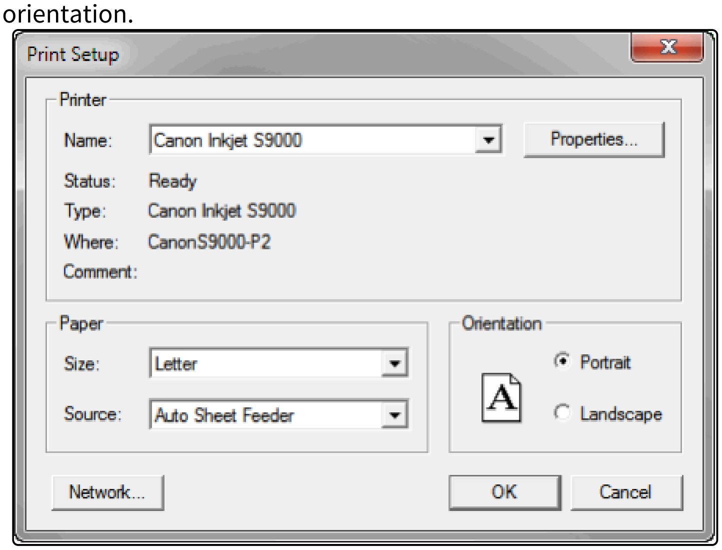


184 Figure 2 Print Options Dialog - HEC-RAS

From this **Print Options** dialog, you can set up the format of the report for printing.

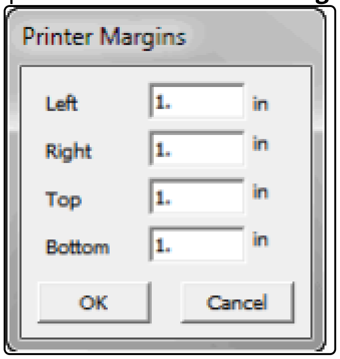
Following is a list of the available print options:

- **Setup:** Click **Setup**, and the **Print Setup** dialog (Figure 3) opens. From the **Print Setup** dialog, you can select a printer and change the printer properties, you can change the paper size and source, and you can set the orientation.



- **Orientation:** As with the **Print Setup** dialog, you set the orientation using the **Print Options** dialog (Figure 3). To set the orientation, select **Portrait** or **Landscape** from the **Orientation** panel. By default, the pages are printed in **Portrait**.

- **Margins:** The default margin settings are displayed in the **Margins** panel of the **Page Setup** dialog. To set the margins, from the **Print Options** dialog (Figure 2), click **Set Margins**, and the **Printer Margins** dialog (Figure 4) opens. From the **Printer Margins** dialog, you can adjust the left, right, top, and bottom margins.



185 Figure 4 Printer Margins Dialog - HEC-RAS

- **Additional Options:** From the **Additional Options** panel in Figure 2, you can choose to **Print Grey Headings**, **Print Grid Lines**, include **Page Numbers**, and set the **Starting Page**.
- **Table Options:** From the **Table Options** panel in Figure 2, you set the **Max Font Size** and choose if you want to print the **Entire Table** or a **Selected Range** of the table.

32.3.7 HEC-FIA

The **HEC-FIA Result Reports** are printable, and you can export the reports to an ASCII text file. From the HEC-FIA reports, click **Print**, the **Properties** dialog will open. See Tables for further details on printing these reports. To export to an ASCII text file, click **Save To File** and the **Table Export Options** dialog will open. See Tables for further details on exporting these reports.

33 Stream Alignment

A common schematic is used to share effectively common data between HEC software (for example, HEC-HMS and HEC-ResSim) for a watershed. Use of this common schematic ensures consistency of computation nodes, routing reaches, and so on.

In the Setup module, you use tools to create the common schematic. Setup module provides an overview of the setup module and the tools available in the module. The tools allow you to depict elements of the common schematic—but not change the configuration of the native models. **Changes made with the tools are not accounted for in the native models' computations.**

The first item to create in the common schematic is the stream alignment. A stream alignment is the backbone for all the other elements that make up a common schematic. This section will describe the tools and concepts for creating a stream alignment.

33.1 Definition of a Stream Alignment

Stream alignments represent the stream network in the watershed. The alignment shows flow direction and indicates where confluences and bifurcations occur. It can also provide a sense of distance and scale. Stream alignments are composed of a series of stream elements, stream nodes, and stream junctions (Figure 1). A stream element is a segment of the stream alignment. A stream element typically begins or ends at a confluence, bifurcation, or at the boundary of the river system. It is composed of vertex points, with the beginning and ending vertex points being stream nodes. Stream junctions are also stream nodes but are located at points where two stream elements meet.
Stream elements are drawn in segments defined by vertex points. Each stream element must have at least two stream nodes, defining the beginning and end of the stream element. Stream elements, stream nodes, and stream junctions, are illustrated in Figure 2.

HEC-RTS creates and labels stream stations (or river stations if the user defines the stationing as such) along each stream element. Stream stationing is a reference system for locating features with distance along a stream. The station at the downstream end of an element is Station 0. The stationing increases upstream along the element according to the linear distance between vertices on the stream element in the watershed coordinate system. 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 and by adding additional nodes along the stream element.

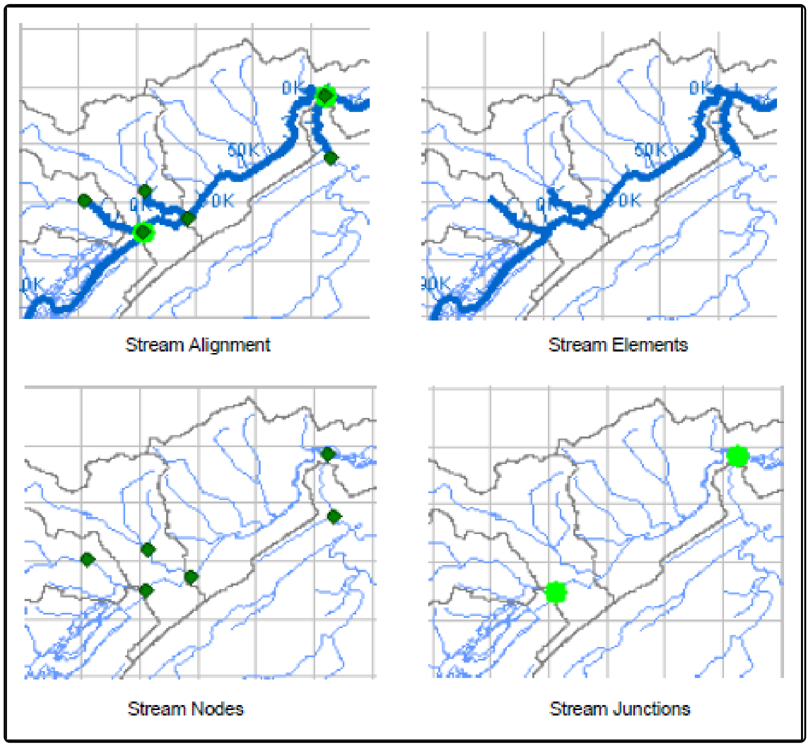
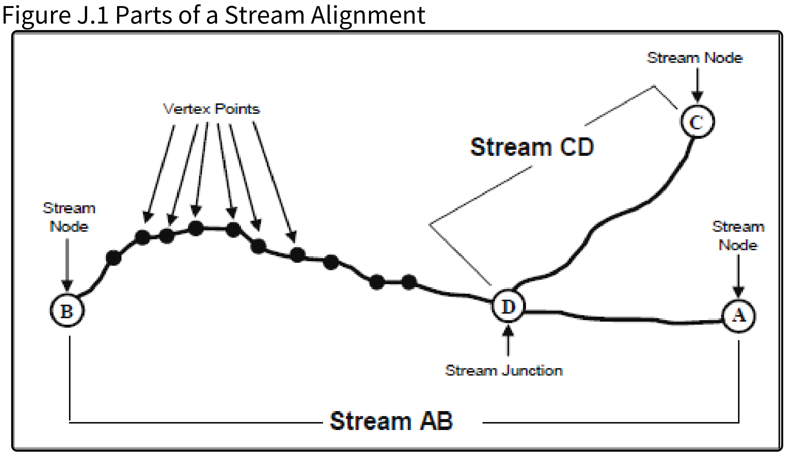
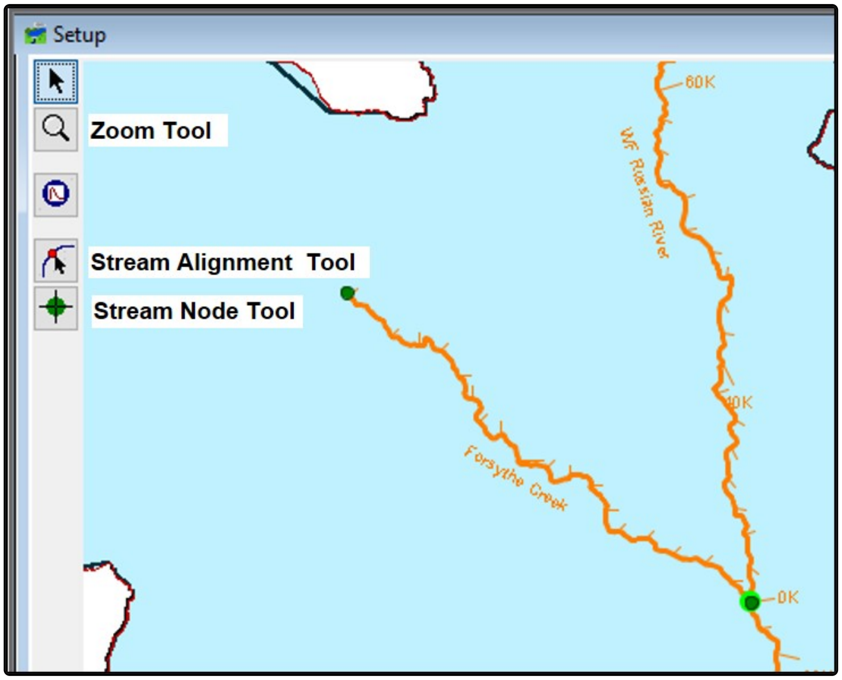


Figure J.1 Parts of a Stream Alignment



Stream stationing is based on the geographic extents that are defined when a stream element is drawn. If the geo-extents are based on a map layer, the units for the stream stationing are the same as the map layer (that is, feet or meters). There is no indication of what the units are. A stream alignment created without an underlying map layer will use the default coordinate system of X-Y. This default system does not have any units associated with it. Setting Up the Coordinate System describes how to set up coordinate systems.

The primary tools used to define the stream alignment are the stream alignment tool and stream node tool (Figure 3). The zoom tool (Setup Module) is also useful. Use it to magnify the stream network. This makes defining the vertices along the stream easier. The three tools are located in the map window toolbar of the setup module.



186 Figure 3 Setup Map Window - Displaying Stream Alignment

If the stream alignment and stream node tools are not displayed in the map window tool bar, click the **Setup** tab. Expand the **Stream Alignments** layer in the layer tree. Double-click the **Stream Alignment** node.

33.2 Creating a New Stream Alignment

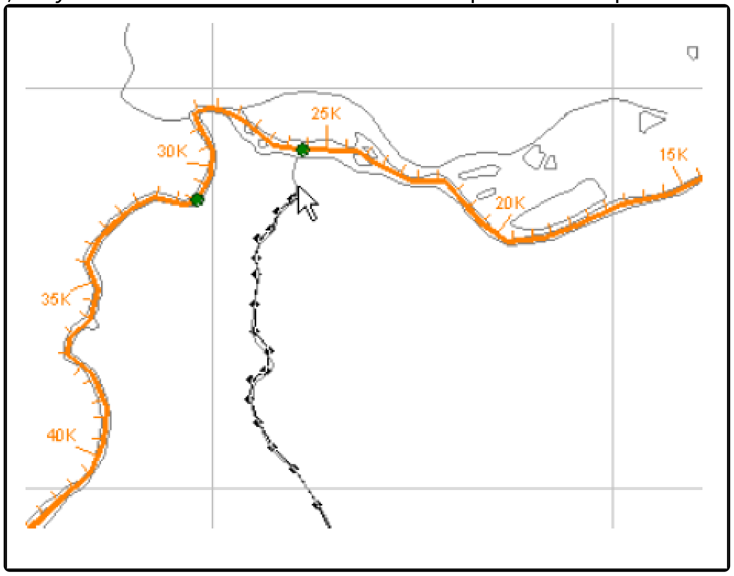
If your stream alignment is going to be based on a map layer, you must first add the map layer to the watershed (Map Layers). The map layer should contain watershed features, such as channels and reservoirs, so that it can be a guide for creating the stream alignment. The units of the labels on the stream alignment will be the same as the units of the map layer. That is, if the map layer is projected in state plane coordinates (feet), then the labels on the stream alignment will be in feet.

You can either create a new stream alignment by hand or import a stream alignment from an Arc shapefile. The following are steps for creating a new stream alignment.

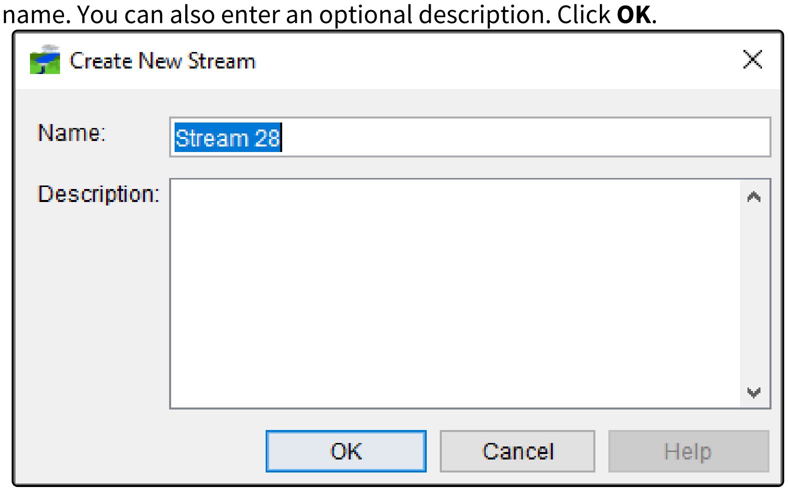
To create a stream alignment:

1. Select the **Stream Alignment Tool** from the map window toolbar.
2. Find a location (typically by viewing a background map that you have added as a map layer) on the map window where you want to start a stream element. **Important:** Stream elements should be created from ***upstream to downstream***. Hold down the **CTRL** key while clicking the mouse button. Each click creates a vertex point (Figure

1) on your stream element. The first vertex point is the upstream or beginning stream node of a stream element.



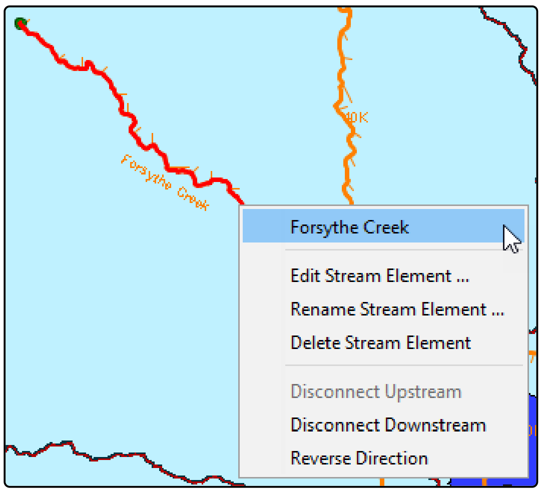
3. For the most downstream or last vertex point, release the **CTRL** key, and then click the mouse button. This creates the ending stream node of the stream element. You have created a single stream element.
4. The **Create New Stream** dialog will open (Figure 2). You can either accept the default name or enter a new name. You can also enter an optional description. Click **OK**.



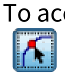
5. Repeat steps 2 through 4 to add more stream elements to your stream alignment.
6. To connect a stream element to an existing stream element, place the first or last click of the new stream element on the existing stream element where you want the two stream elements to join. After naming the new stream element, a dialog will appear asking if you want to connect the new stream element to the existing stream element. Click **Yes**.
7. Once you have completed your stream alignment, save your work. On the file menu, click **Save Watershed**.

33.3 Edit an Existing Stream Alignment

After drawing a stream alignment in the map window, you can rename a stream element, edit a stream element description, delete a stream element, reverse the direction of a stream element, disconnect a stream element at both the upstream or downstream end, and view the stream stationing and coordinates of the stream nodes on the stream element. All of these tasks are accomplished through the stream element shortcut menu (Figure 1).



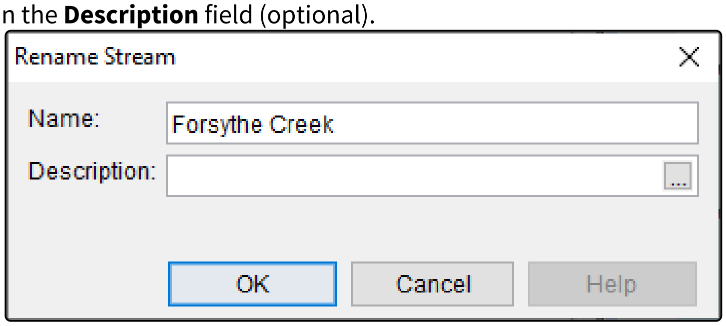
187 Figure 1 Stream Element Shortcut Menu

To access the stream element shortcut menu, use the stream alignment tool to select a stream element and right click  .

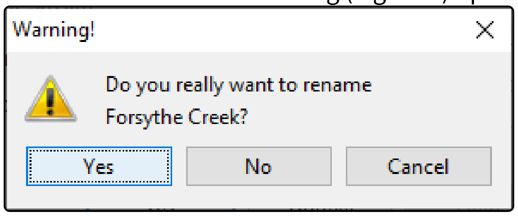
33.3.1 Rename a Stream Element

To rename a stream element:

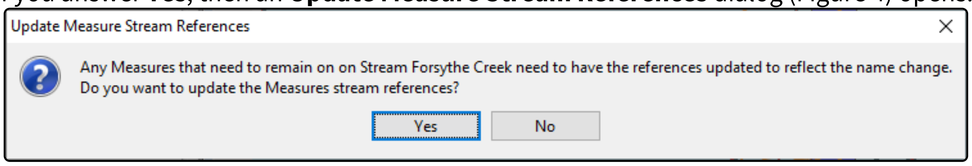
1. Select the stream alignment tool from the map window toolbar.
2. Right-click on the stream element you want to rename, and click **Rename Stream Element** from the shortcut menu.
3. The **Rename Stream** dialog (Figure 2) opens. Enter the new name in the **Name** field (required) and a description in the **Description** field (optional).



4. Click **OK**. A confirmation dialog (Figure 3) opens asking you if you really want to rename the stream.



5. If you answer **Yes**, then an **Update Measure Stream References** dialog (Figure 4) opens.

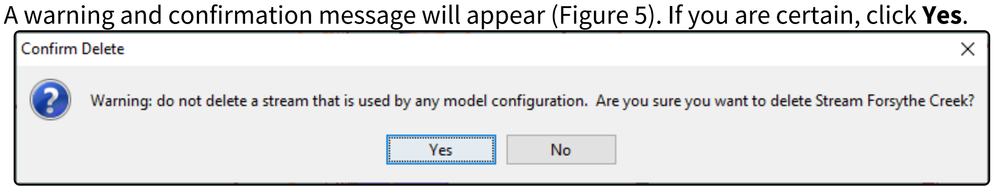


6. If you answer **Yes** on the **Update Measure Stream References** dialog, then the stream element references are updated, and the dialog will close. Otherwise, if you answer **No**, then the stream element references are not updated, and the dialog will close. In either case, the new name of the stream element will be shown in the map window.

33.3.2 Delete a Stream Element

To delete a stream element:

1. Select the stream alignment tool from the map window toolbar.
2. Right-click the stream element you want to delete, and click **Delete Stream Element**.
3. A warning and confirmation message will appear (Figure 5). If you are certain, click **Yes**.



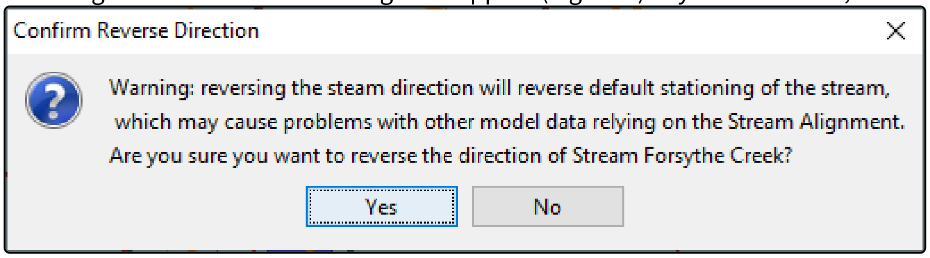
33.3.3 Reverse Direction of a Stream Element

If you draw a stream element in the wrong direction, from downstream to upstream, you can reverse the direction instead of having to delete and redraw the stream element.

To reverse the direction of a stream element:

1. Select the stream alignment tool from the map window toolbar.
2. Right-click the stream element you want to reverse, and click **Reverse Direction**.

3. A warning and confirmation message will appear (Figure 6). If you are certain, click **Yes**.



33.3.4 Disconnect a Stream Element

If you want to disconnect a stream element from another stream element, you may do so at either the upstream or downstream end of the stream element.

To disconnect a stream element from another stream element:

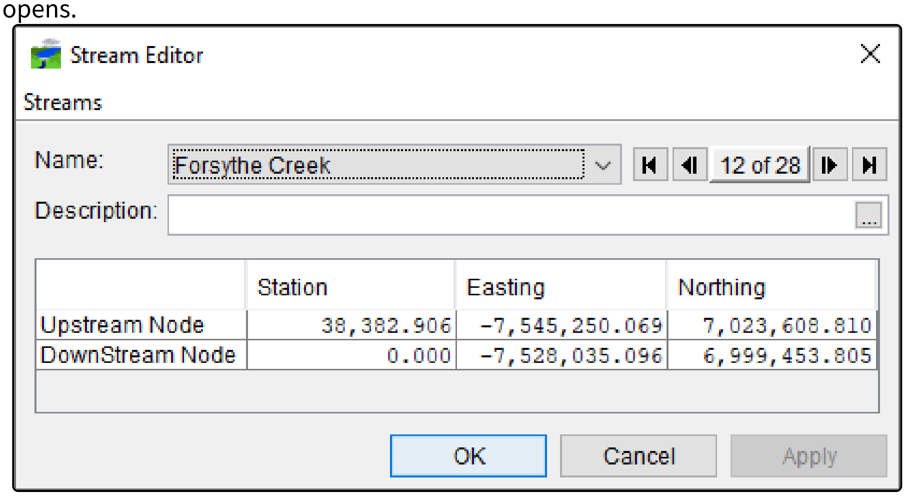
1. Select the stream alignment tool from the map window toolbar.
2. Right-click the stream you want to disconnect, and click either **Disconnect Upstream** or **Disconnect Downstream**.
3. The stream element will be disconnected from the stream element at the location you chose. In the map window, the upstream or downstream point will physically move away from the connecting stream element, and a stream node will remain on the connecting stream element.

33.3.5 Edit a Stream Element Description

The stream editor, available from the shortcut menu of a stream element, allows you to edit the stream element description and view the stream stationing and coordinates of the upstream and downstream stream nodes, as well as any stream nodes in between the two.

To edit the description of a stream element:

1. Select the stream alignment tool from the map window toolbar.
2. Right-click the stream element you want to edit, click **Edit Stream Element**, and the **Stream Editor** (Figure 7) opens.



3. From the name drop-down or the arrow buttons select the name of the stream you want to edit.
4. You can either enter a new description or edit the description of the stream element in the **Description** field.
5. The table provides you with the stream stationing and coordinates of the stream nodes that are on the selected stream element.

33.4 Adjust a Stream Element

After drawing the stream alignment in the map window, there are many options available for adjusting the stream alignment. The zoom tool can be used to help magnify the stream elements. You can refine and re-shape the stream alignment by adding, moving, and deleting vertex points from stream elements. Adjacent vertex points are connected by straight lines. Thus, when you delete a vertex point, the alignment will adjust so that the two vertex points that had surrounded the deleted vertex point are connected by a straight line. The alignment will adjust similarly when you move vertex points.

33.4.1 Add Vertices to a Stream Element

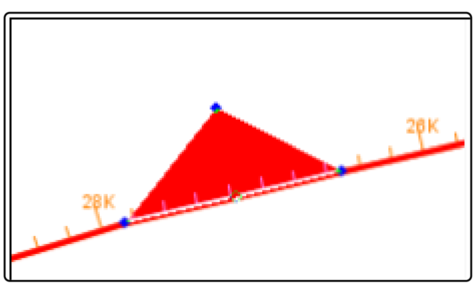
To add vertex points to an existing stream element:

1. Select the stream alignment tool from the map window toolbar.
2. Double-click the stream element that you want to add a vertex point to. The stream element will turn red, and vertex points will be displayed in blue.
3. Place the pointer where you want to add a vertex point. Hold down the **CTRL** key, and click.

33.4.2 Move the Vertices of a Stream Element

To move vertex points on an existing stream element:

1. Select the stream alignment tool from the map window toolbar.
2. Double-click the stream element where you want to move a vertex point. The selected stream element will turn red. The vertex points will be displayed in blue.
3. Click the vertex point you wish to move, and drag it to a new location. This is illustrated in Figure 1.



188 Figure 1 Move a Vertex Point

33.4.3 Delete Vertices from a Stream Element

To delete vertex points from an existing stream element:

1. Select the stream alignment tool from the map window toolbar.
2. Double-click the stream element where you want to delete a vertex point. The selected stream element will turn red. The vertex points will be displayed in blue.
3. Hold down the Shift + CTRL keys, and click on the vertex point to be deleted. The vertex point will disappear from the stream element.

33.5 Stream Nodes

Stream nodes are part of the representation of the stream network in the watershed. HEC-RTS creates a stream node at the beginning and ending of each stream element and where stream elements meet (stream junctions). Stream nodes are used to establish stationing for a stream. To modify the default stream stationing, you must edit the stream nodes.

Though they appear similar, stream nodes are not the same as computation points. A computation point is a location where time-series data and information is exchanged between analysis software. Stream nodes are useful in placing computation points.

33.5.1 Add a Stream Node

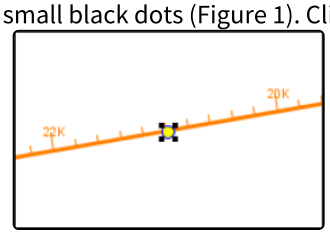
As noted above, two stream nodes are created automatically at the beginning and end of a stream element. You can add stream nodes to a stream element:

1. Select the stream node tool from the map window toolbar.
2. Place the pointer on the stream element where you want to add a stream node. Hold down the **CTRL** key, and click.

33.5.2 Move a Stream Node

To move a stream node:

1. Select the stream node tool from the map window toolbar.
2. Double-click the stream node that you want to move. It becomes a yellow circle with a black outline and four small black dots (Figure 1). Click and drag the stream node along the stream element to its new position.

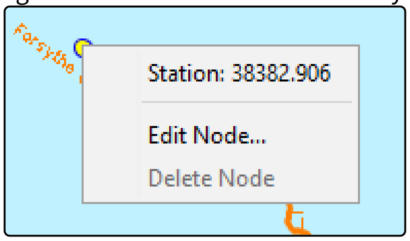


33.5.3 Edit Stream Node to Establish Stream Stationing

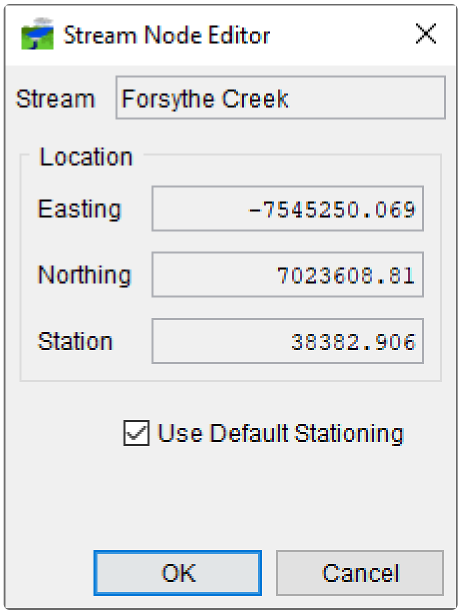
After placing a stream node on the stream alignment, you can edit the stream node to establish stream stationing for a stream element. To edit a stream node:

1. Select the stream node tool from the map window toolbar.

2. Right-click on the stream node that you want to edit. A shortcut menu will appear (Figure 2).



3. Click **Edit Node**. The **Stream Node Editor** (Figure 3) opens.



4. The default stream station for the downstream node is 0. The stream station for the upstream node is based on the coordinate system you used when digitizing the stream alignment. To change the default stream stationing, clear **Use Default Stationing**. Edit the station in the **Station** field.
5. Click **OK**. Based on what station number you enter; HEC-RTS will automatically adjust the stream stationing for the stream element. This includes the upstream and downstream nodes.

33.5.4 Delete Stream Node

To delete a stream node:

1. Select the stream node tool from the map window toolbar.
2. Right-click on the stream node that you want to delete. Click **Delete Node**.
3. A confirmation message opens. If you are certain, click **Yes**.

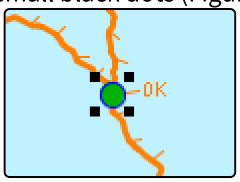
33.6 Stream Junction

When two stream elements are connected, a stream junction is created. A stream junction is where multiple stream nodes exist at one location. A stream junction is represented by a dark green circle with a light green halo around the circle. You can move a stream junction and edit a stream junction to adjust its stream stationing.

33.6.1 Move a Stream Junction

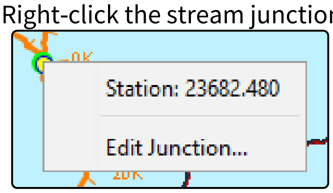
To move a stream junction:

1. Select the stream node tool from the map window toolbar.
2. Double-click the stream junction that you want to move. It becomes a green circle with a black outline and four small black dots (Figure 1). Click and drag the stream junction along the stream element to its new position.

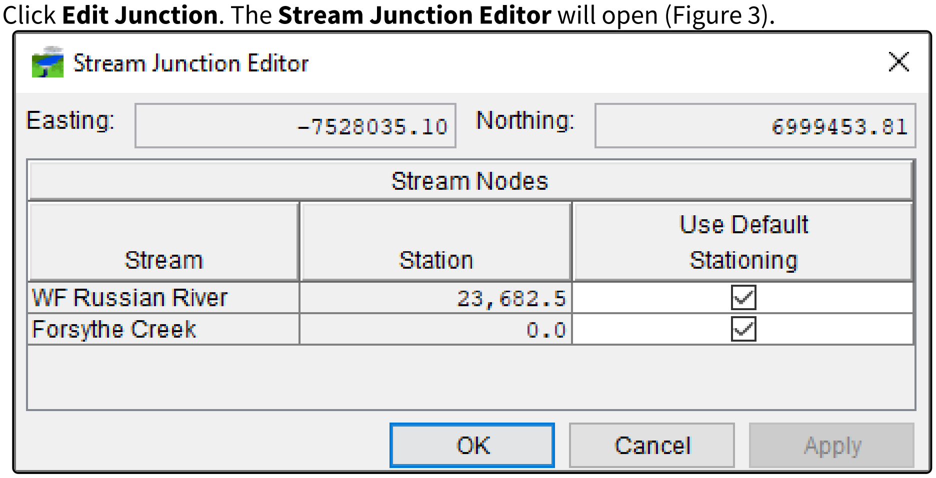


33.6.2 Edit a Stream Junction To edit a stream junction:

1. Select the stream node tool from the map window toolbar.
2. Right-click the stream junction that you want to edit. A shortcut menu (Figure 2) will appear.



3. Click **Edit Junction**. The **Stream Junction Editor** will open (Figure 3).



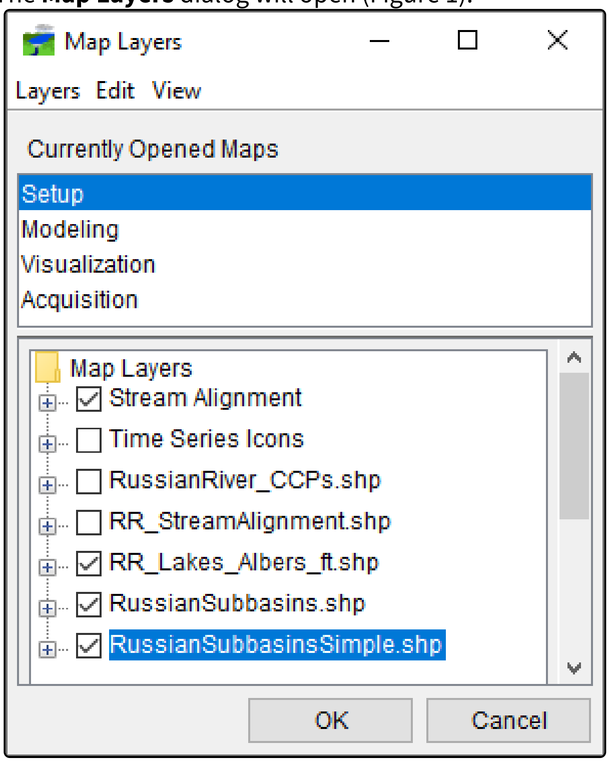
4. The editor displays information about the stream nodes associated with the stream junction. To change the stream stationing for a stream node, clear the checkbox for the stream node in the **Use Default Stationing** column of the table. The cell in the **Station** column will become active. Based on what you enter, HEC-RTS will adjust the stream stationing for the stream associated with the stream node automatically. Click **OK**.

33.7 Change Stream Alignment Properties

Stream alignment properties that can be changed include the color and width of stream elements, stream nodes, and stream junctions, font for the stream name, tic mark properties, and visualization scaling for the stream alignment.

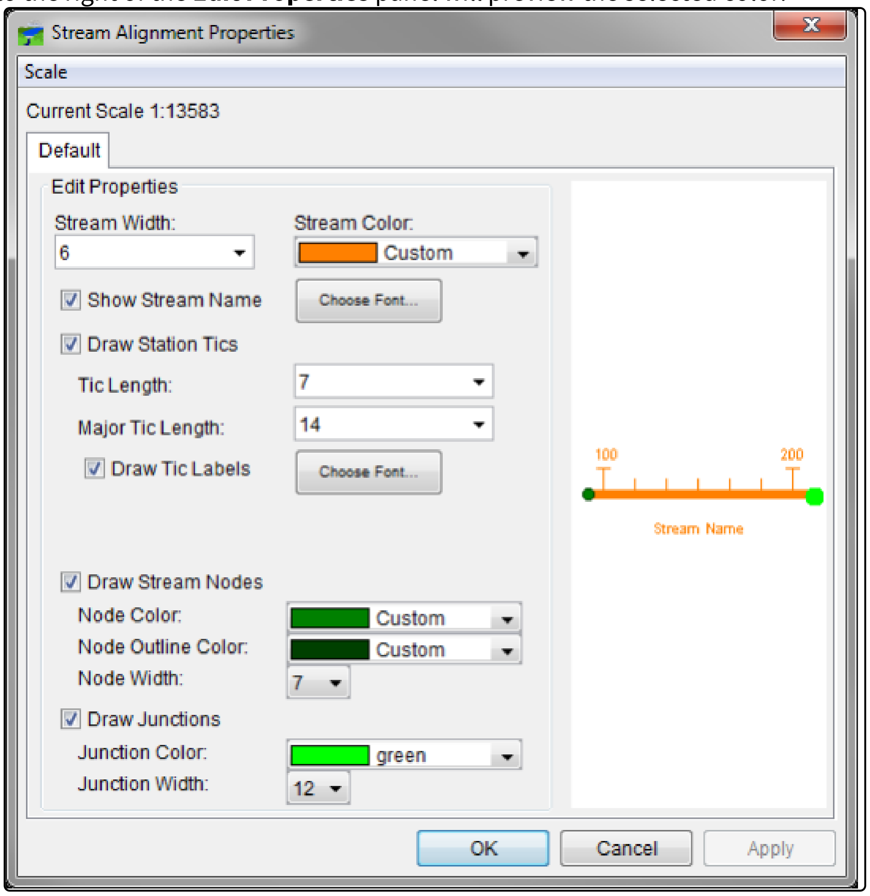
To change stream alignment properties:

1. In the setup module, click **Map Layers** on the **maps** menu.
2. The **Map Layers** dialog will open (Figure 1).



3. Right-click the **Stream Alignment** layer in the tree. Click **Properties**.
4. The **Stream Alignment Properties** dialog will open (Figure 2).

5. To change the color of the stream alignment layer, select a color from the **Stream Color** list. The preview panel to the right of the **Edit Properties** panel will preview the selected color.



6. Click **OK,** and the color change will appear in the map window.
7. Repeat steps 5 and 6 for any other changes you wish to make to the stream alignment properties.
8. To set the visualization scale, click the **Scale** menu.

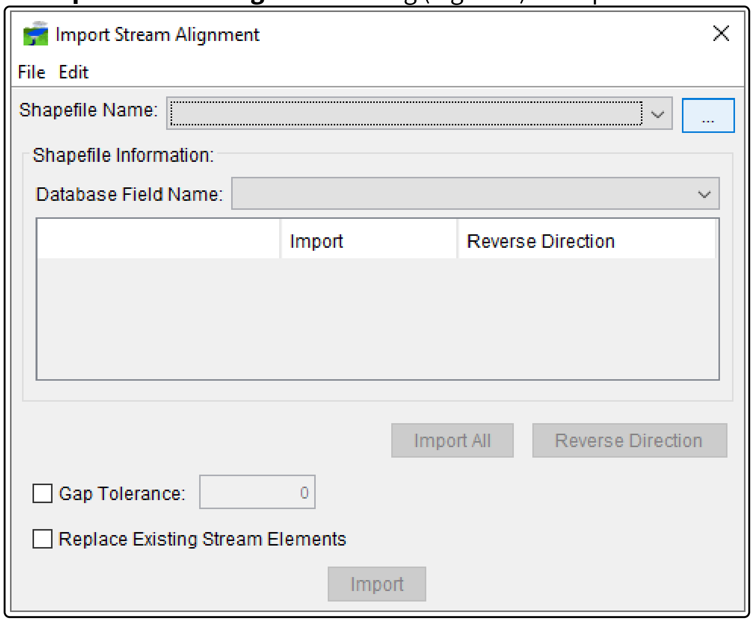
33.8 Import a Stream Alignment

To import a stream alignment, you must have added an Arc shapefile (Map Layer Formats) map layer (Adding Map Layers) that represents the stream network of your watershed. This shapefile must have an attribute that is the name of each stream, and each stream must form a contiguous line with one upstream point and one downstream point.

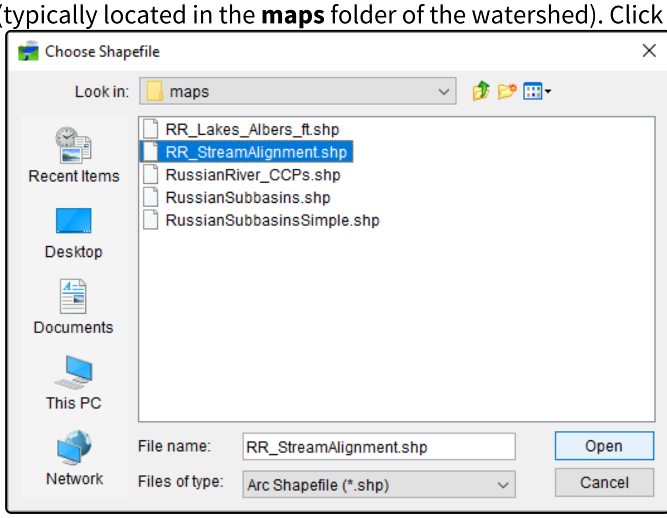
To import a stream alignment:

1. From the **Setup** module, from the **Watershed** menu, point to **Stream Alignment**, click **Import**.

2. The **Import Stream Alignment** dialog (Figure 1) will open.

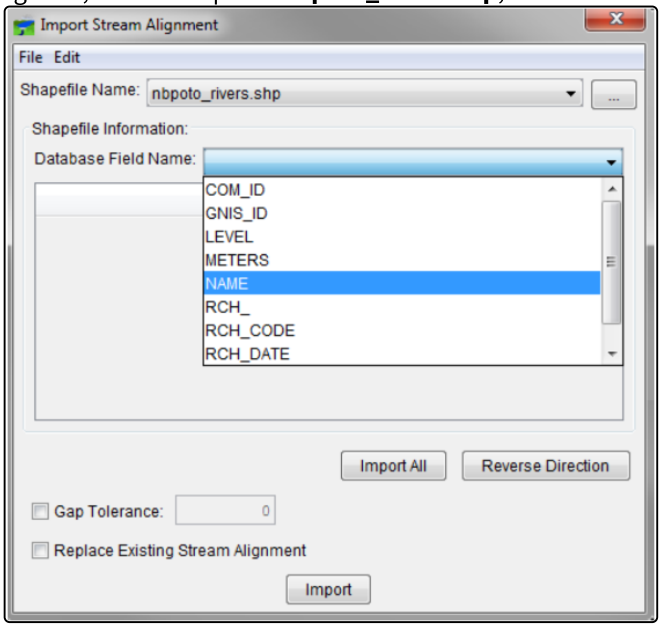


3. In the **Shapefile Name** drop-down list, the name of the ArcGIS® shapefile that HEC-RTS has chosen is displayed. HEC-RTSS automatically selects the first shapefile that contains polylines. If this is not the correct shapefile, on the **File** menu, click **Choose Shapefile**.
4. A browser **Choose Shapefile** dialog will open (Figure 2). Use the dialog to select the shapefile to be imported (typically located in the **maps** folder of the watershed). Click **Open**.

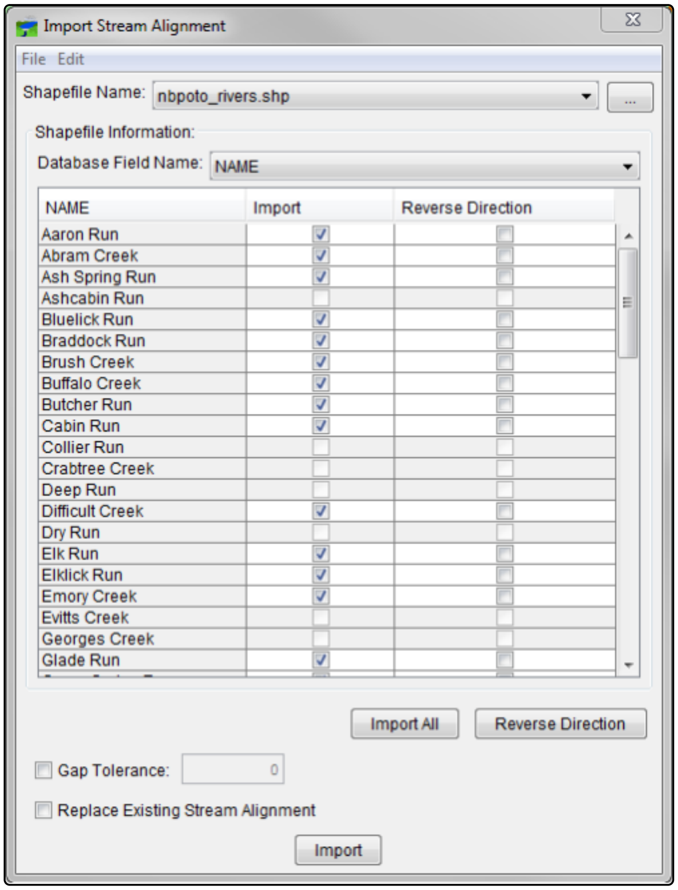


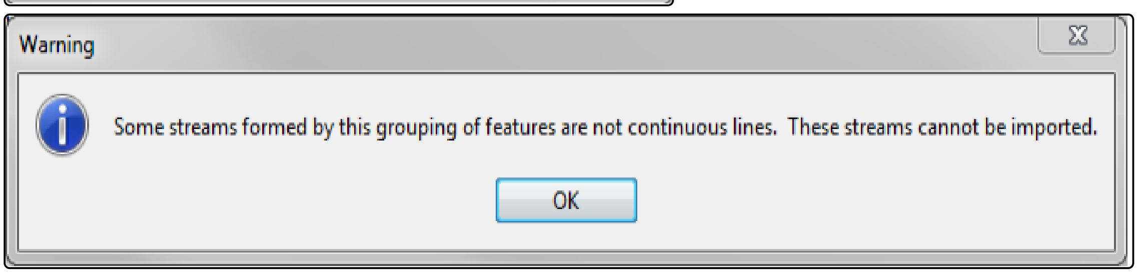
5. Once the correct shapefile has been chosen, the program automatically searches for a database field name of **Stream_ID**. If there is no **Stream_ID** in the shapefile you chose, you must select the database field name that contains the names of each stream from the **Database Field Name** drop-down list. In the example shown in

Figure 3, for the shapefile **nbpoto_rivers.shp**, the database field name is **NAME**.



6. When you have selected the correct database field name, the table will list the stream names (Figure 4). HEC-RTS requires a stream to be a contiguous line with one upstream point and one downstream point. If a stream is a contiguous line, the checkbox in the **Import** column of the table will be set. If the line that represents a stream is non-contiguous, the checkbox will not be set, and a **Warning** message (Figure 5) will appear indicating that the lines are not continuous and that the streams cannot be imported. Click **OK** to close the message and to continue defining the import options.





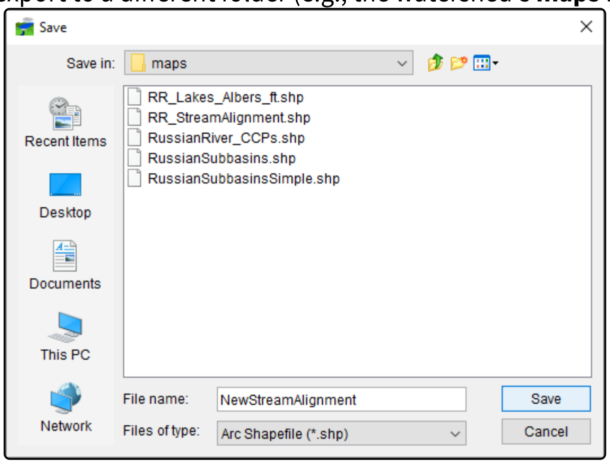
7. You can use the **Import All** and **Reverse Direction** buttons to select each of the checkboxes in the **Import** and **Reverse Direction** columns, respectively.
8. Gap tolerance is used to connect stream elements that have a gap between the end point and the junction with another stream. By default, the gap tolerance is set to zero, so HEC-RTS can connect streams to other streams. To allow gap tolerance, check the **Gap Tolerance** checkbox. Enter a value larger than zero in the **Gap Tolerance** field. HEC-RTS will connect streams that have a gap between them that is equal to or less than this value.
9. The **Replace Existing Stream Alignment** check box allows you to replace the existing stream alignment with the one you are importing.
10. Once everything is set, click **Import**. The stream alignment will be automatically drawn in the map window.
11. Click **OK** on the confirmation dialog.

33.9 Export a Stream Alignment

If you have digitized a stream alignment, you can save that stream alignment as an Arc shapefile. To export a stream alignment as a shapefile, you must have a stream alignment displayed in the map window.

To export a stream alignment as an Arc shapefile:

1. From the **Setup** module, from the **Watershed** menu, point to **Stream Alignment**, click **Import**.
2. A browser will open so that you can choose where the shapefile should be saved (Figure 1). Navigate to the appropriate folder. The default stream alignment is stored in the watershed's main folder, but you can choose to export to a different folder (e.g., the watershed's **maps** folder). Enter a name in the **File name** field. Click **Save**.



33.10 Save Stream Alignment

To save your stream alignment, on the **File** menu click **Save Watershed**. This command will save all of the stream alignment revisions and all other revisions you have made to your watershed.

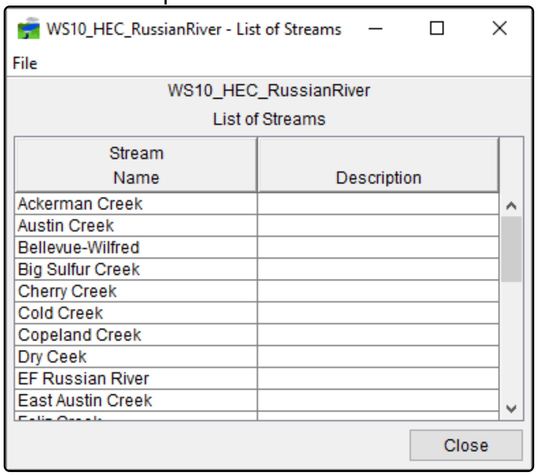
33.11 List of Streams

Once you have defined your stream alignment, you can review the streams included.

To view the **List of Streams** report:

1. In the **Setup** module, on the **Watershed** menu, choose **List of Streams**. The **List of Streams** report will open (Figure 1). The report provides a list of streams that have been input for the stream alignment, along with the

stream's description.



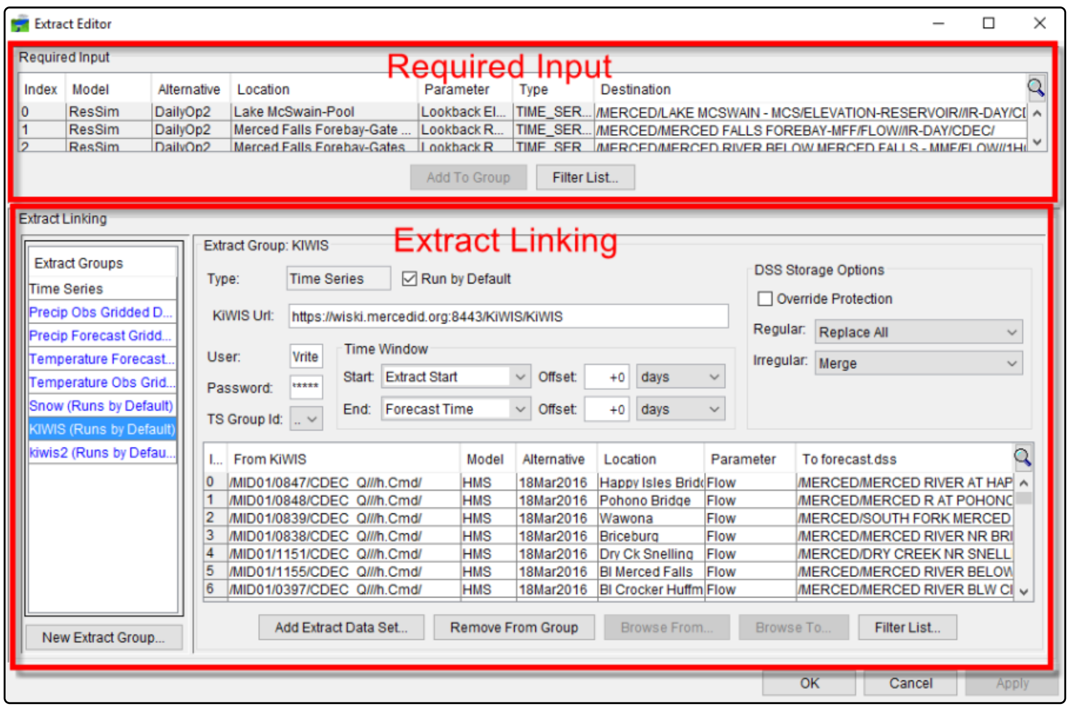
2. Using the **File** menu, you can print the report or export the report to an ASCII tab delimited file.

34 WISKI Extract and Post

34.1 WISKI Extract Post Integration

Kisters WISKI is a relational database designed to acquire, store, and validate data for water management. WISKI is comprised of an SQL database and a web application (KiWIS). HEC-RTS provides access to the WISKI database by using the web application KiWIS. WISKI data can be used to provide input data to an HEC-RTS forecast. HEC-RTS can also disseminate data from a *forecast.dss* file to a WISKI database through KiWIS. Together, the system can be used to edit and transform data.

This appendix provides instructions on how to use the WISKI database with HEC-RTS and includes: retrieving data from a WISKI database to provide input for a forecast compute using the **Extract Editor** (Figure 1), and disseminating forecast data to the WISKI database using the **Post Editor**.



189 Figure 1 Extract Editor

34.2 Extracting WISKI Time Series Data

An **extract** refers to the process of obtaining data from either a DSS file or a WISKI database and mapping that data to a model alternative's required input record. The extract retrieves the source data and copies it into records stored in a centralized DSS file called *forecast.dss*. The *forecast.dss* file is then used to provide input data during a forecast's compute process. While DSS records can contain both gridded data and time series data, the WISKI connection currently supports time series data. The **Extract Editor** is used to set up extract groups that retrieve data from a WISKI database.

The **Extract Editor** allows you to connect required model alternative input data from multiple sources to the *forecast.dss* file that provides input data to the models during a forecast compute. The required input data is classified into extract groups that link the source data information to the required input records. Information about the **Extract Editor** and extract groups is detailed in Data Extract and Post Editors. This appendix will provide information on WISKI and KiWIS details.

34.3 WISKI Extract Editor

The **Extract Editor** allows you to connect required model alternative input data from multiple sources to the *forecast.dss* file that provides input data to the models during a forecast compute. The required input data is classified into extract groups that link the source data information to the required input records. Information about the **Extract Editor** and extract groups is detailed in Data Extract and Post Editors. This appendix will provide information on WISKI and KiWIS details.

34.4 KiWIS Extract Group

When an extract group is selected that is a KiWIS extract group, information displayed in the **Extract Linking** section (Extract Editor) is the data **Type**, **KiWIS Url**, and the **User** name and **Password** needed to access a WISKI database. The

TS Group Id lists all the group ID's in a WISKI database. The **TS Group Id** can be used to filter WISKI records, which will increase the speed of the query process for large databases when accessing them through HEC-RTS.

The **Extract Linking** table (Extract Editor) displays the following information when a KiWIS extract group has been selected:

From KiWIS	This column displays the source data pathnames that's data will fill the DSS record displayed in the **To forecast.dss** column.
Model	This column lists which software application is requesting the required input record.
Alternative	This column lists the model alternative name that is requesting the input record.
Location	This column displays the name of the model location object that is associated with the input record.
Parameter	This column displays the parameter associated with the input record's data.
To forecast.dss	This column dipslays the actual input record pathname that the model requires. This is the pathname that will be written to the *forecast.dss* file and used by a model during the forcast compute.

Index	Each required destination pathname is assigned an **Index** number when it is loaded into the **Extract Editor**. This number has no real usage except keeping the initial filtered list set. There can be multiple indices for a single DSS pathname.

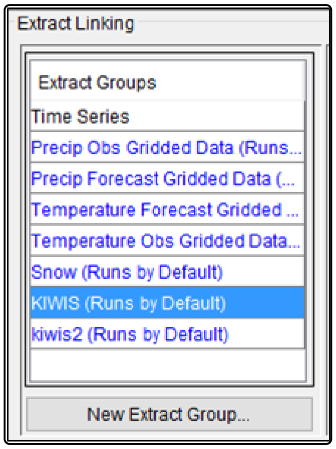
This information does not display by default in the **Extract Linking** table, you will need to set up the table view, see Extract Editor.

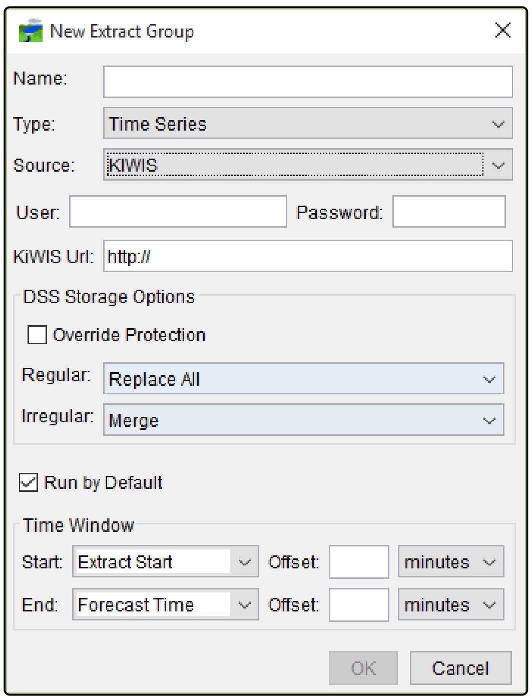
34.4.1 Create a KiWIS Extract Group

A KiWIS extract group maps time series data from the WISKI database to the required model alternative's input time series records that will be stored in the forecast.dss file. You will need the KiWIS URL, a username, and a password to set up a KiWIS extract group.

To create a KiWIS extract group:

1. From the HEC-RTS main window, click on the **Setup** tab, from the **Models** menu, click **Edit Extract**, the **Extract Editor** will open (Extract Editor).
2. From the **Extract Editor**, from the **Extract Groups** box (Figure 1), click **New Extract Group.** The **New Extract Group** dialog will open (Figure 2).





3. In the **Name** box (Figure 2), enter a name for your KiWIS extract group.
4. From the **Type** list (Figure 2), select **Time Series** for the data being extracted and mapped.
5. From the **Source** list (Figure 2), select **KIWIS**.
6. From the **User** box, enter your user login (Figure 2) and from the **Password** box, enter the password that is associated with the entered user login. Both the username and password are case sensitive. From the **KiWIS Url** box, enter the url for the WISKI database.
7. The items in the **DSS Storage Options** box (Figure 2) are only applicable to Oracle and DSS source files. No changes to these values are necessary when using KiWIS as your data source.
8. By default, **Run by Default** (Figure 2) is selected, which means the extract group will run automatically when a forecast is being computed.
9. Specify the **Time Window** to extract the data from the **Start** and **End** lists (Figure 2); available options are: **Extract Start, Start of Simulation, Forecast Time, End of Simulation**. Offsets for the start and end times can also be included, available options are: **hours**, **minutes**, or **seconds**. In the **Offset** box, you will enter the offset

increment. This time window is relative to the forecast time window the you defined when you created the forecast.

10. When you are done, click **OK,** the **New Extract Group** dialog will close (Figure 2). The new extract group will display at the bottom of the **Extract Groups** box (Figure 1), and the information about the extract group will display.

Note: Extract groups are run in the order they are displayed in the ***Extract Groups*** *box. If you run more than one extract group during the extract process and there are records that are mapped in more than one of the selected extract groups, the last extract group mapping will replace all previous mappings.*

34.4.2 Adding Required Input Data to Extract Groups

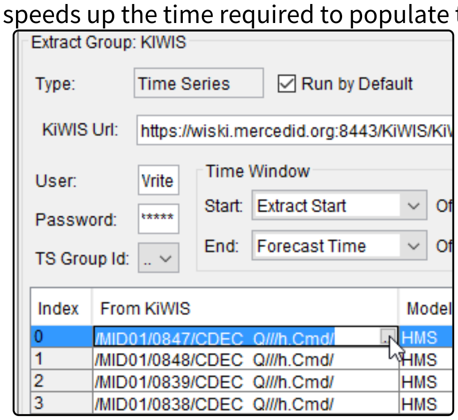
Once an extract group is defined, you can select what **Required Input** data you want to map to the selected source data. In this case the source data is the WISKI database. Detailed information is provided in **Extract Groups**.

34.4.3 Selecting KiWIS Source Records

When **Required Input** records are added to a KiWIS extract group, they display the name of the required input record under the **From KiWIS** column in the **Extract Linking** table (Extract Editor). The **From KiWIS** record pathnames need to be replaced with WISKI database source time series records that will provide data to the **To forecast.dss** column of the **Extract Linking** table (Extract Editor).

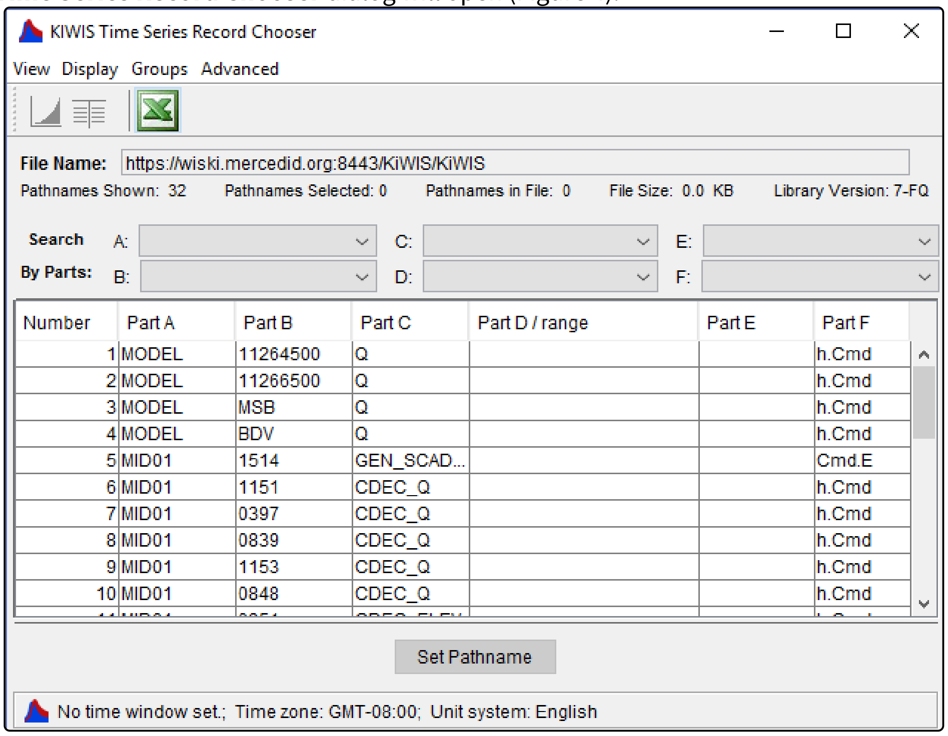
To select a WISKI database time series record for the **From KiWIS** record:

1. The **TS Group Id** list (Figure 3), in the **Extract Linking** section, provides a list of all the time series groups created in KiWIS. The **TS Group Id** is used in the **Extract Editor** to manage the size of a WISKI database request and speeds up the time required to populate the list of records when selecting KiWIS records for the extract.



2. Double-click the row in the **From KiWIS** column (Figure 3) you would like to set the KiWIS time series record for. Click the ellipse that appears in the right section of the cell with the **From KiWIS** name (Figure 3). The **KiWIS**

Time Series Record Chooser dialog will open (Figure 4).



3. You can filter the KiWIS pathnames displayed in the table by using the **Search By Parts Parameters** at the top of the **KiWIS Time Series Record Chooser**.
4. To select a KiWIS time series record to replace the one displayed in the selected row back on the **Extract Linking** table, click the time series record row in the **KiWIS Time Series Record Chooser** dialog (Figure 4), click **Set Pathnames**.
5. The WISKI database time series record you selected will display back on the **Extract Linking** table in the row you original selected.
6. To set another KiWIS time series record, click on the next row in the **Extract Linking** table where you would like to replace the record. From the **KiWIS Time Series Record Chooser** dialog (Figure 4), click on the WISKI database time series record you want, click **Set Pathname**.
7. When all of the KiWIS time series records have been set in the **Extract Linking** table, from the **KiWIS Time Series Record Chooser** dialog (Figure 4), from the **File** menu, click **Close**.

34.4.4 Ad-Hoc Records - Extract

Additional input data not listed in the **Required Extract** table may be required for a forecast compute. You can add additional time series records from the WISKI database using the **Ad-Hoc Record** feature. An Ad-Hoc record allows you to add WISKI database time series records to a DSS time series record in the *forecast.dss* file.

To add a WISKI Ad-Hoc record:

1. From the **Extract Editor**, click **Add Extract Data Set**, the **Adhoc: KiWIS Time Series Record Chooser** dialog will open.
2. From the **Search By Parts** area, use the various filter lists to locate the desired WISKI database time series record.

3. Select the pathname row for the record, click **Set Pathname**.
4. Once you have selected all the pathnames for all the necessary time series records, from the **File** menu, click **Close**. The **Adhoc: KiWIS Time Series Record Chooser** dialog will close.

The **From KiWIS** and **To forecast.dss** time series records (**Extract Linking** table) will display the same pathname. The time series record pathname displayed in the **To forecast.dss** column of the **Extract Linking** table will need to be changed to a conventional DSS pathname. You can either enter the DSS pathname, or select one through the **KiWIS Time Series Record Chooser** dialog (Figure 4).

To select a DSS pathname using the **KiWIS Time Series Record Chooser**:

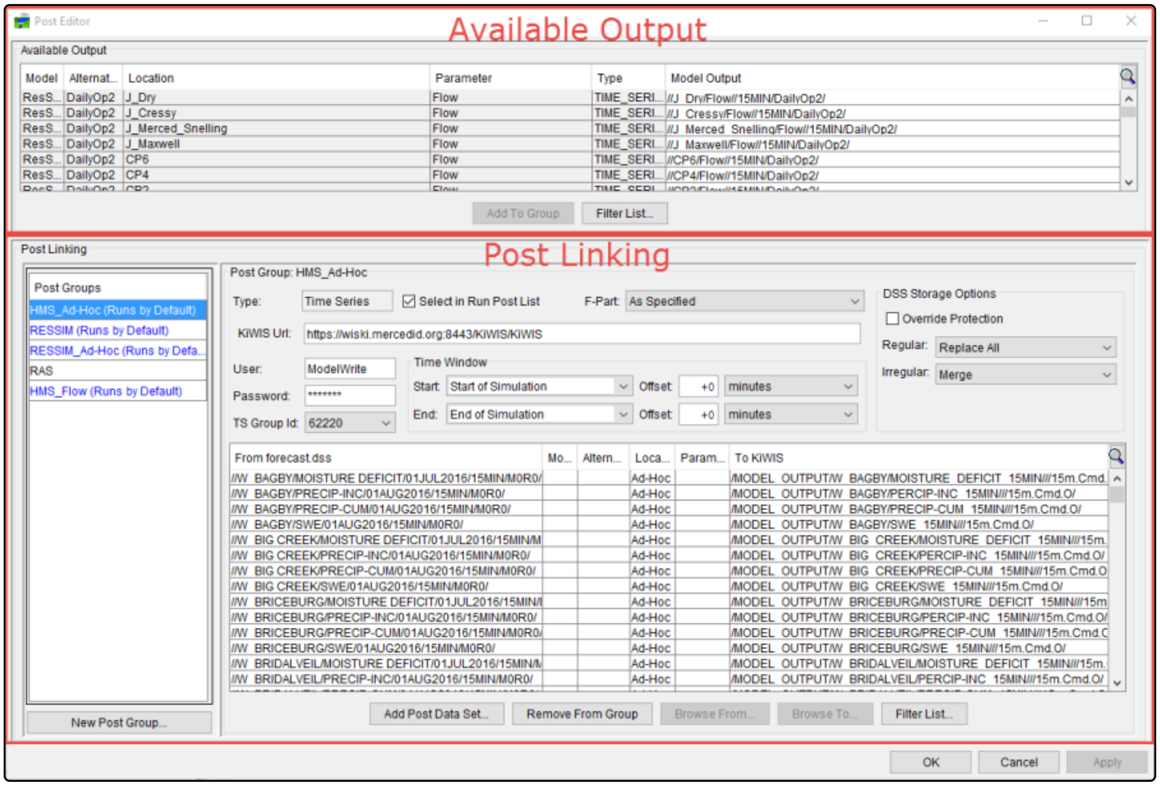
1. From the **Extract Editor**, from the **Extact Linking** table, select the **To forecast.dss** record you would like to change.
2. Click **Browse To**, the **KiWIS Time Series Record Chooser** dialog will display (Figure 4).
3. From the **KiWIS Time Series Record Chooser** dialog (Figure 4). from the **File** menu, click **Open**. An **Open HEC-DSS File** browser will open, browse to location of the DSS file you will be copying time series record to, select the filename, click **Open**. The **Open HEC-DSS File** browser will close, the DSS file is displayed.
4. Select the pathname for the DSS record listed in the table, click **Set Pathname**. This will copy the selected record to the **Extract Linking** table (Extract Editor).
5. When you are finished selecting the DSS pathnames, from the **File** menu, click **Close**. The **KiWIS Time Series Record Chooser** dialog will close (Figure 4).

34.4.5 Running Extracts

Now that you have defined and setup extract groups, you can run extracts. HEC-RTS provides two ways to run extracts - automatic or manually. Running Extracts provides details.

34.5 Posting WISKI Time Series Data

A **post** refers to the process of saving data from the *forecast.dss* file to either a DSS file or a WISKI database. A post selects an output time series from the *forecast.dss* file and disseminates it in an external data location. Compute output can be stored as historical results to be viewed or used in future forecasts. HEC-RTS supports posts for DSS gridded data and time series data, and WISKI time series data. The **Post Editor** (Figure 1) is used to create and define post groups that disseminate data to a WISKI database.



190 Figure 1 Post Editor

34.6 Post Editor with WISKI

Now that you have successfully computed a forecast and reviewed results, you might want to save data from the *forecast.dss* file to other DSS files. This task is done from the **Post Editor** Information about the **Post Editor** and post groups is detailed in Data Extract and Post Editors. This appendix will provide information on WISKI and KiWIS details.

34.7 KiWIS Post Group

When a post group is selected that is a KiWIS post group, information displayed in the **Post Linking** section (Post Editor) is the data **Type**, **KiWIS Url**, and the **User** name and **Password** needed to access the KiWIS database. The **TS Group Id** lists all the time series group ID's that are in the KiWIS database. The **TS Group Id** can be used to filter KiWIS records, which will increase the speed of the query process for large databases when accessing them through HEC-RTS. The **Post Linking** table (Post Editor) displays the following information when a KiWIS post group has been selected:

From forecast.dss	this column lists the selected model alternative output records for the post group. This data will fill with the KiWIS record displayed in the **To KiWIS** column.
Model	this column lists which software application is generating the output.

Alternative	this column lists the model alternative name that is creating the output record.
Location	this column displays the name of the model location object that is associated with the output record.
Parameter	this column displays the parameter associated with the output record's data.
To KiWIS	this column shows the actual input record pathname that the output record will be written to during the post process.
Index	Each required destination pathname is assigned an **Index** number when it is loaded into the **Post Editor**. This number has no real usage except keeping the initial filtered list set. There can be multiple indices for a single DSS pathname.

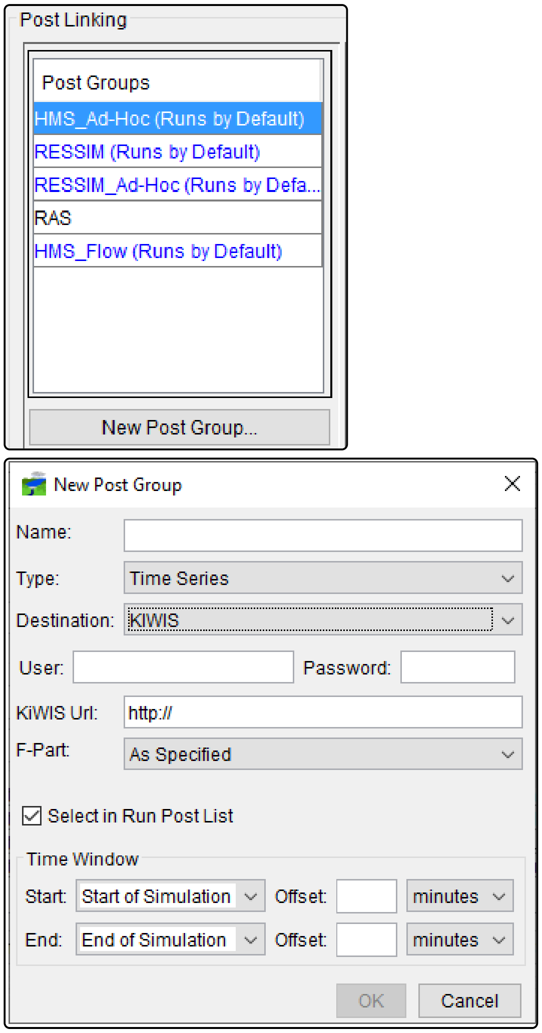
This information does not display by default in the **Post Linking** table, you will need to set up the table view, see Post Editor.

34.7.1 Create a KiWIS Post Group

A KiWIS post group maps the model alternative output time series data from a *forecast.dss* file to a WISKI database. You will need the KiWIS URL, a username, and a password to set up a KiWIS post group. While a TS Group ID is not required, it can simplify the setup process.

To create a KiWIS post group:

1. From the HEC-RTS main window, click on the **Setup** tab, from the **Models** menu, click **Edit Post**, the **Post Editor** will open.
2. From the **Post Editor**, from the **Post Groups** box (Figure 1), click **New Extract Group**. The **New Extract Group** dialog will open (Figure 2).



3. In the **Name** box (Figure 2), enter a name for your KiWIS post group.
4. For a post group the only data type is **Time Series**.
5. From the **Destination** list (Figure 2), select **KIWIS**.
6. From the **User** box, enter your user login (Figure 2) and from the **Password** box, enter the password that is associated with the entered user login. Both the username and password are case sensitive. From the **KiWIS Url** box, enter the url for the WISKI database.
7. The items in the **DSS Storage Options** box (Figure 2) are only applicable to Oracle and DSS source files. No changes to these values are necessary when using KiWIS as your data source.
8. By default, **Select in Run Post List** (Figure 2) is selected, which means the post group will run automatically when a post process is running.
9. Specify the **Time Window** to post the data from the **Start** and **End** lists (Figure 2); available options are: **Extract Start, Start of Simulation, Forecast Time, End of Simulation**. Offsets for the start and end times can also be included, available options are: **hours**, **minutes**, or **seconds**. In the **Offset** box, you will enter the offset increment. This time window is relative to the forecast time window the you defined when you created the forecast.
10. When you are done, click **OK,** the **New Post Group** dialog will close (Figure 2). The new post group will display at the bottom of the **Post Groups** box (Figure 1), and the information about the post group will display.

Note: Post groups are run in the order they are displayed in the ***Post Groups*** *box. If you run more than one post group during the post process and there are records that are mapped in more than one of the selected post groups, the last post group mapping will replace all previous mappings.*

34.7.2 Adding Output Data to Post Groups

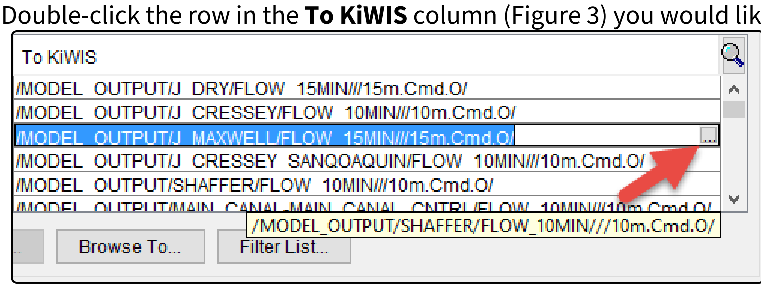
Once a post group is defined, you can select which model alternative output data you want to map to the selected destination data. In this case the destination data is the WISKI database. Detailed information is provided in Post Groups.

34.7.3 Selecting KiWIS Destination Records

When **Available Output** records are added to a KiWIS post group, the name of **Available Output** record, is displayed in the **Post Linking** table (Post Editor), under the **To KiWIS** column. The **To KiWIS** record pathnames need to be replaced with WISKI database destination time series record names.

To select a WISKI database time series record for the **To KiWIS** record:

1. The **TS Group Id** list (Post Editor), in the **Post Linking**, provides a list of all the time series groups created in KiWIS. The **TS Groupd Id** is used in the **Post Editor** to manage the size of the WISKI database request and speed up the time required to populate the list of records when selecting KiWIS records for the post.
2. Double-click the row in the **To KiWIS** column (Figure 3) you would like to set the KiWIS time series record for.



3. Click the ellipse that appears in the right side of the cell with the To KiWIS name. The **KiWIS Time Series Record Chooser** dialog will open.
4. You can filter the KiWIS time series pathnames displayed in the table by using the **Search By Parts Parameters** at the top of the **KiWIS Time Series Record Chooser**.
5. To select a KiWIS time series record to replace the one displayed in the selected row in the Post Linking table (Post Editor), select the time series record row in the **KiWIS Time Series Record**, click **Set Pathnames**.
6. The WISKI time series record you selected will display back on the **Post Linking** table in the row you original selected.
7. To set another KiWIS time series record, from the **Post Linking** table click on the WISKI time series record you want to replace. From the **KiWIS Time Series Record Chooser** dialog, click **Set Pathname**.
8. When all of the KiWIS time series records have been set in the **Post Linking** table, from the **KiWIS Time Series Record Chooser** dialog, from the **File** menu, click **Close**.

34.7.4 Ad-Hoc Records - Post

Additional output data not listed in the **Available Output** table may be needed from the *forecast.dss* file. You can add additional time series records to the WISKI database using the **Ad-Hoc Record** feature in a post group. An Ad-Hoc record allows you to save *forecast.dss* time series record data to the WISKI database even if those records were not listed in the **Available Output** table.

To add a WISKI Ad-Hoc record:

1. From the **Post Editor**, click **Add Post Data Set**, the **Adhoc: KiWIS Time Series Record Chooser dialog** will open.
2. From the **Search By Parts** area, use the various filter lists to locate the desired *forecast.dss* time series record.
3. Select the pathname row for the record, click **Set Pathname**.
4. Once you have selected all the pathnames for all the necessary time series records, from the **File** menu, click **Close**. The **Adhoc: KiWIS Time Series Record Chooser** dialog will close.

The **From forecast.dss** and **To KiWIS** time series records (**Post Linking** table) will display the same pathname. The time series record pathname displayed in the **To KiWIS** column of the **Post Linking** table will need to be changed to match the KiWIS time series name. You can either enter the KiWIS pathname, or select one through the **KiWIS Time Series Record Chooser dialog**.

To select a DSS pathname using the **KiWIS Time Series Record Chooser**:

1. From the **Post Editor**, from the **Post Linking** table, select the **To KiWIS** record you would like to change.
2. Click **Browse To**, the **KiWIS Time Series Record Chooser** dialog will display.
3. From the **KiWIS Time Series Record Chooser** dialog. from the **File** menu, click **Open**. An **Open HEC-DSS File** browser will open, browse to location of the DSS file you will be copying time series record to, select the filename, click **Open**. The **Open HEC-DSS File** browser will close, the DSS file is displayed.
4. Select the pathname for the KiWIS record listed in the table, click **Set Pathname**. This will copy the selected record to the **Post Linking** table (**Post Editor**).
5. When you are finished selecting the KiWIS pathnames, from the **File** menu, click **Close**. The **KiWIS Time Series Record Chooser** dialog will close.

34.8 Running Posts with WISKI

Now that you have defined and setup post groups, you can run posts. HEC-RTS provides two ways to run post - automatic or manually. Running Posts provides more details.

34.9 KiWIS Time Series Record Chooser

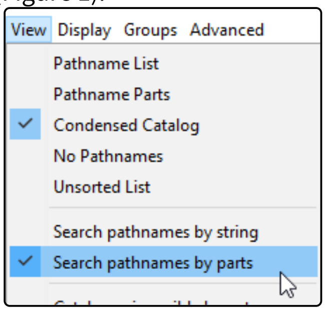
The **KiWIS Time Series Record Chooser** dialog, allows you to browse the WISKI database to view and select data to use for the extract and post groups. The features supported for the KiWIS data are discussed in the following sections and include filtering records and viewing data in plots and tables.

34.9.1 Filtering KiWIS Time Series Records

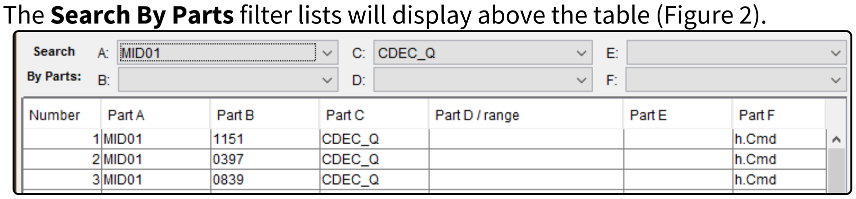
KiWIS time series data can be filtered using pathname parts or by searching for the whole pathname.

To search the KiWIS data using pathname parts:

1. From **KiWIS Time Series Record Chooser** dialog, from the **View** menu, click **Search pathnames by parts** (Figure 1).



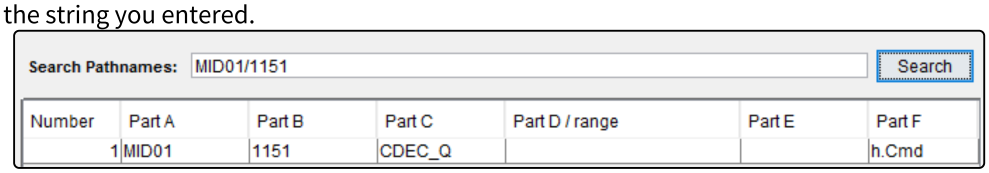
2. The **Search By Parts** filter lists will display above the table (Figure 2).



3. Using the **Search By Parts** filter lists, select the record parts you would like to search on. The table will update with each filter list selection.

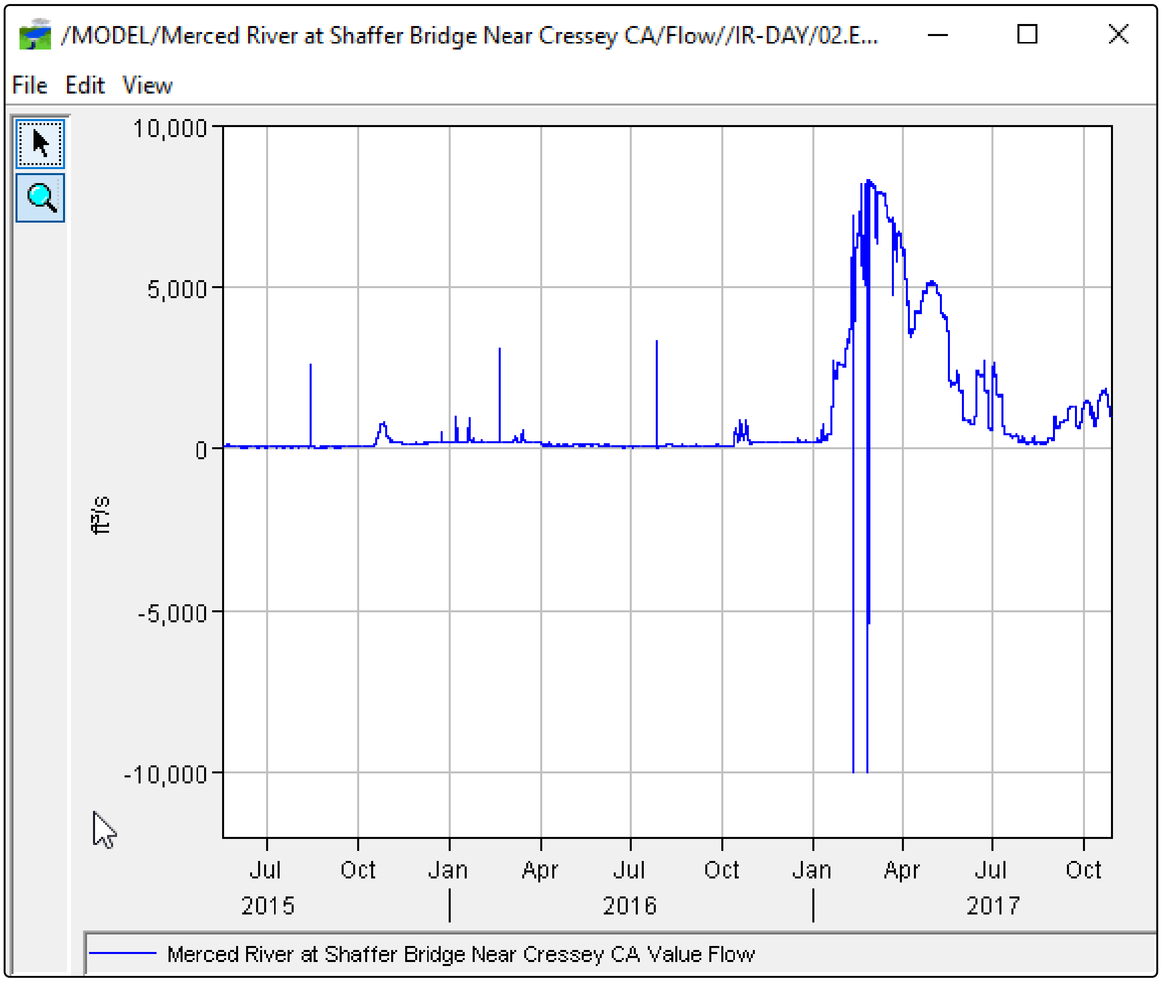
To search the KiWIS data using pathname parts:

1. From **KiWIS Time Series Record Chooser** dialog, from the **View** menu, click **Search pathnames by string** (Figure 1).
2. Enter the KiWIS pathname you are searching for in the **Search Pathnames** box displayed about the table (Figure 3). You do not need to enter the whole pathname. Click the **Search,** the table will filter the pathnames, based on the string you entered.



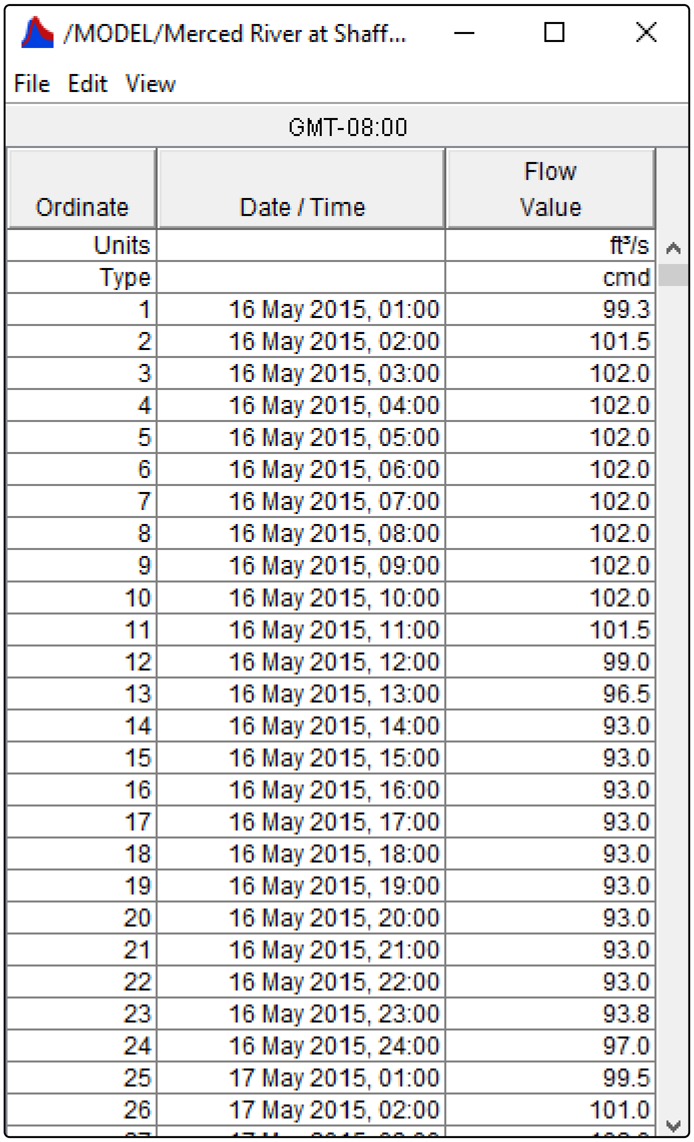
34.9.2 Plotting KiWIS Time Series Records

KiWIS data can be viewed in time series plots from the **KiWIS Time Series Record Chooser** dialog. Select a record, from the toolbar, click  . A **Plot** dialog (Figure 4) will open, displaying the selected KiWIS data in a plot.



34.9.3 Tabulating KiWIS Time Series Records

KiWIS data can be viewed in tabular form from the **KiWIS Time Series Record Chooser** dialog. Select a record, from the toolbar, click . A **Tabulate** dialog (Figure 5) will open, displaying the selected KiWIS data in a tabular form.



Ordinate	Date / Time	Flow Value
Units		ft³/s
Type		cmd
1	16 May 2015, 01:00	99.3
2	16 May 2015, 02:00	101.5
3	16 May 2015, 03:00	102.0
4	16 May 2015, 04:00	102.0
5	16 May 2015, 05:00	102.0
6	16 May 2015, 06:00	102.0
7	16 May 2015, 07:00	102.0
8	16 May 2015, 08:00	102.0
9	16 May 2015, 09:00	102.0
10	16 May 2015, 10:00	102.0
11	16 May 2015, 11:00	101.5
12	16 May 2015, 12:00	99.0
13	16 May 2015, 13:00	96.5
14	16 May 2015, 14:00	93.0
15	16 May 2015, 15:00	93.0
16	16 May 2015, 16:00	93.0
17	16 May 2015, 17:00	93.0
18	16 May 2015, 18:00	93.0
19	16 May 2015, 19:00	93.0
20	16 May 2015, 20:00	93.0
21	16 May 2015, 21:00	93.0
22	16 May 2015, 22:00	93.0
23	16 May 2015, 23:00	93.8
24	16 May 2015, 24:00	97.0
25	17 May 2015, 01:00	99.5
26	17 May 2015, 02:00	101.0

191 Figure 5 Tabulate Dialog - KiWIS Data

35 KiWIS Write Extension

The KiWIS Write Extension enables you to write time a series data into WISKI database. This is applicable for standard time series as well as ensemble/forecast time series. The extension does not enable editing of metadata or creating new time series.

This document was written by the KISTERS Group, Version 1.6.0 (20.04.2017).

35.1 Configuration

On the data source level <Datasource> of the KiWIS *config.xml* the connection information and the service have to be added:

<WAPIConnection>
Enables a WISKI API connection. The API must be active in the WISKI Server and reachable via its network port. Installation and configuration will usually be done by KISTERS.
<apiURL>http://localhost:<REST API Port>/</apiURL> (1)
The target URL.
<apiVersion>0</apiVersion> (1)
Only version 0 supported at the moment.
<user>ModelWrite</user> (1)
The WISKI user should be equal to the WDPConnection user
<password>###</password> (1)

The WISKI password (encrypted) should be equal to the WDPConnection password.
<domain>WISKI</domain> (1)

The WISKI domain should be equal to the WDPConnection domain. </WAPIConnection>

<KiWS/>
Enables the KISTERS WriteServices, currently without further options.

35.2 KISTERS WriteServices Functionality

Analogues to the KiQS, the WriteServices are called with *service=kisters&type= writeservices* in the URL. However, since all commands manipulate or add data, instead of a standard HTTP GET request, a POST or PUT has to be sent. Currently, KiWIS does not make any difference between these two verbs.

1. setTimeseriesValues**:**

This command is intended to insert time series data into existing normal time series. The request requires a single *ts_id* or *ts_path* to identify the target time series. Optionally the *flage truncate=true|false* can be set, default is *false*. If set to true, the sent data will **replace the content of the whole time series**.

The body of the request needs to have JSON content with header '*Content- Type: application/json*', the actual JSON equals the familiar dajson format, just without the metadata. Required is the set of columns and the data array containing the values:

```
[
    {
```

```
    "columns":"Timestamp,Value",
    "data": [
            ["2012-05-28T00:00:00.000+01:00",236],
            ["2012-05-29T00:00:00.000+01:00",234],
            ["2012-05-30T00:00:00.000+01:00",226]
            ]
    }
]
```

The columns 'Timestamp' and 'Value' (case insensitive) will be sufficient to write single column data into time series. The quality code will be set to the default value for new values in WISKI. If a specific code should be set, the column name 'Status' can be used:

```
[
    {
    "columns":"Timestamp,Value",
    "data": [
            ["2012-05-28T00:00:00.000+01:00",236, 120],
            ["2012-05-29T00:00:00.000+01:00",234,120],
            ["2012-05-30T00:00:00.000+01:00",226,120]
            ]
    }
]
```

The given quality code must be defined in WISKI or it will be ignored. Any other value or status columns can be written by adding the WISKI internal column name to the column's entry. This also means that it is **NOT** possible to use any other defined KiWIS return field names like 'Absolute Value' to write data, instead the actual column name from WISKI has to be used. Mappings may be added in the future.

Full example that utilizes the cURL command line tool:

```
curl -X POST -H "Content-Type: application/json" -d "[
 {
 \"columns\":\"Timestamp,Value\",
 \"data\": [
   [\"2012-05-28T00:00:00.000+01:00\",236],
   [\"2012-05-29T00:00:00.000+01:00\",234],
   [\"2012-05-30T00:00:00.000+01:00\",226]
   ]
 }
]"
"http://localhost:8080/KiWIS/KiWIS?datasource=0&service=kisters&t ype=writeServices&req
uest=settimeseriesvalues&ts_id=1587010"
```

35.2.1 **2.** setEnsembleTimeseriesValues**:**

This command is intended to insert time series data into existing ensemble/forecast time series. The request requires a single *ts_id* or *ts_path* to identify the target time series. The body of the request needs to have JSON content with header '*Content-Type: application/json*', the actual JSON equals the familiar dajson format for ensemble time series.

Example body:

```
[
     {
        "ensembleDate": "2016-09-29T07:00:00+01:00",
        "ensembleDispatchInfo": "2016-09-29",
        "quality": 200,
        "columns": "Timestamp,Forecast", "data": [
          [
            "2016-09-29T07:00:00.000+01:00", "0.2"
          ],
          [
            "2016-09-29T07:15:00.000+01:00", "0.1"
          ]
        ]
     }
]
```

The ensembleDate is the timestamp where the ensemble will be stored. The ensembleDispatchInfo is a unique key to identify a single ensemble or model run, both fields are mandatory. The optional quality defines the quality code for the whole ensemble, default is zero (0).

The columns attribute has to list the Timestamp plus all ensemble members that should be written for an ensemble. Unlike with setTimeseriesValues where arbitrary WISKI defined columns may be written as long as the keys are known, the column list here matches the one returned by KiWIS for the same time series.

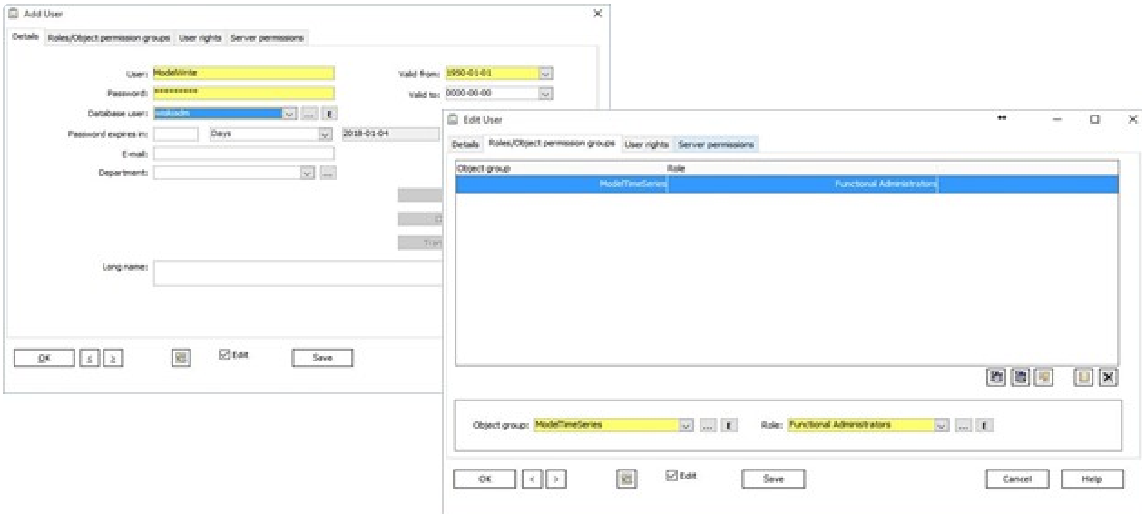
Full example that utilizes the cURL command line tool:

```
curl -X PUT -H "Content-Type: application/json" -d "[
 {
   \"ensembledate\": \"2016-09-29T07:00:00+01:00\",
   \"ensembleDispatchInfo\": \"2016-09-29\",
   \"quality\": 200,
   \"columns\": \"Timestamp,Forecast\",
   \"data\": [
     [
      \"2016-09-29T07:00:00+01:00\",
      \"0.2\"
     ],
   [
     \"2016-09-29T07:15:00+01:00\",
```

```
      \"0.1\"
    ]
    ]
  }
]"
"http://localhost:8080/KiWIS/KiWIS?request=setEnsembleTimeseries Values&ts_id=1057920
10&service=kisters&type=writeServices&datasource=0"
```

35.3 Recommendations

1. The timestamps used in all commands can be sent as any format that KiWIS understands as long as the precision is sufficient.
2. Sending data for an existing time range will overwrite all data in this time range (first and last timestamp in the sent data are relevant).
3. KiWIS currently cannot check whether a time series is editable or if the time series configuration is suitable for writing data. Therefore, it is very important to validate the target ID or path before writing data.
   It is helpful to organize similar writable time series in a time series group or ensure in the client that only a certain path/shortname can be written.
   Write requests to non-writable time series may result in either an exception or a success message even though no data has been written.
   Write requests to editable time series that have agents configured for their own calculation may trigger data loss or unexpected side effects depending on the agent type.
4. For security reasons no externally accessible datasource should have the KiWS service enabled without further protection.
5. Generally, it is recommended to configure a special WISKI user (ModelWrite) who can only access the desired writable time series Object Group and define a second datasource in KiWIS with the KiWS activated and with this user configured.
6. If external users should be able to post data (via Internet), recommendation 5 should be used together with an authentication- protected datasource (see main manual). In that case HTTP Basic Auth can be used to send authenticated requests.


Add User
Details
Roles/Object permission groups
User rights
Server permissions
User:
Password:
Database user:
Password expires in:
E-mail:
Department:
Long name:
Valid from:
Valid to:
OK
Edit
Save
Edit User
Object group
Role
ModelTimeSeries
Functional Administrators
Object group:
Role:
Cancel
Help