US Army Corps
of Engineers
Hydrologic Engineering Center

HEC-RAS
River Analysis System

Supplemental to HEC-RAS Version 5.0 User's Manual

Version 5.0.4
April 2018

Approved for Public Release. Distribution Unlimited.

CPD-68d

REPORT DOCUMENTATION PAGE

Form Approved OMB No. 0704-0188

The public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to the Department of Defense, Executive Services and Communications Directorate (0704-0188). Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number.

PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ORGANIZATION.

1. REPORT DATE *(DD-MM-YYYY)*	2. REPORT TYPE	3. DATES COVERED *(From - To)*
April 2018	Computer Program Documentation	

4. TITLE AND SUBTITLE
HEC-RAS
River Analysis System
Supplemental User's Manual
Version 5.0.4

5a. CONTRACT NUMBER

5b. GRANT NUMBER

5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S)
Gary W. Brunner, 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-68d

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 Hydrologic Engineering Center's (HEC) River Analysis System (HEC-RAS) software allows the user to perform one-dimensional (1D) steady flow, and 1D and two-dimensional (2D) unsteady flow river hydraulics calculations. HEC-RAS is an integrated system of software, designed for interactive use in a multi-tasking, multi-user network environment. The system is comprised of a graphical user interface (GUI), separate hydraulic analysis components, data storage and management capabilities, graphics and reporting facilities.

The HEC-RAS system contains four hydraulic analysis components for: (1) steady flow water surface profile computations; (2) One and two-dimensional unsteady flow simulations; (3) movable boundary sediment transport computations; and (4) water temperature and constituent transport modeling. A key element is that all four components use a common geometric data representation and common geometric and hydraulic computations routines. In addition to the four hydraulic analysis components, the system contains several hydraulic design features that can be invoked once the basic water surface profiles are computed.

The current version of HEC-RAS supports Steady and Unsteady flow water surface profile calculations, sediment transport computations, and water quality analyses. The software also contains tools for performing inundation mapping directly inside the software. New features and additional capabilities will be added in future releases.

15. SUBJECT TERMS
water surface profiles, river hydraulics, steady flow, unsteady flow, software, HEC-RAS, HEC, one-dimensional, hydraulic, analysis, two-dimensional hydraulic analyses, computations, sediment transport, water quality; calculations, integrated system, graphical user interface

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT	b. ABSTRACT	c. THIS PAGE			19b. TELEPHONE NUMBER
	U	U	UU	76	

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

HEC-RAS
River Analysis System

Supplemental User's Manual

April 2018

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

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

CPD-68d

River Analysis System, HEC-RAS, User's Manual

The Hydrologic Engineering Center (HEC) River Analysis System (HEC-RAS) executable code and documentation was developed with U.S. Federal Government resources and is therefore in the public domain. It may be used, copied, distributed, or redistributed freely. However, it is requested that HEC be given appropriate acknowledgment in any subsequent use of this work.

HEC cannot provide technical support for this software to non-USACE users; therefore, non-USACE individuals and organizations should use any internet search engine to locate a vendor that can provide support for the HEC software of interest. However, HEC will respond to all documented instances of program errors. Documented errors are bugs in the software due to programming mistakes not model problems due to user-entered data.

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

Microsoft, *Windows*, and *Excel* are registered trademarks of Microsoft Corp.

Solaris and *Java* are trademarks of Sun Microsystems, Inc.

ArcGIS, *ArcView* and *ArcInfo* are trademarks of ESRI, Inc.

AutoCAD is a trademark of Autodesk, Inc.

Terms and Conditions of Use:

Use of the software described by this document is controlled by certain terms and conditions. The user must acknowledge and agree to be bound by the terms and conditions of usage before the software can be installed or used. The software described by this document can be downloaded for free from our internet site (www.hec.usace.army.mil).

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

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

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

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

The name "HEC-RAS" must not be used to endorse or promote products derived from the Software. Products derived from the Software may not be called "HEC- RAS " nor may any part of the "HEC- RAS " name appear within the name of derived products.

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

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

Waiver of Warranty:

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

Limitation of Liability:

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

THE UNITED STATES GOVERNMENT'S LIABILITY, AND THE LIABILITY OF ITS AGENCIES, OFFICIALS, REPRESENTATIVES, AND EMPLOYEES, INCLUDING ITS CONTRACTORS AND SUPPLIERS, TO YOU OR ANY THIRD PARTIES IN ANY CIRCUMSTANCE IS LIMITED TO THE REPLACEMENT OF CERTIFIED COPIES OF HEC-WAT WITH IDENTIFIED ERRORS CORRECTED. Depending on state law, the above limitation or exclusion may not apply to you.

Indemnity:

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

Assent:

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

Table of Contents

List of Figures

List of Tables

CHAPTER 1

Introduction

1.1 Overview

Welcome to the Hydrologic Engineering Center's (HEC) River Analysis System (HEC-RAS) computer software Version 5.0.4. This software allows users to perform one-dimensional (1D) steady, one and two-dimensional (2D) unsteady flow hydraulics, sediment transport/mobile bed computations, water temperature modeling, and generalized water quality modeling (nutrient fate and transport).

This Version supersedes HEC-RAS Version 5.0.3, and all previous versions. This supplemental user's manual describes all of the new features that were added and modifications made to HEC-RAS Version 5.0.4. The information contained in this document supersedes the information in the HEC-RAS 5.0 User's Manual. However, users should refer to the HEC-RAS Version 5.0 *User's Manual* and HEC-RAS Version 5.0 *2D Modeling User's Manual* for general HEC-RAS software information. For more information on What's New and Bug's Fixed from the last version, see the HEC-RAS 5.0.4 *Release Notes*.

1.2 Installation

The installation program and all documentation for HEC-RAS Version 5.0.4 are available on the HEC website at http://www.hec.usace.army.mil . This new release is installed independently of any previous versions of the software. Users may have the new version and previous versions of HEC-RAS software installed simultaneously for parallel use or testing. HEC-RAS Version 5.0.4 is fully compatible with projects developed in any previous version of the software.

Warning!
Once a project has been opened in HEC-RAS Version 5.0.4 and saved, it may not be possible to open it with an older version of the software and reproduce the old results.

The new installation package for HEC-RAS Version 5.0.4 is designed to be easy to use. The installation process includes steps for selecting a directory for the HEC-RAS Version 5.0.4 software files and other settings (e.g., creating a desktop shortcut, if desired). Use the following steps to install the program on the Microsoft Windows® operating system:

1. Download the HEC-RAS Version 5.0.4 installation package from the HEC website to a temporary folder on the computer.

2. Run the installation program. In Windows Explorer, double-click the icon for the installation program. Note, administrator privileges are required to run the installer.

3. Follow the on-screen prompts to install the program.

1.3 Organization of Supplemental Manual

This supplemental manual is organized as follows:

CHAPTER 2, Hydraulics: This chapter discusses all new and modified HEC-RAS capabilities which relate to hydraulic inputs and computations. Topics discussed include enhancements to the storage area 2D area hydraulic connections between geometric features, a new option to use variable time steps in an unsteady flow analysis, additional capability to include internal boundary condition locations in 2D unsteady flow hydraulics, and a new velocity term that allows velocity to be computed at the boundaries of 2D areas or between 1D and 2D connections.

CHAPTER 3, RAS Mapper: This chapter describes the new editing capabilities and tools available in RAS Mapper. This chapter begins by restating the conditions for developing features in RAS Mapper, which include (i) specifying a coordinate system, (ii) loading a terrain model or other background data, and (iii) creating a new geometry group. Next, detailed instructions for utilizing the new and updated RAS Mapper editing tools are discussed.

CHAPTER 4, Sediment Transport: This chapter describes the new and updated sediment transport modeling capabilities. Topics include the new and modified transport equations, as well as user interface changes.

CHAPTER 5, References: This chapter provides references for this document.

CHAPTER 2

Hydraulics

2.1 Enhancements to SA/2D Hydraulic Connections

The hydraulic structures capabilities for the storage areas/2-dimensional (SA/2D) Area Hydraulic Connection has been upgraded to include all of the same capabilities included in the Inline and Lateral Hydraulic structures. Users can now have spillways, weirs, gates, culverts, rating curves, and time series outlets all in the same structure. Additionally, users can now specify an X and Y geospatial coordinates for the upstream and downstream ends of each hydraulic outlet (culverts, gates, rating curves, and time series outlets). The information contained in this section is supplemental to Chapter 6 of the HEC-RAS 5.0 *User's Manual*.

The example SA/2D Area Connection displayed in Figure 2-1, can be used to model a levee with two culverts going through the levee. Note that the culverts cross over multiple cells on the inside of the levee. This is accomplished by the fact that users can now enter X and Y coordinates for the centerlines of the culverts, as well as gates, rating curves, and flow time series outlets.

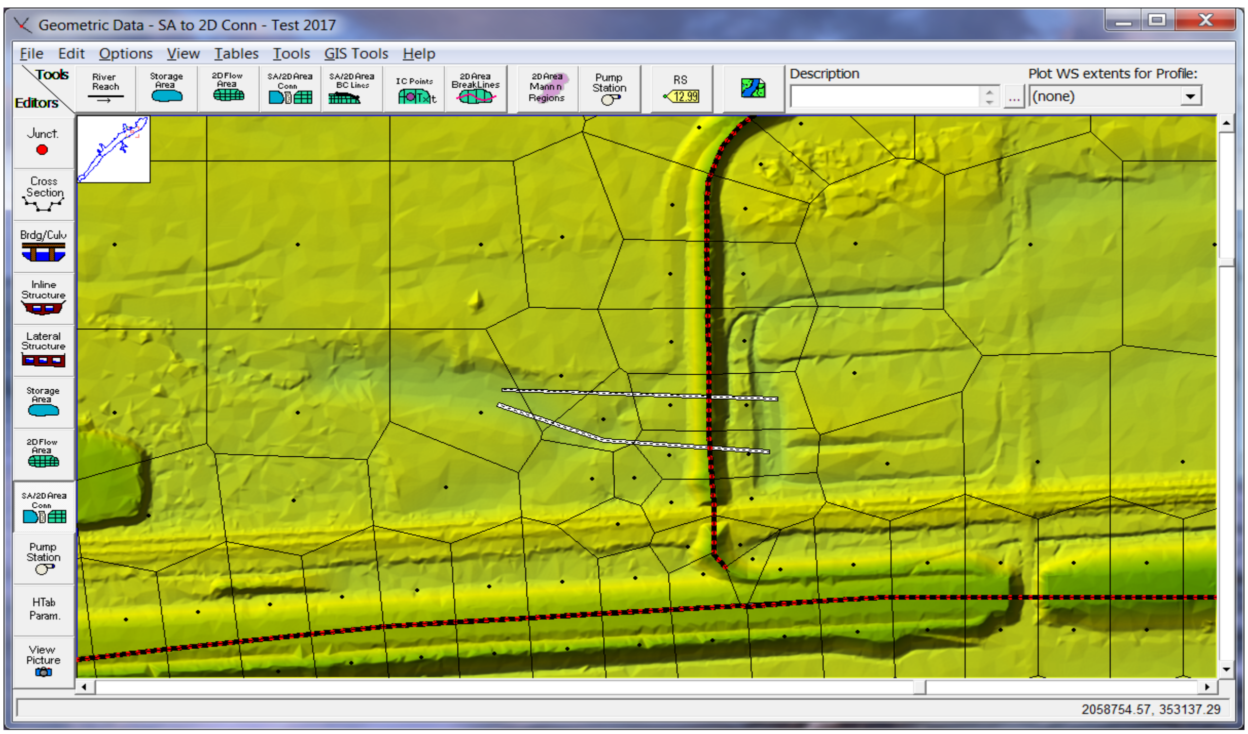


Figure 2-1. SA/2D Area Connection for Modeling a Levee (*drawn left to right looking downstream*) with Culverts (*drawn upstream to downstream*).

From the **Geometric Data** editor, click the **SA/2D Area Conn** 

button (Figure 2-1), and the updated SA/2D Area **Connection Data Editor** will open, as shown in Figure 2-2.

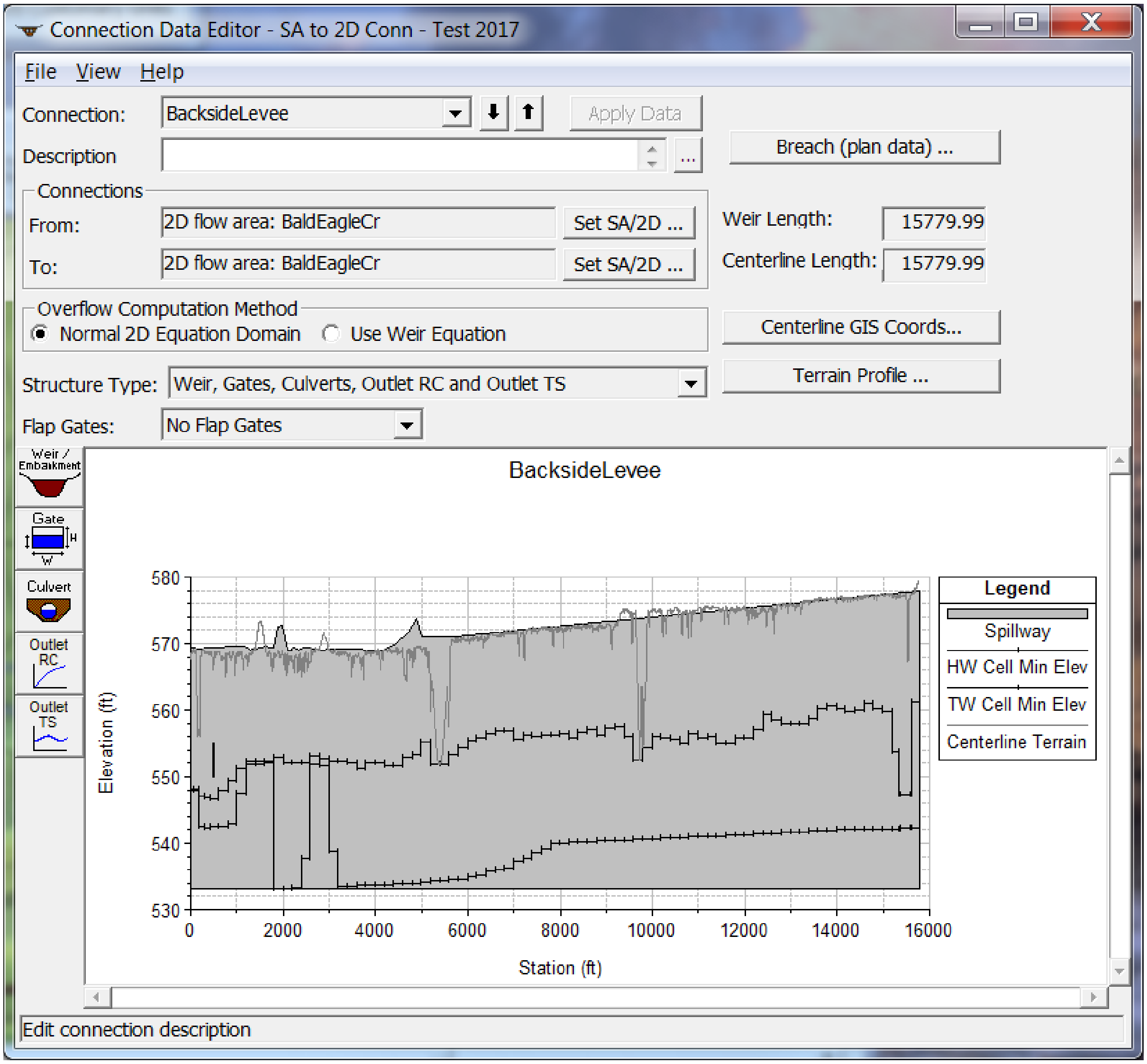


Figure 2-2. SA/2D Area Connection editor with Levee and Culverts

As shown in Figure 2-2, users can define several hydraulic opening types (spillways/weirs, gates, culverts, rating curves, and time series outlets) utilizing the buttons on the left side of the window and all within the same structure. The water going over the structure can be modeled with a weir equation or with the solution of the normal 2D Flow equations (Full St. Venant or Diffusion wave). Culverts can have flap gates on either side, or no flap gates. In the example shown in Figure 2-1 and Figure 2-2, there is a culvert group with two culvert barrels. If the user selects the **Culvert** 

button on the left, the **Culvert Data Editor** will open (Figure 2-3).

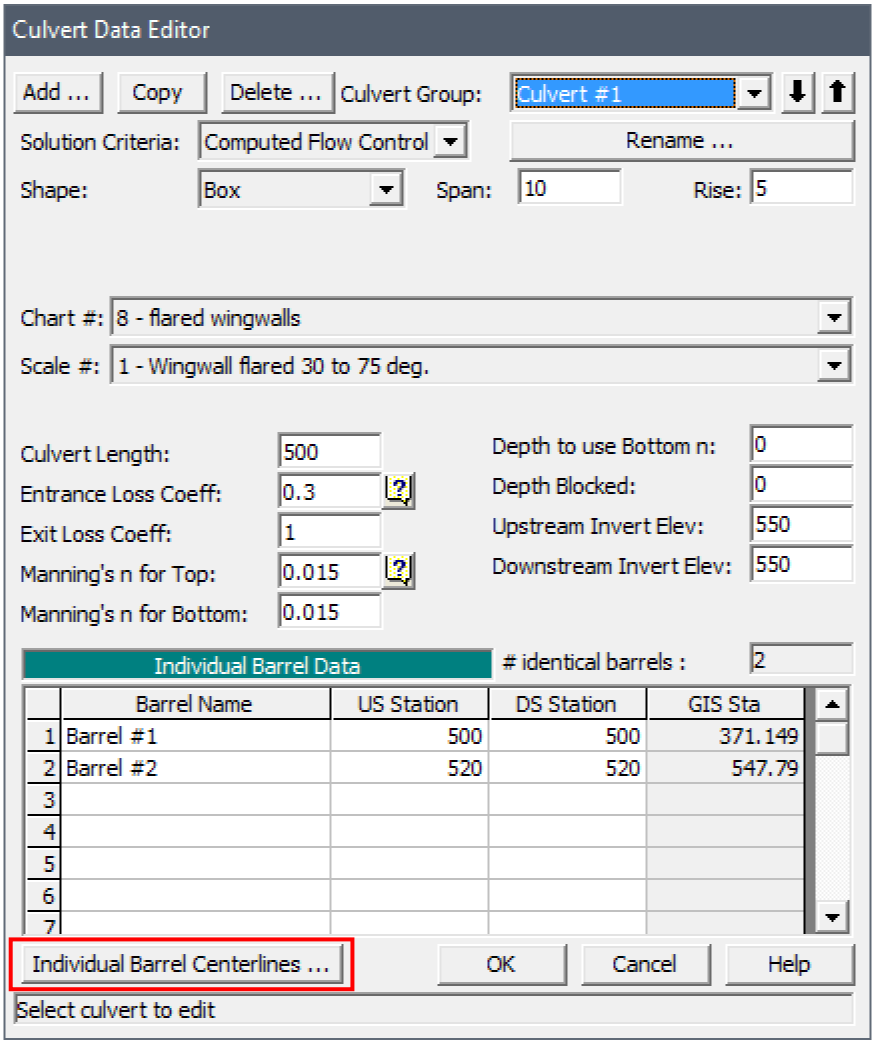
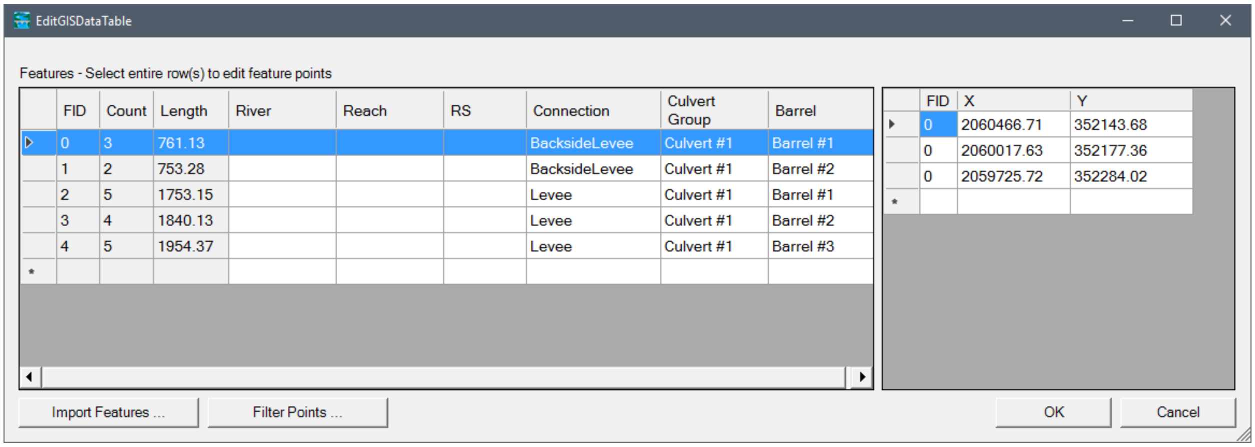


Figure 2-3. Culvert Data Editor for the SA/2D Area Connection.

The **Culvert Data Editor** is the same as in HEC-RAS Version 5.0.3, except in HEC-RAS Version 5.0.4 there is a new button on the lower left portion of the editor labeled "**Individual Barrel Centerlines**" (r*ed box* in Figure 2-3). This new button opens the **Edit GIS Data Table** editor (Figure 2-4), for entering the X and Y coordinates for the centerline of each culvert barrel added to the model, which allow users to view the barrels spatially. **[Note:** Separate centerlines must be added even for identical barrels within the same culvert group, and the barrels may also be connected to different cells.**]**

All Culvert centerlines (as well as gates, rating curves, and flow time series outlets), must be drawn from upstream to downstream. Keep in mind that is how the original centerline of the SA/2D Area Connection is drawn which defines upstream and downstream. Therefore, when users draw the centerline for the SA/2D Area Connection, it is drawn from left to right looking in the downstream direction. Based on that convention, when the centerlines for the hydraulic outlets (culverts, gates, rating curves, etc.) are drawn, yet again the centerlines must be drawn from the upstream side of the structure to the downstream side of the structure. For the example provided in Figure 2-2, the structure being used to model the levee was drawn from the south end of the levee to the north end of the levee. Therefore, the culverts were drawn from the right hand side of the structure (head water) to the left hand side of the structure (tailwater).

If the user presses the button labeled "**Individual Barrel Centerlines**" from the **Culvert Data Editor**, the **EditGISDataTable** centerline coordinate editor opens (Figure 2-4).



FID	Count	Length	River	Reach	RS	Connection	Culvert Group	Barrel
0	3	761.13				BacksideLevee	Culvert #1	Barrel #1
1	2	753.28				BacksideLevee	Culvert #1	Barrel #2
2	5	1753.15				Levee	Culvert #1	Barrel #1
3	4	1840.13				Levee	Culvert #1	Barrel #2
4	5	1954.37				Levee	Culvert #1	Barrel #3

FID	X	Y
0	2060466.71	352143.68
0	2060017.63	352177.36
0	2059725.72	352284.02

Figure 2-4. Hydraulic Structure Centerline Data Table

As shown in Figure 2-4, The Feature Centerline Table contains the centerline X and Y coordinate data for all of the structures in the model. When the **Edit GIS Data Table** editor is opened, it will highlight the culvert that was open and selected in the **Culvert Data Editor**. To use this editor, from the Features table (located on the left of the editor), select a single culvert barrel (highlight it), and then past in the X and Y coordinates for the barrel into the data table (on the right hand side of the editor). **Hint:** The easiest way to define the culvert barrel X and Y centerline coordinates is to go back to the **Geometric Data** editor, hold down the **Ctrl** key, and digitize the culvert barrel centerline from the headwater side of the structure to the tailwater side of the structure. This digitized line can be copied to the clipboard from the **Measure Line** editor that pops up once digitizing the line is complete. Once all of the barrel coordinates have been entered, close all of the SA/2D Area Connection windows, and the digitized culvert(s) will appear in the **Geometric Data** editor window, similar to what is shown in Figure 2-1. Additionally, there is an option to import the culvert X and Y centerline coordinates at the bottom of the editor.

Note: In HEC-RAS Version 5.0.4, the process for adding centerline X and Y coordinates for individual hydraulic outlets (gates, rating curves and flow time series, etc.) is exactly the same as described above for culverts.

2.2 Variable Time Step Capabilities

Some new variable time step capabilities have been added to the unsteady flow engine for both 1-dimensional (1D) and 2D unsteady flow modeling. Two new options are available. One is a variable time step based on monitoring Courant numbers (or residence time within a cell), while the other method allows users to define a table of dates and time step divisors. The variable time step option can be used to improve model stability, as well as reduce computational time (not all models will be faster with the use of the variable time step). The information contained in this section is supplemental to Chapter 8 of the HEC-RAS 5.0 *User's Manual*.

These new Variable Time Step options are available from the **Unsteady Flow Analysis** window and also by going to the **Computational Options and Tolerances** window. There is a new button right next to the **Computation Interval** directly on the **Unsteady Flow Analysis** window, which will take users to this new feature (shown in Figure 2-5 below).

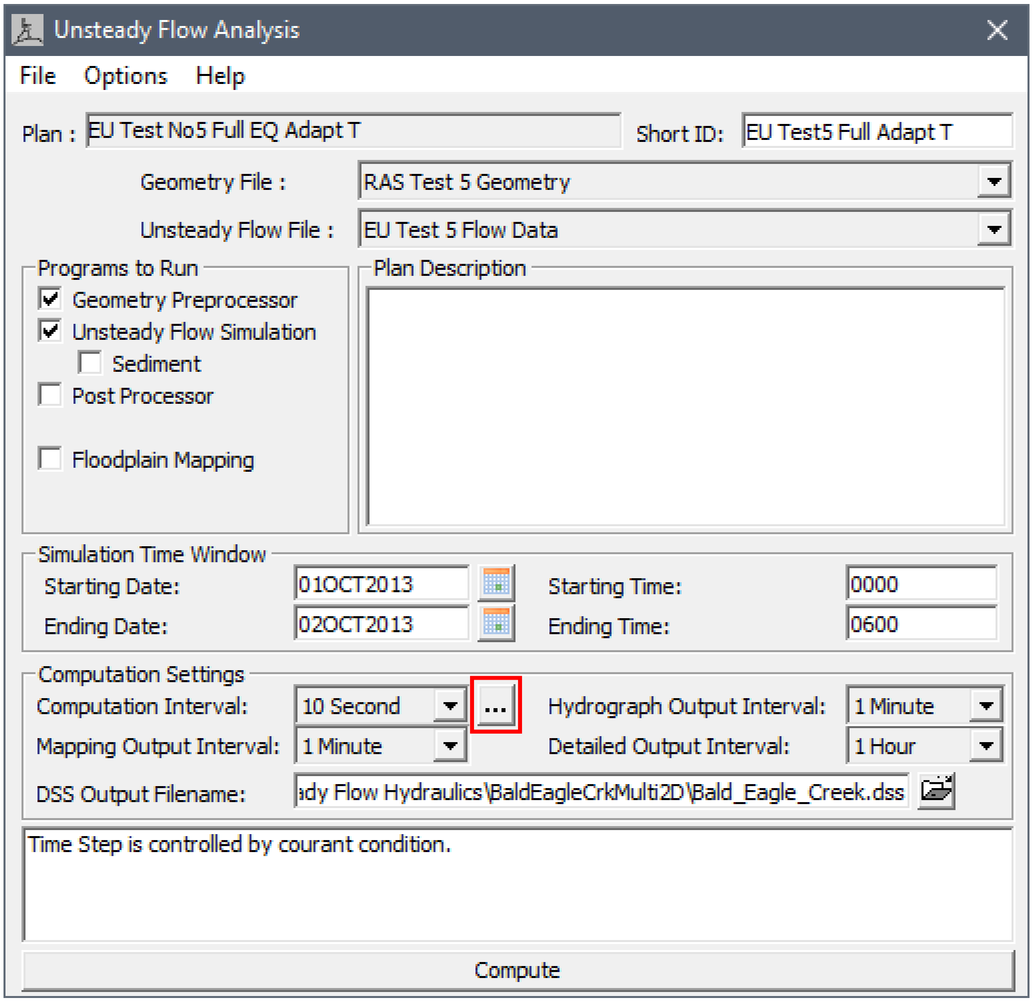


Figure 2-5. Button for Variable Time Step Option Shown on Unsteady Flow Analysis Window.

When the Variable Time Step control editor ellipse button is clicked, the modified **HEC-RAS Unsteady Computation Options and Tolerances** window opens to the new **Advanced Time Step Control** tab (Figure 2-6). Alternatively, users can open the window and navigate to the new tab, from the **Options | Computation Options and Tolerances** menu option.

As shown in Figure 2-6, the new **Advanced Time Step Control** tab now has three different methods for selecting and controlling the computational time step: (i) **Fixed Time Step** (default); (ii) **Adjust Time Step Based on Courant** which is a variable time step based on the Courant number; and (iii) **Adjust Time Step Based on Time Series of Divisors** which is a variable time step based on a user entered table of dates, times, and time step divisor's. The two new variable time step options (Courant number and Time Step Divisor) are discussed in the following sections.

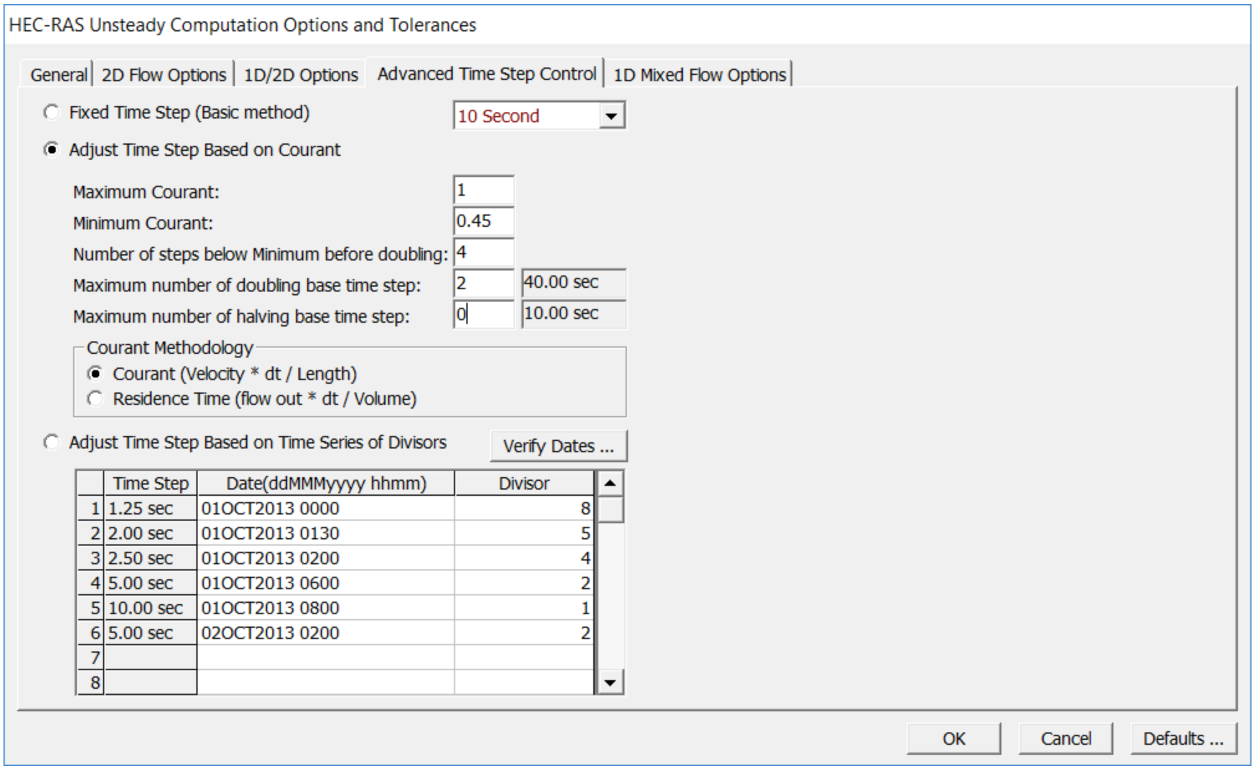


Figure 2-6. Variable Time Step Editor within the Computational Options and Tolerances.

2.2.1 Courant Number Method

The first new variable time step option is to use the Courant number method, from the new **Advanced Time Step Control** tab. In the example shown in Figure 2-5 and Figure 2-6, the Courant number is being used for the variable time step. To use this method, select the **Adjust Time Step Based on Courant** option and provide the following information:

- **Maximum Courant:** This is the maximum Courant number allowed at any 2D cell or 1D cross section. If the maximum Courant number is exceeded, then the time step is cut in half for the very next time interval. Because HEC-RAS uses an implicit solution scheme, Courant numbers can be greater than one, and still maintain a stable and accurate solution. In general, if the flood wave is rising and falling slowly (depth and velocity are changing slowly), the model can handle extremely high Courant numbers. For these types of cases, users may be able to enter a Maximum Courant number as high as 5.0 or more. However, if the flood wave is very rapidly changing (depth and velocity are changing very quickly over time), then the Maximum Courant number will need to be set closer to 1.0. The example shown in Figure 2-6 is for a Dam break type of flood wave, in which depth and velocity are changing extremely rapidly. Because of the rapid changes in depth and velocity, the Maximum Courant number was set to 1.0.

- **Minimum Courant:** This is the minimum Courant number threshold for 2D cells and 1D cross sections. If the Courant number at "all" locations goes below the minimum, then the time step will be doubled. However, the time step will only be doubled if the current time step has been used for enough time steps in a row to satisfy the user entered criteria called "Number of steps below Minimum before doubling" (see below for an explanation of this field). The "Minimum Courant" value should always be less than half of the "Maximum Courant" value. If the Minimum Courant value is equal to or larger than half the Maximum Courant value, the HEC-RAS Version 5.0.4 software will just flip back and forth between halving and doubling the time steps. In the example shown in Figure 2-6, since the Maximum Courant number was set to 1.0, the minimum was set to 0.45 (less than half of the maximum), which allowed the model to stay stable, but also run faster.

- **Number of steps below Minimum before doubling:** This field is used to enter the integer number of time steps in which the Courant number must be below the user specified minimum before the time step can be increased. This can prevent the model from increasing the time step too quickly and/or from flipping back and forth between time steps. Typical values for this field may be in the range of 5 to 10.

- **Maximum number of doubling base time step:** This field is used to enter the maximum number of times the base time step can be doubled. For example, if the base computation interval is 10 seconds, and the user wants to allow it to go up to 40 seconds, then the value for this field would be 2 (i.e., the time step can be doubled twice: 10s to 20s to 40s). The value displayed in the box to the right of the user entered value is what the entered maximum time step will end up being.

  NOTE: The HEC-RAS Version 5.0.4 software requires that all time steps end up exactly hitting the **Mapping Output Interval**. This requirement is because output for HEC-RAS Mapper must be written to the output file for all cross sections, storage areas, and 2D cells at the mapping interval. Because of this fact, if users enter a "Maximum number of doubling base time step" that results in a computation interval that does not exactly land on the mapping output interval, then the unsteady flow computational program will compute its own time steps that work with the parameters entered in the Adjust Time Step Based on Courant section. Furthermore, the base time step will be changed to something close to what was entered, but when doubling it, all values will still line up with the mapping output interval. When the model runs it will list what time step it is currently using in the message window of the computational output window.

- **Maximum Number of halving base time step:** This field is used to enter the maximum number of time that the base computation interval can be cut in half. For example, if the base computation interval is 10 seconds, and the user wants to allow it to go down to 2.5 seconds, then the maximum number of halving value would need to be set to 2 (i.e., the time step can be cut in half twice: 10s to 5s to 2.5s). The value displayed in the box to the right of the user entered value is what the entered maximum time step will end up being.

For the Courant number method, the default approach for computing the Courant number is to take the velocity times the time step divided by the length (between 1D cross sections, or between two 2D cells). For 2D, the velocity is taken from each face and the length is the distance between the two cell centers across that face. For 1D, the velocity is taken as the average velocity from the main channel at the cross section, and the length is the distance between that cross section and the next cross section downstream.

An optional approach to using a traditional Courant number method is to use **Residence Time**. With this method, the HEC-RAS Version 5.0.4 software is computing how much flow is leaving a 2D cell over the time step, divided by the volume in the cell. The Residence Time method is only applied to 2D cells. When this method is turned on, it is used for the 2D cells, but 1D cross sections still use the traditional Courant number approach.

2.2.2 User Defined Dates/Time vs Time Step Divisor

Another option available from the **Advanced Time Step Control** tab, is to set the variable time step control based on a user defined table of dates and times verses a time step divisor (Figure 2-7).

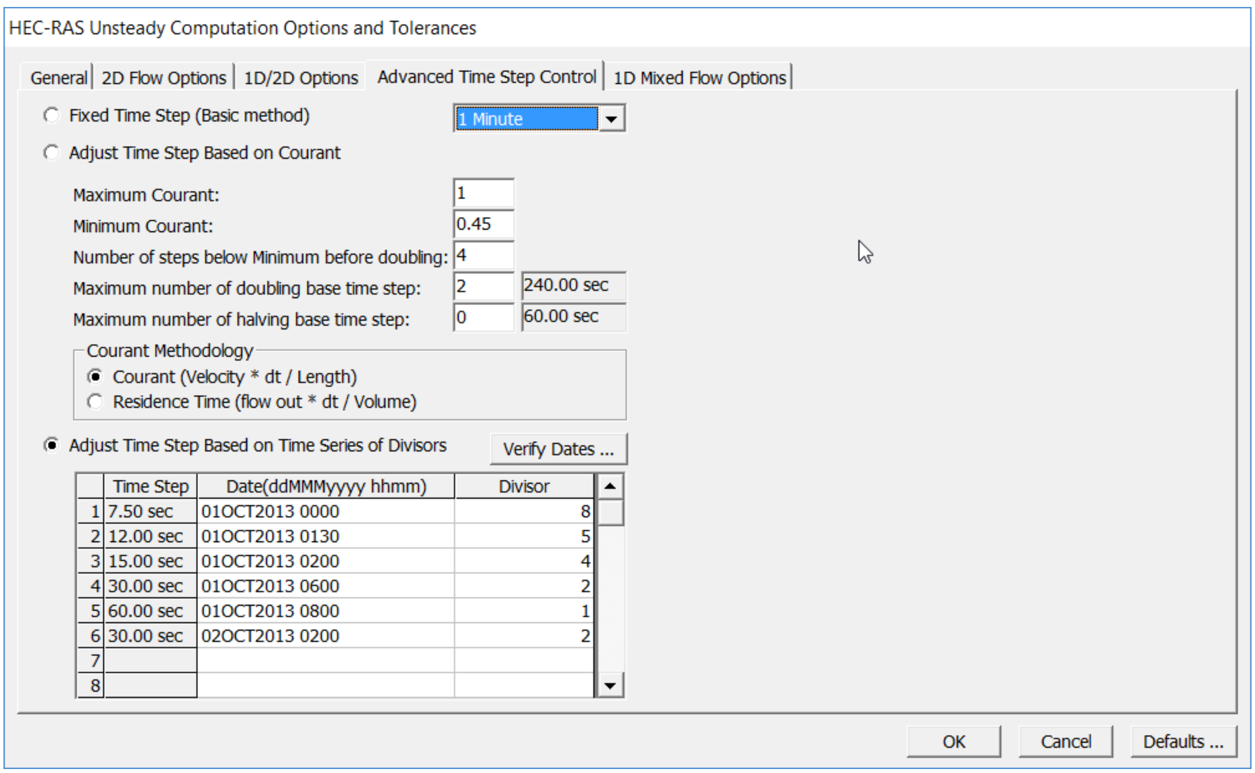


Figure 2-7. User Defined Variable Time Step Table.

As shown in Figure 2-7 the user can select the option called "**Adjust Time Step Based on Time Series of Divisors**", from the **Advanced Time Step Control** tab. When this option is selected the user must enter a table of **Dates** and times verses time step **Divisors**. The first date/time in the table must be equal to the starting date/time of the simulation period. To use this method, enter a base time step equal to the maximum time step desired during the run. Then in the table, under the **Divisor** column, enter the integer number to divide that time step by for the current date/time in the table. Once a time step is set for a date/time, the Unsteady Flow Analysis compute will use that time step until the user sets a new one.

In the example shown in Figure 2-7, the base computational interval was set at 1 minute. Based on the table of dates/times and Divisors entered, the actual time steps that will be used are displayed in the first column labelled "Time Step".

The Time Step Divisor method for controlling the time step requires much more knowledge by the user about the events being modelled, the system being routed through, as well as knowledge of velocities, cross section spacing, and 2D cell sizes. However, if done correctly, this method can be a very powerful tool for decreasing model run times and improving accuracy.

2.3 Internal Boundary Condition Lines for 2D Areas

In the **Geometric Data** editor, a new ability has been added to the **SA/2D Area BC Lines** 

tool when entering and editing geometric data (Figure 2-8). This new ability allows users to create internal boundary condition location lines inside 2D areas, and then attach flow hydrographs to these lines. Internal Boundary Condition (BC) lines are added to the geometry the same way as external BC lines (as described in Chapter 6 of the HEC-RAS 5.0 *User's Manual*).

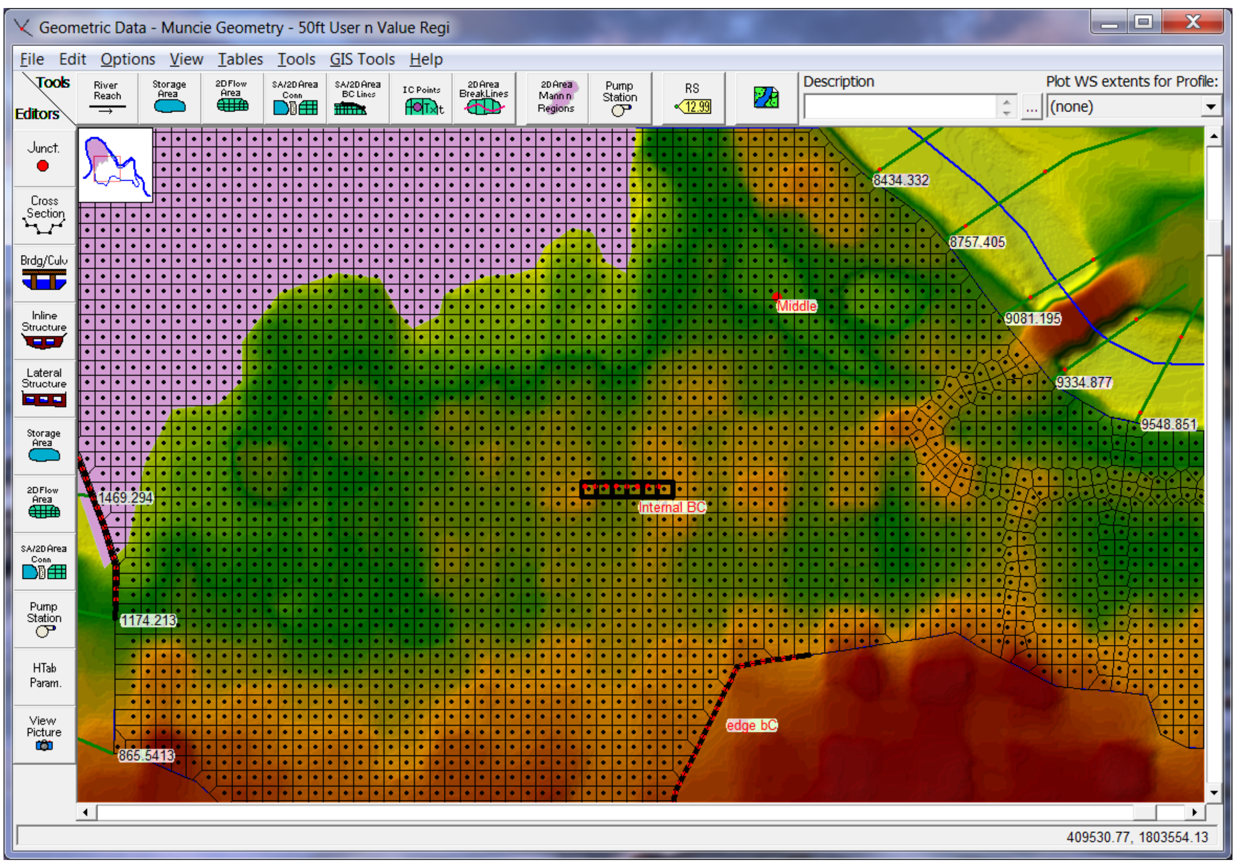


Figure 2-8. Example Internal BC Line for Attaching a Flow Hydrograph inside a 2D Area.

To add the internal boundary condition, from the **Tools** button bar click the **SA/2D Area BC Lines** tool (Figure 2-8). The internal boundary condition line can encompass one or more 2D cells. Once the line is drawn, a name for the BC line must be entered. Figure 2-8 provides an example of a created internal BC line named “*Internal BC*”, as well as an example of the original tool’s ability to add only external BC line (labeled “*edge bC*”).

After an internal BC line is drawn in the **Geometric Data** editor, the user can go to the **Unsteady Flow Data** editor (opened from the HEC-RAS main window by clicking the button) to attach flow hydrographs to the internal BC lines. Once the editor is open, select the **Boundary Conditions** tab, and view in the “Storage/2D Flow Areas” table that there is a row for the newly created internal boundary condition line(s) (e.g., *2D Interior Area BCLine: Internal BC* in Figure 2-9) that were laid out in the **Geometric Data** editor.

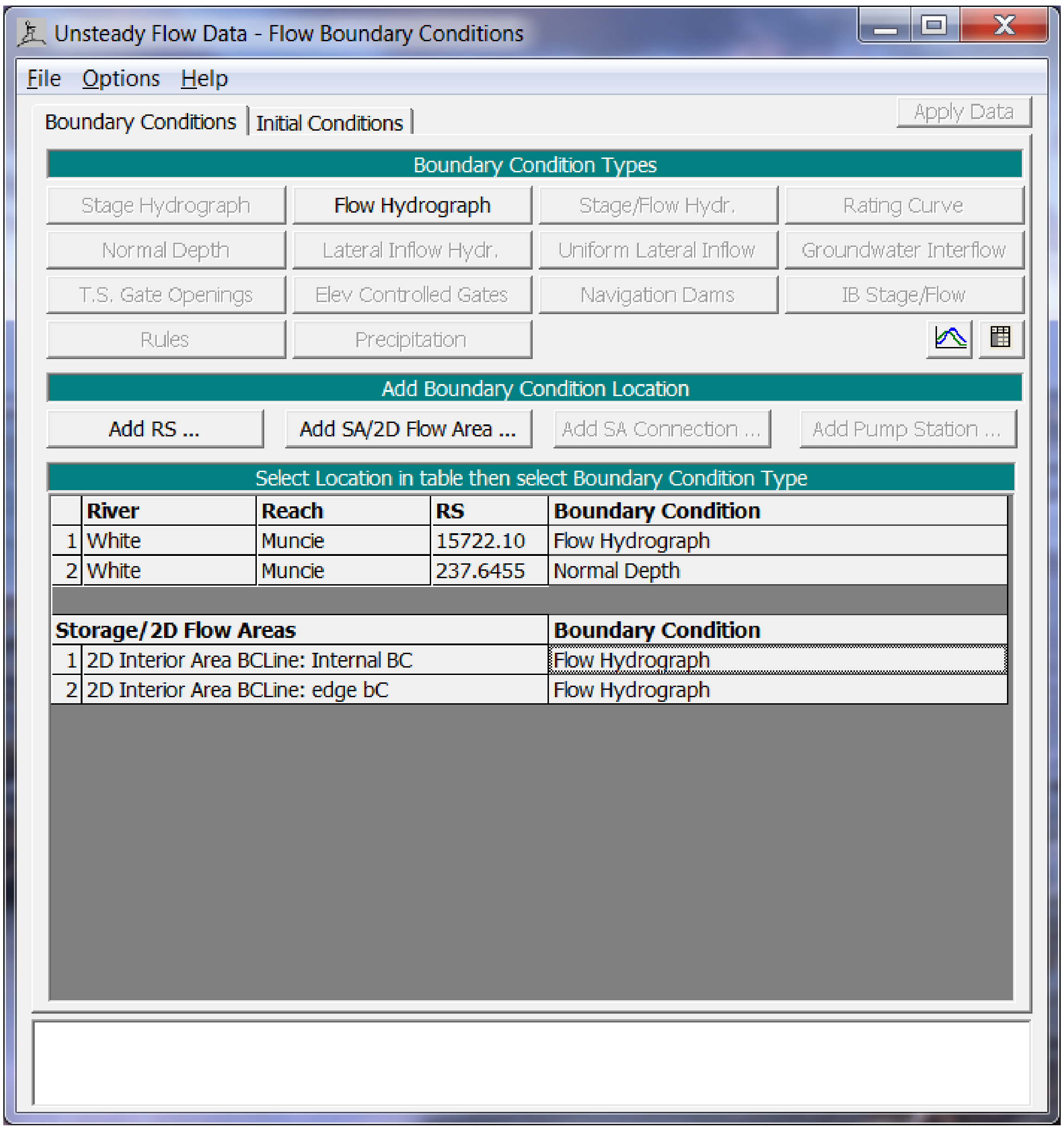


Figure 2-9. Unsteady Flow Data Editor with Example Internal BC Line.

As shown in Figure 2-9, the internal BC lines will show up in the boundary conditions table, and will allow the user to attach a flow hydrograph to that BC line. If the BC line crosses more than one cell, flow is distributed across the cells based on the percentage of the line length that crosses that cell. For example, if a cell contains 20% of the line length, it will receive 20% of the flow each time step. Chapter 8 in the HEC-RAS 5.0 *User's Manual* contains more information regarding unsteady flow analysis.

2.4 New Velocity Term for 2D Flow Area Boundary Conditions and 1D/2D Connections

Flow hydrograph boundary conditions (BC) and 1D to 2D direct connections (1D reach going into or directly out of a 2D area) now have a velocity computed at the boundary. In previous versions of HEC-RAS, flow was just dropped into the cells that the BC line was connected too, without adding velocity. Now in HEC-RAS Version 5.0.4, velocity is also computed at the boundary in addition to the flow for each cell. For the flow boundary conditions, the velocity is based on the user entered energy slope and using Manning's equation to compute the water surface, flow, and velocity across each cell.

Users can enter the user-defined energy slope value in the "EG Slope for distributing flow along BC Line" box (*red box* in Figure 2-10) from the **Flow Hydrograph** editor (opened from the **Unsteady Flow Data** editor's "Boundary Condition Types" by clicking the **Flow Hydrograph** button as displayed in Figure 2-9).

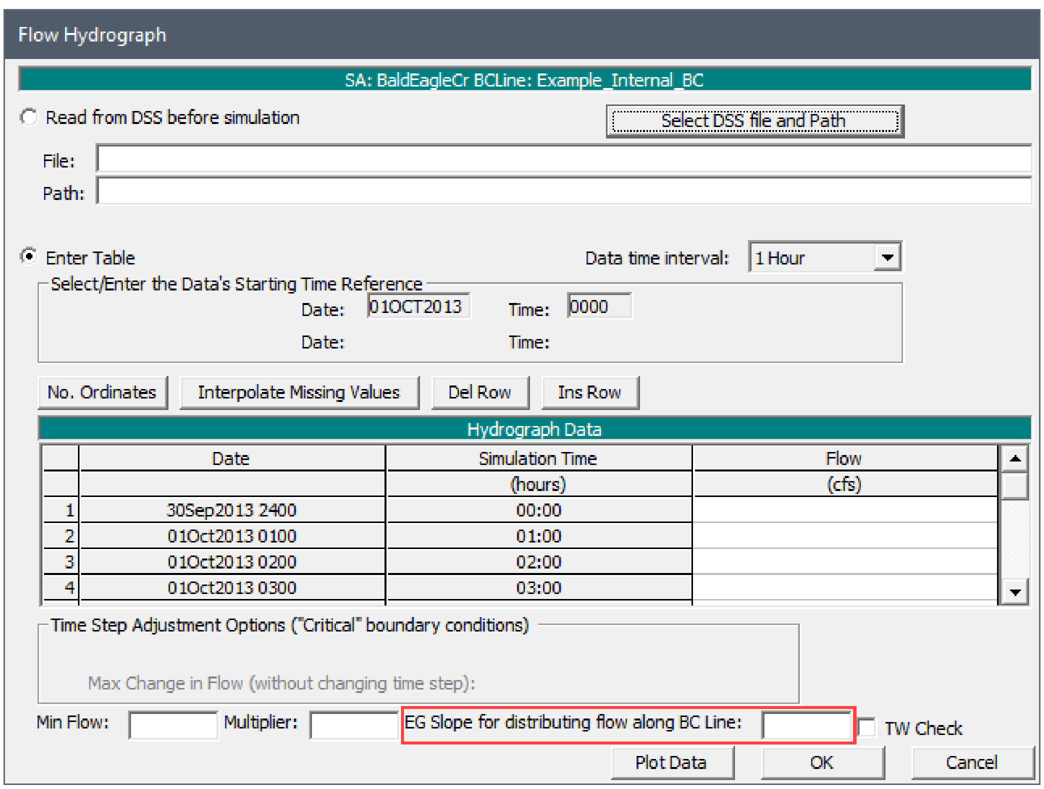


Figure 2-10. User-Defined Energy Slope (*red box*) for Distributing Flow.

Also notice in Figure 2-10 that there is a new Tail Water (TW) check. If this option is turned on, the software checks to see if the water surface inside of the 2D area is higher than what is being computed from Manning's equation and the user entered energy slope. If the tail water in the 2D area is higher, then that water surface is used to compute the flow distribution and velocities at the boundary condition.

For the direct 1D to 2D connections, the velocity is based on the computed water surface elevation at the connections.

Additionally, it is now optional to incorporate velocity into lateral structures that are connected to 2D flow areas (the default is off, and only a flow transfer is used). If the "2D Boundary" **Use Velocity** option at a lateral structure is turned on, from the **Lateral Structure Editor** (Figure 2-11), then the velocity of the flow going over the structure and through any breach is computed and used as a boundary condition, along with the flow, at the 2D cells connected to the structure. Users can access the **Lateral Structure Editor** from the **Geometric Data** editor, click the **Lateral Structures** 

button (Figure 2-8), and the updated editor will open (Figure 2-11).

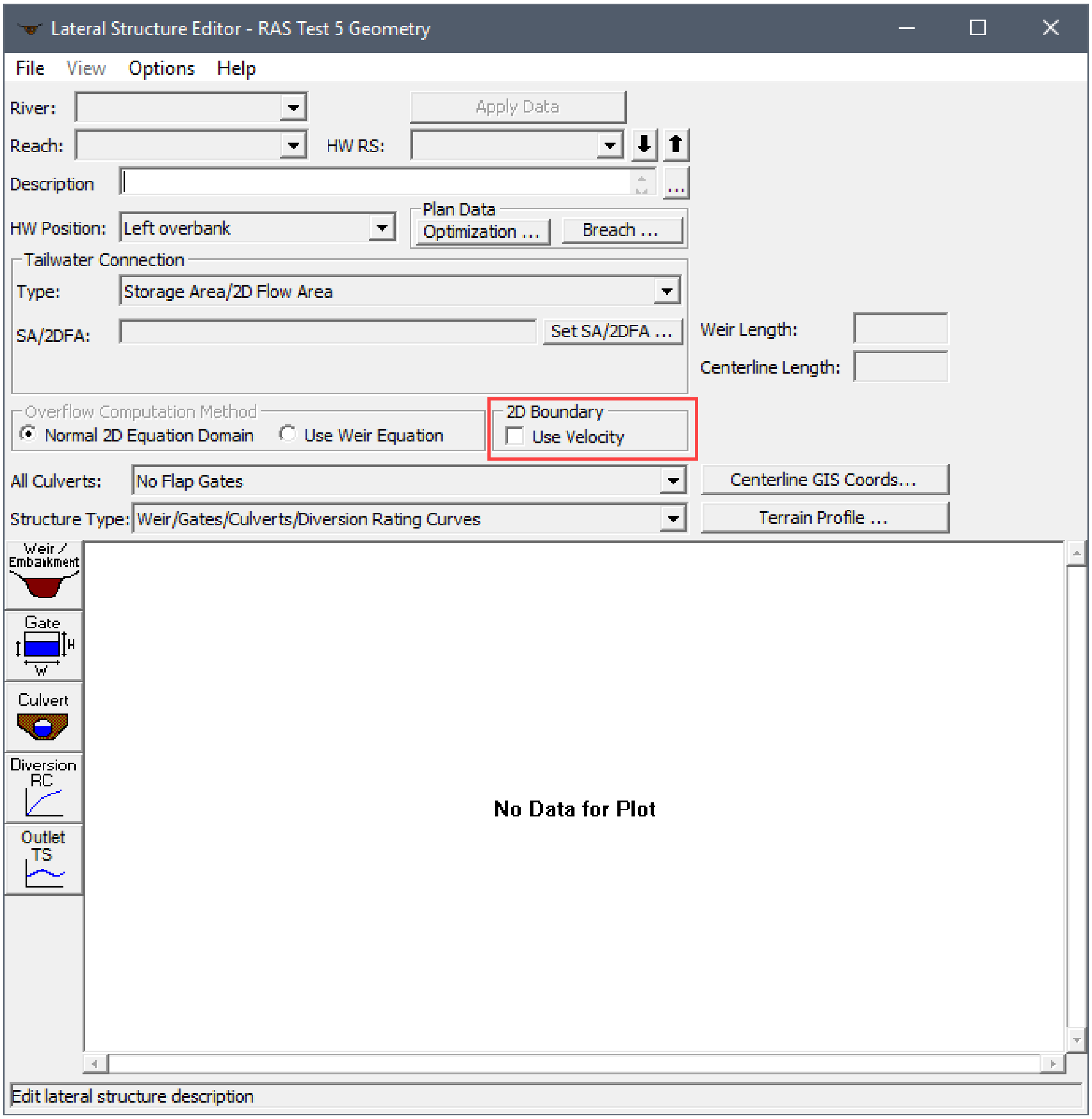


Figure 2-11. Lateral Structure Editor Option (*red box*) to Incorporate Velocity into Lateral Structures Connected to 2D Flow Areas.

CHAPTER 3

RAS Mapper

The release of HEC-RAS 5.0.4 introduces new Editing Tools in RAS Mapper. The information contained in this section is supplemental to Chapter 20 of the HEC-RAS 5.0 *User's Manual*. The editing capabilities will allow the HEC-RAS user to geospatially create 1D and 2D geometry directly in HEC-RAS, based on an underlying Terrain dataset. While these tools are considered "Beta" at this time (they have had "limited" testing and do not have the full functionality we would hope to have), the tools are fairly comprehensive and will greatly assist the user in creating the necessary features such as river centerlines and cross sections for a 1D model and 2D area boundaries, breaklines, and refinement regions for 2D modeling.

Every attempt was made to have the editing tools environment be as simple and straight forward as possible. However, in order to make the editing tools flexible enough to handle the complexities of editing, modifying, and processing of geospatial data required more than a few buttons and menus were added. The editing tools have been set up so that the user can right-click on a selected feature to access editors and perform edits, as well as right-click on the layer itself to access other options. In general, right-clicking on a feature works on the selected set, where right-clicking on the layer works on all features.

The integration of editing tools directly in RAS Mapper should make developing HEC-RAS models more efficient and encourage development of a more refined hydraulics model than previously done for the same effort. Users will be able to spend more time refining geometry to give accurate model results and spend less time struggling with data development.

3.1 RAS Mapper Getting Started

The information contained in this section provides instructions for creating the basic conditions for developing features in RAS Mapper, which include (i) specifying a coordinate system, (ii) loading a terrain model or other background data, and (iii) creating a new geometry group.

3.1.1 Terrain Data

Before developing features in RAS Mapper, a terrain model or other background data must be loaded. A good terrain model is the backbone of developing good geometric data. The user must also specify a coordinate system to work with. To specify a coordinate system, select the **Tools | Set Projection for Project** menu item. The RAS Mapper Project Settings will be shown, allowing the user to specify a projection. In the Options dialog provided, use the file browser to navigate to an ESRI projection file (***.prj***). An example of the projection dialog is shown in Figure 3-1. As also shown in Figure 3-1, the Options dialog allow the user to choose how the coordinate system and Terrain data will affect RAS Mapper computations. Settings for the number of Decimal Places for horizontal (distances) and vertical (elevation) data extraction, computation of the XS River Stationing, and the number of Elevation Points for cross sections and lateral structures are available.

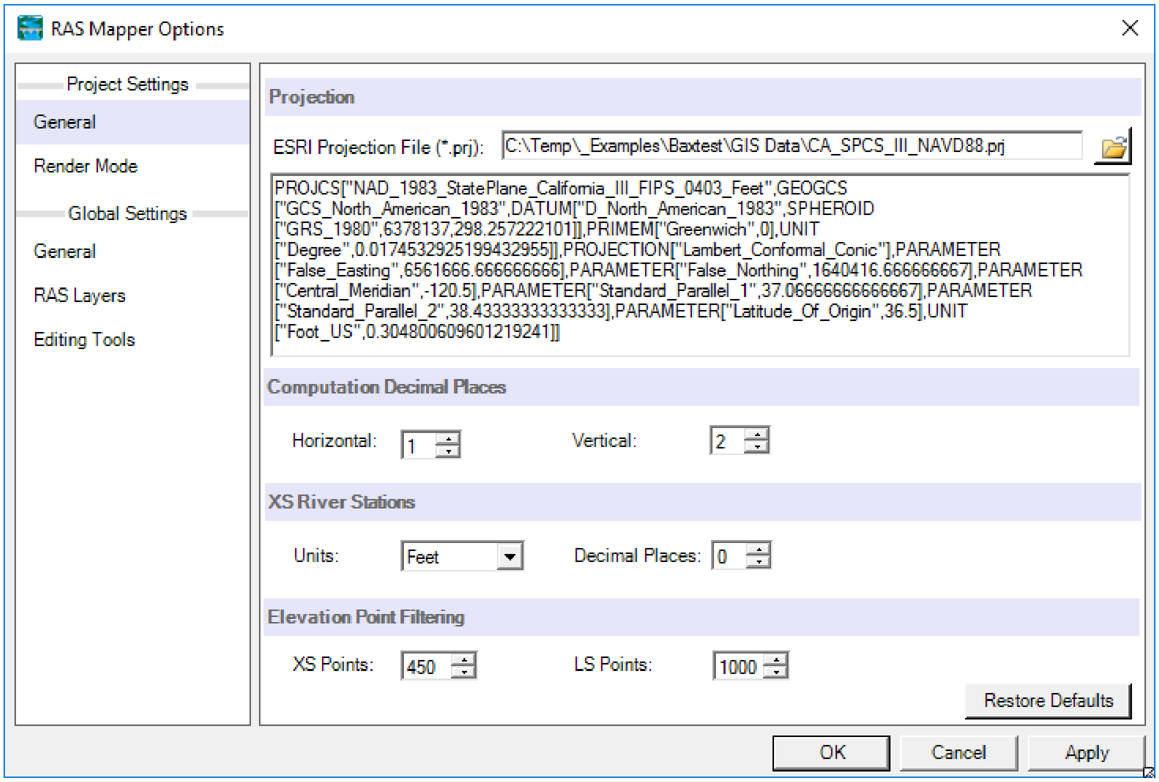


Figure 3-1. Setting the projection and other parameters for the RAS Mapper project.

Once a projection has been defined, add the terrain data by selecting the **Tools | New Terrain** menu item. The **New Terrain Layer** dialog shown in Figure 3-2 will allow the user to select multiple terrain models and specify the order of priority when using the data for extracting elevations and computing results maps. By default, the new terrain model will be saved as "***Terrain.hdf***". After selecting the terrain import parameters and specifying the terrain Filename (e.g., *MuncieWithChannel.tif* in Figure 3-2), click the **Create** button to import the terrain model.

The **New Terrain Layer** dialog has two additional processing options: "Create Stitches" and "Merge Inputs to Single Raster". The **Create Stitches** option is turned on by default and will triangulate the terrain data to fill in any holes in the imported terrain model and interpolate across gaps between neighboring terrain tiles. The **Create Stitches** option can be turned off if using only one terrain tile that has no holes in the data or you are importing multiple tiles that are co-registered and edge matched (same cell size, in the same coordinate system, that line up perfectly). The **Merge Inputs to Single Raster** allows the user to import multiple terrain tiles into a single merged terrain model (and single file on disk). The terrain merge option uses the smallest cell size from all of the input raster tiles and will resample the data into a single tile. Note that merging terrain into a single terrain layer is not the default behavior as this is not the most efficient method of storing elevation data.

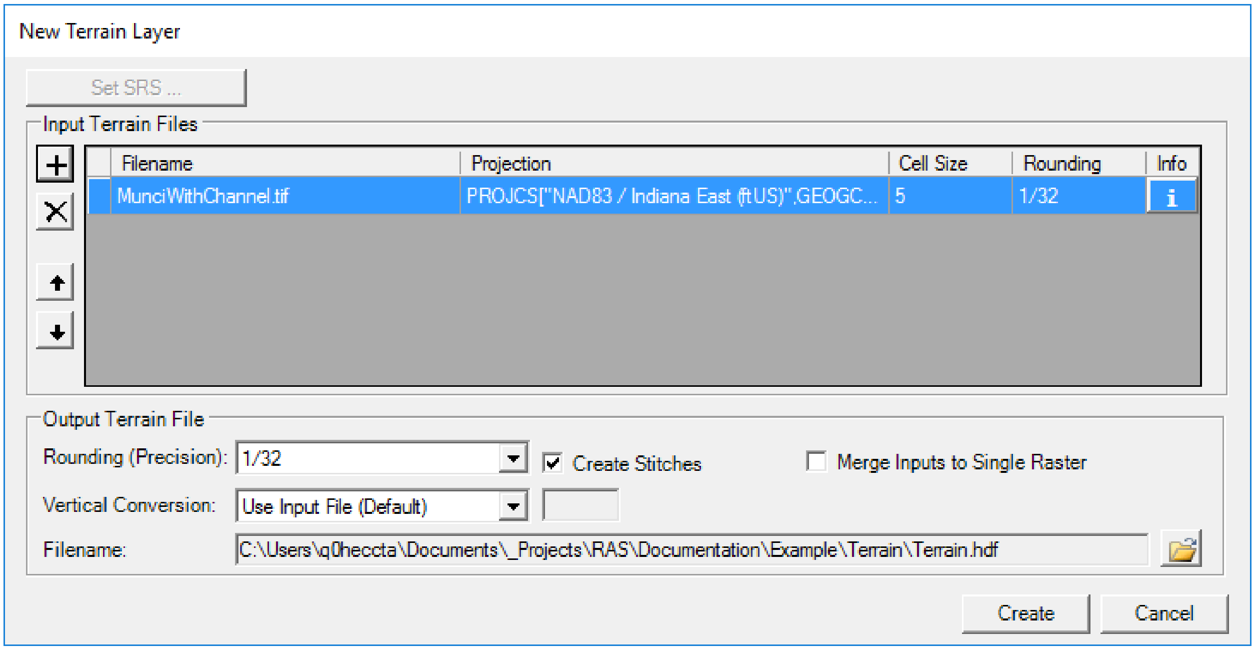


Figure 3-2. New Terrain dialog.

A status dialog will show the progress of creating the terrain. Close the status dialog when finished and the layer will be added to the Layers List in the RAS Mapper Layers Window as a **Terrain** layer (Figure 3-3). The Display Window (or map window) extents will be zoomed to the full extent of the terrain model. Zoom into the area of interest (from the **Standard** toolbar click the **Zoom In** tool) and start editing!

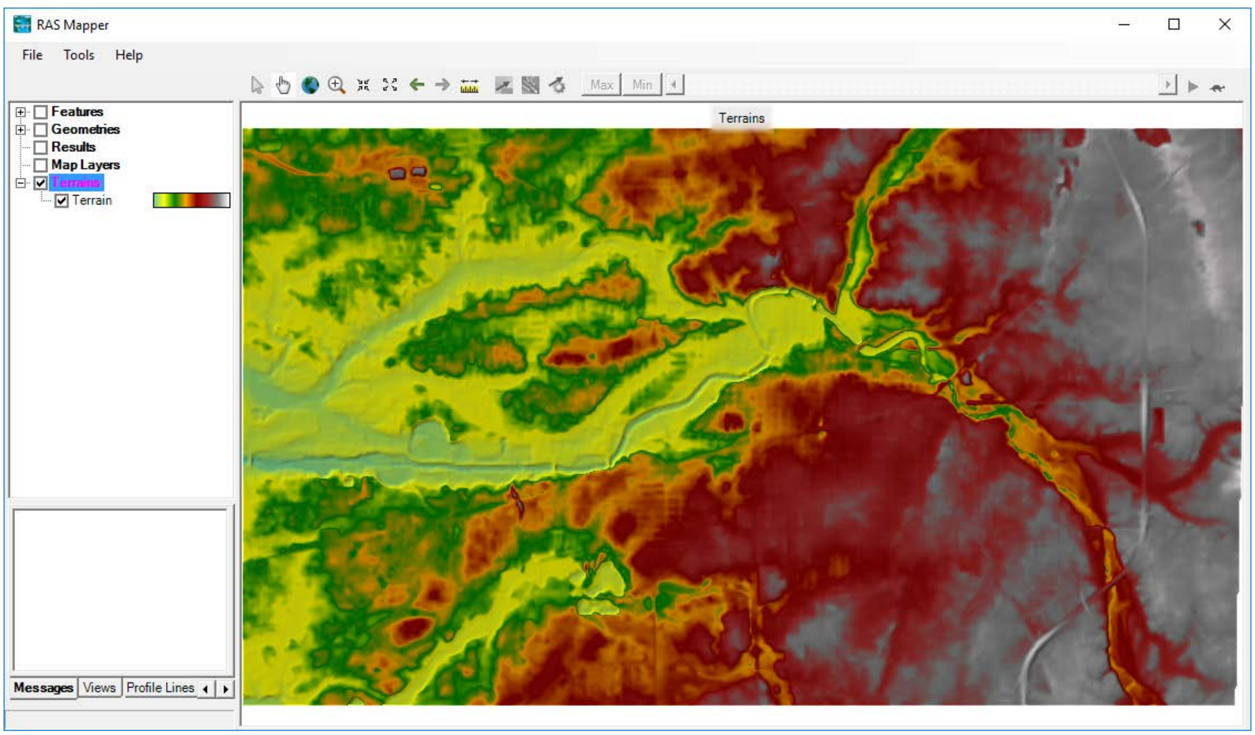


Figure 3-3. Terrain model loaded in RAS Mapper.

3.1.2 Creating a New Geometry

The first step to using the Editing tools in RAS Mapper is to create a new Geometry group. This can be done in the **Geometric Data** editor (Figure 2-8), but can also be done in RAS Mapper by right-clicking the "**Geometries**" group, from the Layers Window, and clicking the **Add New Geometry** shortcut menu item (Figure 3-4). Provide a name for the geometry in the opened **New Geometry Data** dialog (Figure 3-5), which is the same dialog users would see if a new geometry is added from the **Geometric Data** editor.

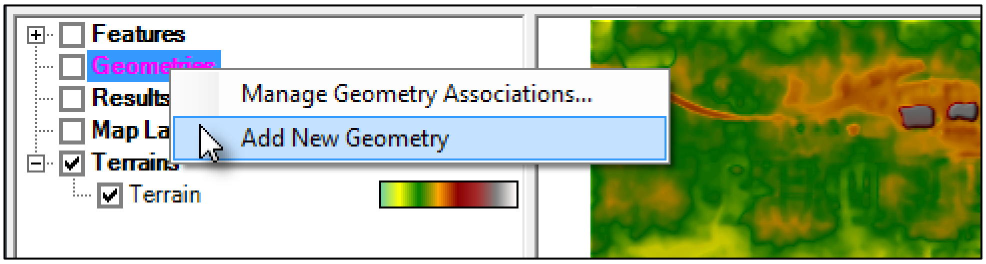


Figure 3-4. Adding a New Geometry in RAS Mapper.

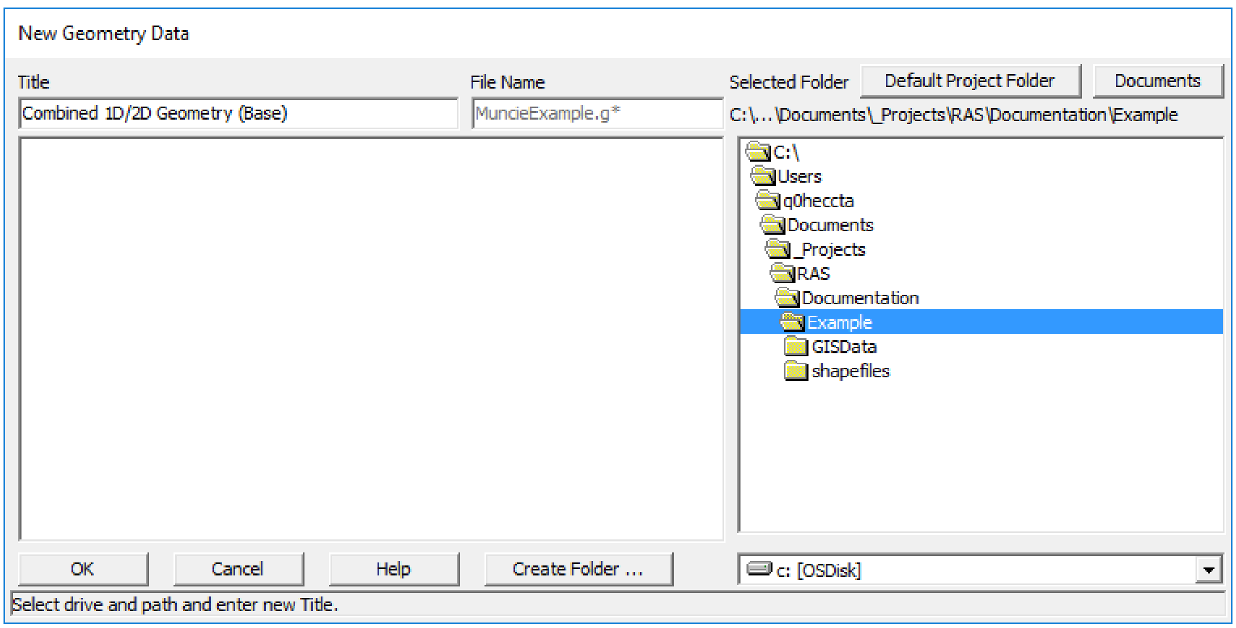


Figure 3-5. Create New Geometry Dialog.

Once a new **Geometry** data file (***.g01** to ***.g99**) has been created, it will be added to the **Geometries** group in RAS Mapper (Figure 3-6). The framework for all of the layers available to the user will be added to the new geometry group that was added. Further, the new geometry will be associated with the first available terrain layer! Users can modify this by clicking the **Associate Terrain Layer** shortcut option (refer to Figure 3-7 and Table 3-1).

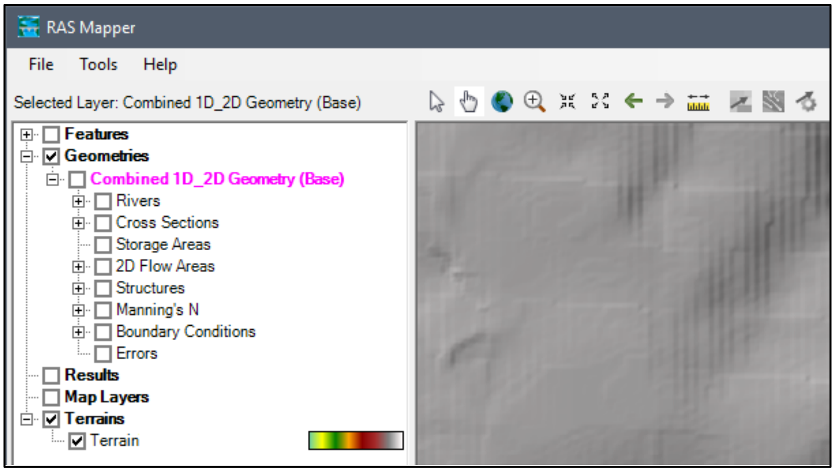


Figure 3-6. Geometry Group Layers Added in RAS Mapper.

WARNING! When using the Editing tools in **RAS Mapper**, users should ***not*** have the **Geometric Data Editor** open. RAS Mapper passes information to the Geometric Data Editor and the Editor passes information to RAS Mapper through the same file. If both RAS Mapper (in Edit mode) and the Geometric Data Editor have the same geometry open, DATA WILL BE LOST! HEC-RAS will close the Geometric Data Editor (as a subtle reminder) when you start editing in RAS Mapper.

3.2 RAS Mapper Editing Tools

Vector features are editable in RAS Mapper, using the **Edit** tools provided. To start an editing session for a specific vector layer (e.g., feature, geometry, or result layers), from the Layers Window, right-click the layer and click either the **Edit Layer** (editing shapefiles) or **Edit Geometry** (editing a RAS Layer) shortcut menu item. Users can also start an editing session for a geometry group by right-clicking on a **Geometry Group** name (e.g., *Combined 1D_2D Geometry (Base)* in Figure 3-7), and clicking **Edit Geometry**. The **Edit Geometry** shortcut option makes all layers for the selected geometry group editable. Furthermore, during an editing session, only the specific layer selected (turns **magenta**, bright pink) from the Layer List will be editable in the Display Window.

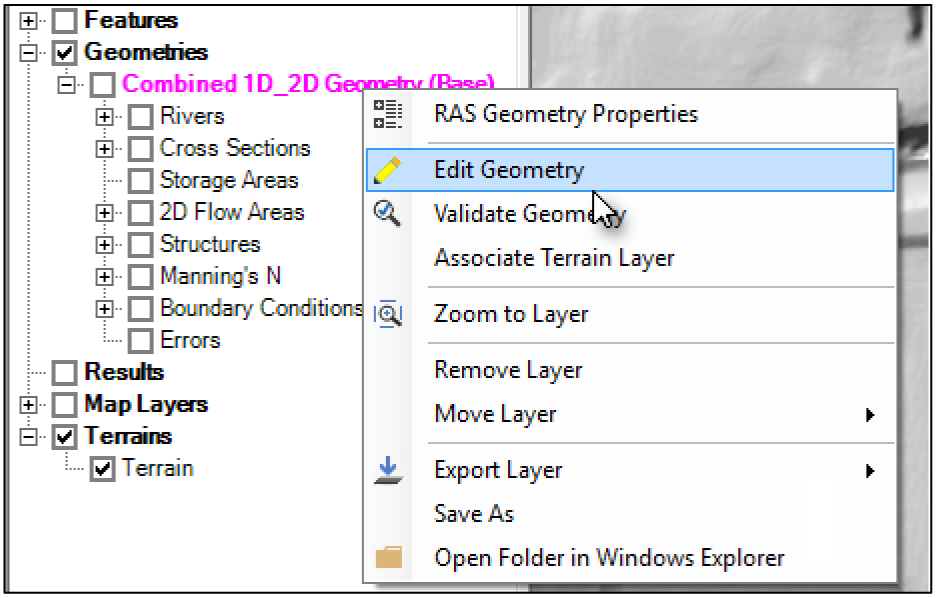


Figure 3-7. Initiate an Edit Session – Edit Geometry Shortcut Menu Option for Geometry Group.

Table 3-1. New General Editing Layer Shortcut Menu Options Available through Right-Click.

Tool	Description
Open Attribute Table	Provides access to view the list of created or edited features and descriptions for the selected layer.
Edit Geometry or **Edit Layer**	Either option begins an editing session for the selected RAS Layer or shapefile layer.
Stop Editing	Ends an active editing session for the selected layer or group of layers.
Validate Geometry	Searches the selected geometry group for errors and creates an **Errors** layer that lists identified issues. Right-click the **Errors** layer and click **Open Attribute Table** to view the list of features and error descriptions. Alternatively, double-click the **Errors** layer to open the attribute table.
Associate Terrain Layer	Opens an interactive help dialog to give the user quick tips on the editing options.

When a geometry is in an edit session, the **Edit** icon will appear next to the name of the **Geometry Group** (e.g., *Combined 1D_2D Geometry (Base)* in Figure 3-8). The **Edit** icon will also appear next to the currently selected layer (e.g., *Rivers*) – the **Edit Layer** – and the **Edit** toolbar will be added to the top of the Display Window, as shown in in Figure 3-8.

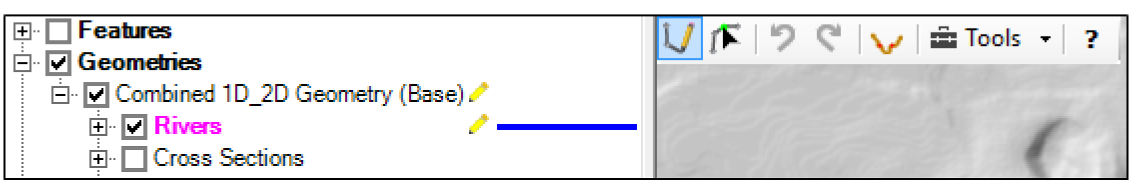


Figure 3-8. The Edit icon is used to show the user the Edit Layer (which is also the Selected Layer).

When finished editing, select the **Stop Editing** shortcut menu item (right-click on the layer) from either the **Geometry Group** (e.g., *Combined 1D_2D Geometry (Base)* in Figure 3-8) or the geometry **Edit Layer** (e.g., *Rivers*). Either way, a dialog will open to request confirmation to save edits before ending the editing session (Figure 3-9).

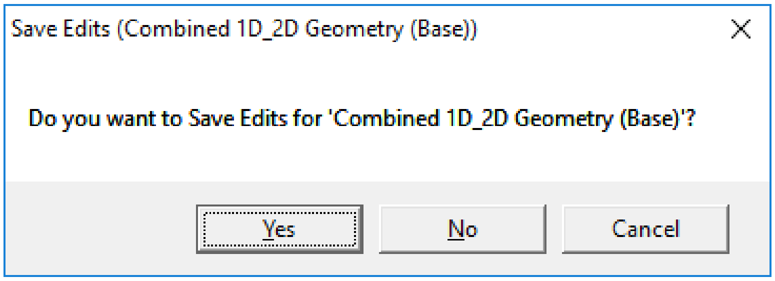


Figure 3-9. Confirmation to Stop Editing and Save Edits.

WARNING! When editing in **RAS Mapper** the Geometric Data Editor should NOT be open in HEC-RAS (with the current geometry loaded). Simultaneous edits in HEC-RAS will overwrite each other resulting in a complete loss of data. Keep the **Geometric Data** editor closed when editing data in RAS Mapper.

3.2.1 Edit Toolbar

The **Edit** toolbar (Figure 3-10), is active when the user is in an edit session and an editable layer is selected (e.g., the *Rivers* layer in Figure 3-8). The **Standard** and **Animation** RAS Mapper toolbars (Figure 3-11) will also be active while during an edit session, so it is possible to switch to the **Pan** or **Zoom** tools while editing; however, the **Select Features** tool must be clicked to reactivate the **Edit** toolbar. At this time, only vector layers are editable. A summary of **Edit** tools is provided in Table 3-2. Refer to Chapter 20 in the HEC-RAS 5.0 *User's Manual* for information regarding the RAS Mapper **Standard** and **Animation** tools.

Figure 3-10. The RAS Mapper Edit Toolbar.

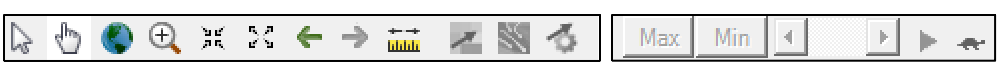


Figure 3-11. The RAS Mapper Standard (*left set*) and Animation (*right set*) Toolbars.

In the Display Window there are default colors (which are modifiable) available to help the user visualize what will happen next within the editing session. The three main visualization symbols useful when editing features in the Display Window are:

- The *Selection* color (**magenta**, bright pink, is default) is used to indicate that a feature is selected and can be copied or feature specific options (e.g., plot terrain profile) can be performed. A selected feature can also indicate that the feature is in edit mode and "opened for editing" (*left box* in Figure 3-12).

- Once a feature is open for editing, the symbology will change from the selection color to the *Vertex Editor* symbology (**grey** line with **black** points, is default). The *Vertex Editor* symbology is used when creating a feature, or when editing the vertex points for a selected feature (*center box* in Figure 3-12).

- The *Action* color (**chartreuse**, bright green, is default) will indicate that an action will occur if the user clicks. For instance, if the mouse hovers over a point, it will change to the action color: This is indicating that the user can click and "grab" to move, delete, or add a point. Further, for line or polygon features, the displayed action color lines indicate how the feature would be modified if the point was inserted (*right box* in Figure 3-12). The control for how close the mouse cursor must be to a line for a point to be selected is the "Near Point" tolerance, and the "Near Line" tolerance controls the maximum distance the mouse cursor can be to insert a point on a line or polygon.

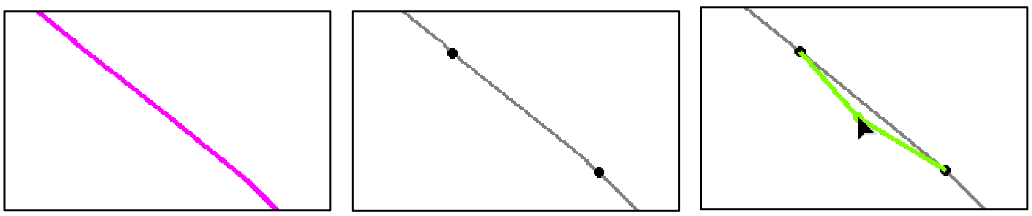


Figure 3-12. Selection (*left box*), Vertex Editor (*center box*), and Action (*right box*) Colors.

Feature visualization colors, tolerance values, and other RAS Mapper symbols can be modified from **RAS Mapper Options** dialog, opened from the **Tools | Options** menu (Figure 3-13).

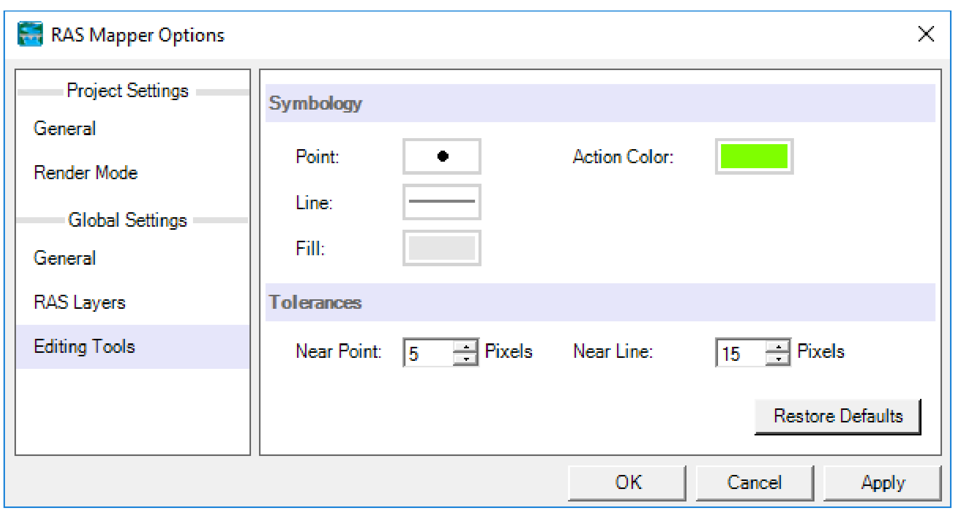


Figure 3-13. RAS Mapper Options Dialog for Modifying Default Symbology.

Table 3-2. The RAS Mapper Edit Toolbar Options.

Tool	Description
Add New	Adds a new feature. Left-click to Add a point. Double-click to Finish editing the feature.
Select/Edit Feature	Select features by left-clicking and dragging. Double-click on a selected feature to Open a feature to start editing. Double-click to stop editing (Close) an open feature. Insert a point with a left-click near the line. Move a point by left-click on a point and drag. Delete a point by selecting it and pressing the **Delete** key.
Undo	Undo the last edit operation.
Redo	Redo the last edit operation.
Plot Elevations	Plot the elevation profile under the selected feature.
Tools	There are many edit operations that will work on the selected set. **Copy**, **Paste** and **Delete** feature options are available. **Reverse Feature** will reverse the order of the points in a line feature. **Merge Feature** will combine the selected features. For line features, this tool is very robust, but for polygon features, the polygons must overlap. **Clip – Preserve** will clip against overlapping polygons, preserving the selected feature. (Polygons must intersect.) **Clip – Discard** will clip against overlapping polygons, discarding the overlapping portion of the selected feature. (Polygons must intersect.) **Buffer Polygon** will enlarge or shrink the selected feature by the buffer value. **Filter** will filter the points on a feature given a user tolerance. The Douglas-Peucker-Ramer and Minimum Area Reduction filtering algorithms are implemented with this tool. **View/Edit Points** will bring up a dialog with all of the points X and Y coordinates in the selected feature. Users can then modify the points in the table.
Help	Opens an interactive help dialog to give the user quick tips on the editing options.
Additional Options	Right-click to re-center the Map View on the cursor. Hold the **Shift** key to **Pan** or use the middle mouse button to **Pan**. Use the **Tab** key to switch between **Add New** and **Edit Feature** modes. **Append** to a line by **opening a line for editing**, click on the **Add New** feature and start adding on to the line. Use the **Spacebar** and click to **Select** a feature, the Selected Layer will switch based on the feature that was clicked.

Add New

During an editing session (indicated by the **Edit** icon) the **Add New** tool can be utilized to create new features in the Display Window. Left-click to place a new point or vertex on a line or polygon. As shown in Figure 3-14, the location of the new vertex point (as well as the new segment that will be created for a polyline or polygon feature) will be shown in the *Action* color (**chartreuse**, bright green, is default) defined in the **RAS Mapper Options** on the **Global Settings ❙ Editor** tab (accessed from the **Tools ❙ Options** menu item).

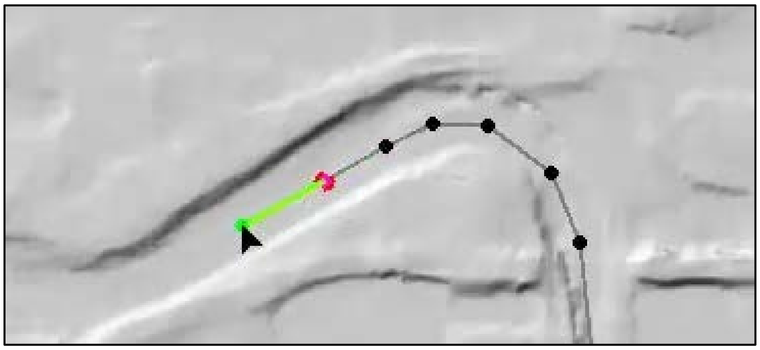


Figure 3-14. Creation of a Polyline Feature.

When finished with adding the new feature, double-click to end the polyline or polygon (point features do not require a double-click). In general, once a new feature has been created, a dialog will open requesting that either the default name be accepted or a user-defined name be provided for the feature. For instance, after completing a river line, the **Rename River and Reach Name** dialog (e.g., Figure 3-18) will open requesting a unique name for the river and the reach. The naming dialog will provide a unique default name for the feature, so it is optional for the user to enter a different unique name.

Append to a line. Appending to a line feature can be accomplished by opening the feature for editing using the **Select/Edit Feature** tool. Once open, select the **Add New** tool (or press the Tab key). The feature will be in “Add New” to the end of the line mode where the user can continue extending the line feature with new vertices.

Switch Edit Layer. If you find yourself adding features to the wrong layer, hold the spacebar and click on a feature from the layer you intended to edit. The Edit Layer (Selected Layer) will swap to the layer of the feature you clicked and you can start adding new features to it.

Select/Edit Feature

Once a feature has been created, the **Select/Edit Feature** tool allows users to edit the existing feature during an editing session (indicated by the **Edit** icon). There are two main editing options available for the **Select/Edit Feature** tool. The first option is to modify the entire feature (move, copy, delete, etc.) and the second option is to modify a portion of the feature (move, add, or delete vertex points, etc.). When utilizing the first option to modify the entire feature, while the **Select/Edit Feature** tool is active, the selected feature will be drawn in the *Selection* color (**magenta**, bright pink, is default as displayed in Figure 3-15). In order to use the second editing option to modify a portion of the feature, the feature must be "open for editing" and the selected feature will be drawn in the *Vertex Editor* symbology (**grey** line with **black** vertex points, is default as displayed in Figure 3-17). Both the *Selection* color and the *Vertex Editor* symbology may be changed in the **RAS Mapper Options** dialog (accessed from the **Tools | Options** menu item).

To Select a feature, click on the feature or perform the selection using the click-drag maneuver (Figure 3-16). Once selected (**magenta** color), users can modify the entire feature by performing one of the available options (e.g., delete, left-click to drag it, right-click to get a list of options available for the feature). Figure 3-15 displays available right-click shortcut options for the selected polyline feature.

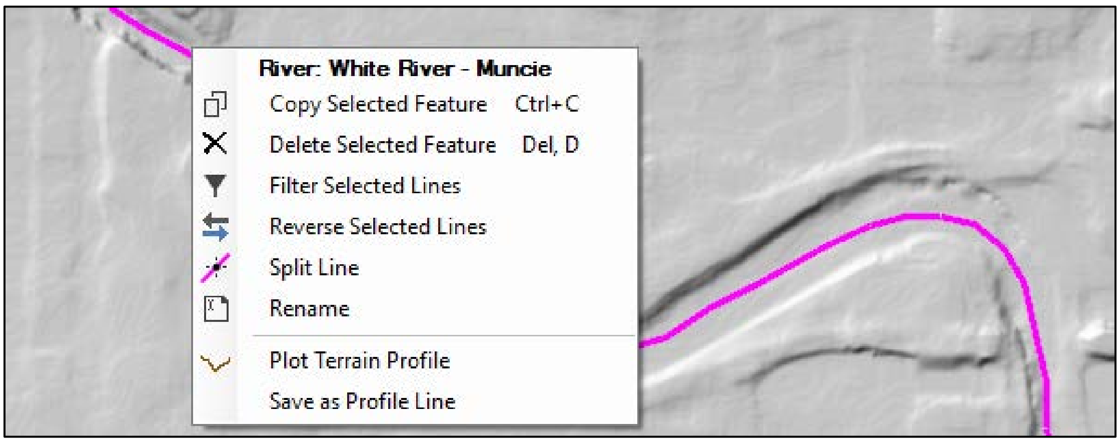


Figure 3-15. Options Available for the Selected Features by Right-Clicking the Feature.

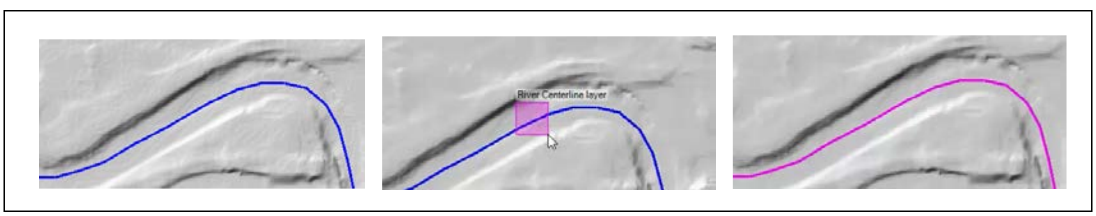


Figure 3-16. Selecting a Feature can be done by drawing a Selection Box over the Feature.

The feature can also be "open for editing" by double-clicking the feature which opens it in the vertex editor symbology (**grey** line with **black** vertex points). Typical editing operations such as inserting, moving and deleting points can then be performed on the open feature through a left-click (Figure 3-17). To perform operations such as deletion of multiple points, perform a selection using click-drag and then press the **Delete** key.

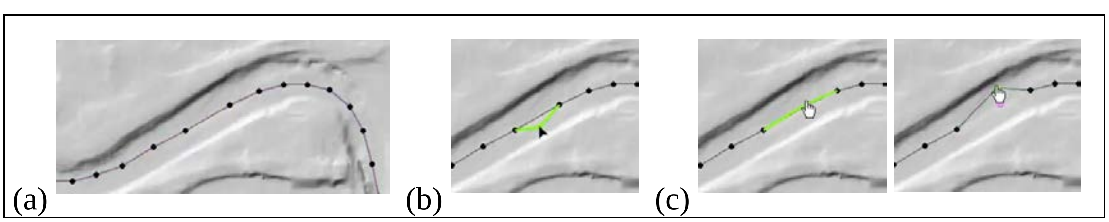


Figure 3-17. A Feature (a) Open for Editing; (b) Inserting a Point; and (c) Moving a Point.

To finish editing a feature, double-click off of the feature to close it. When a feature is closed, RAS Mapper ***may*** re-compute properties for the feature. Properties are always computed for NEW features. Features are considered new if the feature was newly constructed in the current edit session. However, after closing an editing session (right-click the layer and click **Stop Editing** from the shortcut menu), the feature will be considered "old" and properties ***may not*** be automatically updated.

For example, a user just started creating cross section cut lines utilizing the **Add New** tool. In this case, RAS Mapper will compute the elevation profile after each cut line is added. Furthermore, if a cross section line is selected, utilizing the **Select/Edit Feature** tool, and moved during the same editing session, then RAS Mapper will continue to update the cut line elevation data for moved/modified cross sections. However, if the user clicks the **Stop Editing** shortcut option to end the editing session, and then later starts a new editing session (clicks the **Edit Geometry** shortcut option) for the same cross section cut lines, then the user may need to manually have RAS Mapper update the elevation data (e.g., **Compute | Elevation Profiles from Terrain** Cross Section geometry layer shortcut menu option). This manual update will allow users to modify a cross section in the Geometric Data Editor, such as add channel bathymetry, without having RAS Mapper mistakenly overwrite the data during the next edit session.

Auto Updates

"Old" geometric data will not automatically be updated with new information in RAS Mapper. This will allow the user to open an existing RAS model geometry in RAS Mapper without the unintended consequence of updating cross section river stations or changing the elevation data. Manual update of the data can be performed by right-clicking on the Cross Sections Layer and selecting one of the **Compute** options (or right-clicking on a selected set of features). If you would like your data to automatically update as if the geometry were brand new, Turn on the **Auto Update Geometry** menu option available from a right-click on the Geometry group layer. Turning on the auto-update feature will place a check next to the menu option.

By turning on the **Auto Update Geometry** feature, each time a cross section is edited, data associated with the cross sections will be reevaluated: river stations, bank stations, reach lengths, and elevation data will be recomputed for the selected cross section. Reach lengths will also be evaluated for the cross section upstream from the edited cross section. The auto-update capability will also occur for selected features corresponding to specific RAS Layers: if the River or Flow Paths layer is edited, reach lengths will be updated; if the Bank Station layer is edited, bank stations are recomputed. (**Note**: Cross section river stations are NOT recomputed if the River layer is edited. You will have to re-compute river stations manually.)

Individual parameters can be in auto-update mode. These are accessible by right-clicking on the Geometry of interest and selecting the **RAS Geometry Properties** menu item. The **Editor** tab will provide a list of updatable properties. Placing a check next to each property will allow for the auto-update option to occur for just that property. These options are saved with the geometry, s RAS Mapper remember the auto-update settings for the next edit session.

3.2.2 1D Model Layers

RAS Mapper will provide the framework for the geometric layers that need to be constructed and as features are constructed, the user will be prompted to complete information specific to that layer. For instance, when creating a new river line, the user will be prompted to enter a River Name and a Reach Name. The Layers are listed in the order in which the data should be completed; however, the layers are rendered in a specific order that will allow for good visualization of the data. **[Note:** In future versions of HEC-RAS, the user will have full control over rendering order.**]** A summary of layers used to create a 1D model is shown in Table 3-3.

Table 3-3. 1D Modeling Layers.

<table>
<tr><th>Layer</th><th colspan="2">Description</th></tr>
<tr><td rowspan="5">Rivers</td><td colspan="2">The Rivers layer is used to establish the river network. It must be created in the downstream direction. The Rivers layer will be used in concert with the XS Cut Lines layer to establish river stationing for each cross section and compute the main channel reach length between cross sections.

The Rivers Layer organizes similar features in the River Group that determine how water flows in the river network. The River Group includes:</td></tr>
<tr><td>Junctions</td><td>Junctions are automatically created at the confluence of three river reaches.</td></tr>
<tr><td>Bank Lines</td><td>This layer is used to establish the main channel bank stations for the cross sections and should not intersect the Rivers lines.</td></tr>
<tr><td>Flow Paths</td><td>Flow Path lines are used to compute cross section reach lengths from cross section to cross section in the left and right overbanks. The river centerline will be used to compute the main channel reach length. If the flow paths layer is not specified the main channel reach length will be used in the overbanks.</td></tr>
<tr><td>River Station Markers</td><td>The River Station Markers layer is a point layer that can be used to manually assign river stationing along the River Centerline. Values are linearly interpolated between assigned station values.</td></tr>
<tr><td rowspan="2">Cross Sections</td><td colspan="2">Cross Sections (XS) are used to establish the spatial location and alignment of cross sections. Cross section elevation profiles will be extracted from the terrain model. Other cross section properties are extracted based on their intersection with other layers.

The Cross Sections layer organizes cross-section specific layers into a Cross Sections Group, which includes:</td></tr>
<tr><td>Bank Stations</td><td>A point layer which identifies the location of the bank station on the cross section.</td></tr>
</table>

Layer	Description	
	Edge Lines	**Edge Lines** are used connect the ends of cross sections. This layer may be edited *between* cross sections; however, the edge line points at the cross section endpoint may not be modified.
	Interpolation Surface	The **Interpolation Surface** is not editable. This layer is constructed from the River Centerline, XS Cut Lines, Bank Lines, and Edge Lines. The interpolation surface is used for mapping HEC-RAS results.
Storage Areas	**Storage Areas** are a set of polygon features that can be used to extract an Elevation-Volume relationship.	
Errors	The **Errors** layer is designed to assist the user in identifying geometric mistakes. For instance, if a cross section is intersected by the river line more than once, and error will be produced.	

River Centerline
The River Centerline is used to represent the river network for flow connectivity. Rivers layer must be created in the downstream direction. When a River layer has been completed, the **Rename River and Reach** dialog will be invoked (Figure 3-18). Either accept the default names provided or enter new names. **[Note:** River names must be unique throughout the model and reach names much be unique for each river.**]**

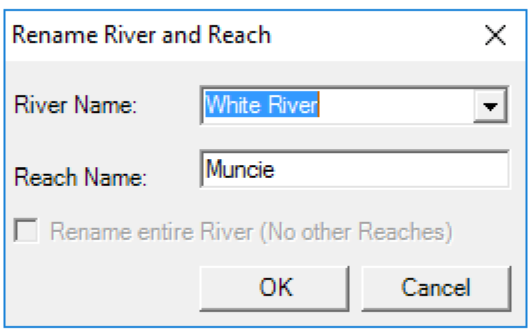


Figure 3-18. Rename River and Reach Name dialog.

The same **Rename River and Reach** dialog is available by right-clicking on a river reach and selecting the **Rename** menu option. If you wish to rename the river, turn on the option to “Rename Entire River”. This will replace the river name for all river segments that had the previous name (all other Reaches).

River Station Markers
By default, RAS Mapper will use the horizontal units of the project for calculating river lengths and corresponding River Stations. Typically, the default units then will be feet or meters. To change the river stationing to miles or kilometers, select the **Tools | Options** menu item and change the **XS River Stationing Units** from the **Project Settings | General** option. The user can also specify the number of decimal places for the river station units.

A more complex option allows users to create river stations along the river centerline. The **River Station Marker** layer is a point dataset that allows the user to specify a specific river station at a point along the river. RAS Mapper will then compute the relative length between the river station marker points and use an interpolated value for the cross section's river station. This option is helpful on rivers that have a historic river station (e.g., a known river station needs to be fixed at one or more locations).

Bank Lines

Bank Lines are used to define the main channel banks for a cross section. If the bank lines are not defined, the bank stations will be set to the ends of the cross sections. Banks lines may be drawn continuous or discontinuous, in either the downstream or upstream direction, but it is recommended to have a left and right bank line for each river line. **[Note:** Make sure the bank lines do not intersect the river lines or each other and make sure the bank lines cross XS cut line only once.**]**

For existing model data, bank lines can be auto-generated from bank stations by from a right-click on the **Bank Lines** layer and selecting the **Compute Bank Lines from XS Bank Station** menu item. The shape of the river line will be used to connect the bank station locations from one cross section to the next.

Flow Paths

Flow Path lines are used to compute the reach lengths between cross sections and are placed at the center-of-mass of flow in the left and right overbanks. They should be drawn in the direction of flow, never intersect river lines and only intersect a cut line once. The river centerline is used for the main channel flow path and should not be created in the Flow Paths layer.

Junctions

Junctions are used to connect river reaches. Junctions will be automatically created for the user within RAS Mapper. There are two ways a junction will be formed: (1) a tributary reach will split a main river reach or (2) the end points of three reaches will be moved and snap together. For either case, RAS Mapper will give the visual feedback to assist the user. Discussion of how to form a junction is provided below.

Splitting a river reach to form a junction is the most-likely scenario. To do so, create the second river such that the downstream endpoint lands on the first river at the location to split the river. Double-click to end the new river. The user will be asked to provide a name for the new river reach. The user will then be prompted to create the junction through a series of steps: (Step 1) splitting the existing river (*Figure 3-19*), (Step 2) renaming the downstream reach of the existing river (*Figure 3-20*), and (Step 3) naming the new junction (*Figure 3-21*).

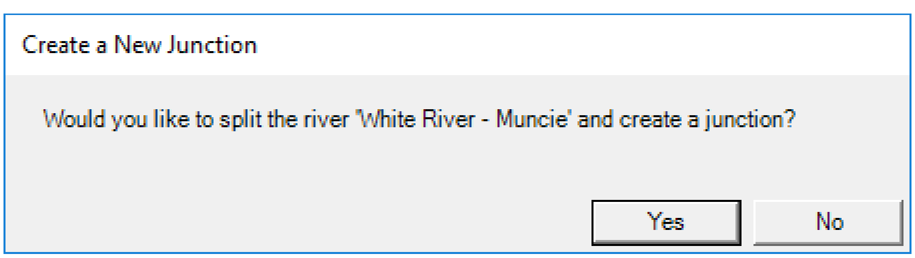


Figure 3-19. Split river reach dialog prompt.

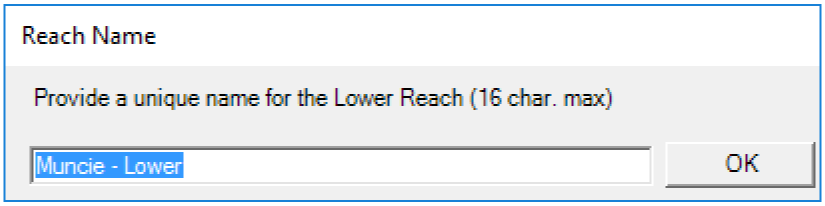


Figure 3-20. Provide a new name for the downstream reach.

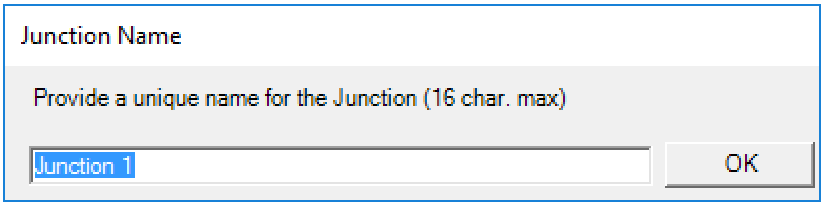


Figure 3-21. Provide a junction name.

An alternative way to create a junction is by moving the endpoints of three reaches close enough together that the points "snap" together and form a junction. If/when all three reach endpoints are within the snapping distance tolerance, a junction will be formed. Note, the junction is formed when the editable reach is closed (double-click to close the open feature). An example of this process is illustrated in *Figure 3-22*.

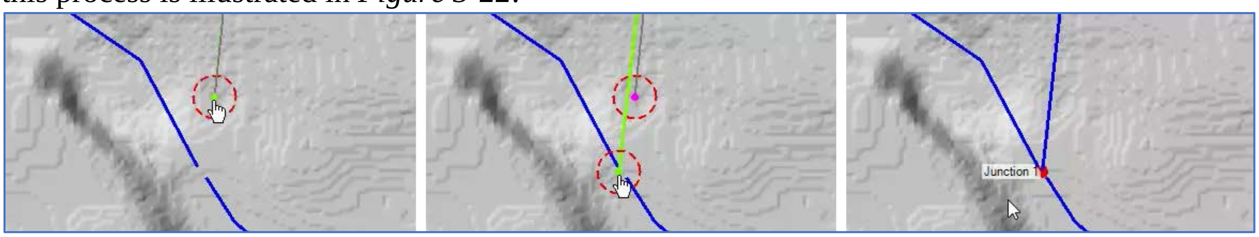


Figure 3-22. A junction is formed when three reach endpoints are within the junction tolerance (indicated by the *red, dashed circle*).

Cross Sections

Cross sections (XS) are developed based on the location layout of XS cut lines. Cross sections should be laid out perpendicular to where water will flow in the channel and overbank areas. Therefore, most cross section cut lines should be created from a minimum of four points. Cross sections will also be visualized when looking in the downstream direction; therefore, they should be created from left to right when looking downstream (RAS Mapper will automatically flip the cut line to have the correct orientation). There are many considerations when developing cross section data for orientation, locating, and spacing. Use the terrain, river centerline, bank lines, flow path lines, inundation mapping, and other data to properly place XS cut lines.

After each cross section is created, RAS Mapper will automatically compute the River Name, Reach Name, River Station, Bank Station, and Reach Length data for the cross section (assuming the corresponding layers were already created) assuming the cross section is considered a "new" feature. Elevation data will also be automatically extracted. Once an editing session has been terminated (Stop Editing), the cross section features will be considered "old" to minimize accidental data changes or breaking existing linkages with flow and boundary condition data. In order to update the properties of "old" (existing) cross sections, use the manual Compute process or turn on the Auto Update Geometry option.

To manually update cross section information such as the elevation profile data in subsequent sessions, users will need right-click on the **Cross Sections** layer and select **Compute | Elevation Profiles from Terrain**. Updating other cross section properties is also done though the **Compute** menu, as shown in *Figure 3-23*.

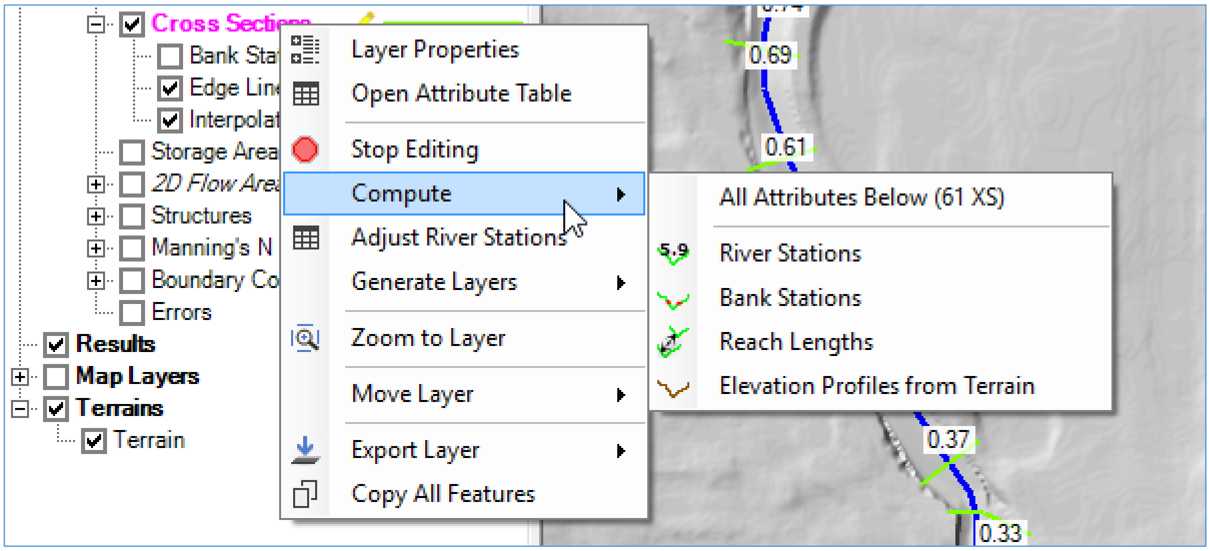


Figure 3-23. Manual update of Cross Section properties using menu items. The number of items that will be processed is designated in parenthesis "(61 XS)".

When working with an existing RAS geometry, cross section river stations will have been derived by hand or using a GIS to calculate the river station. Perhaps the river stations are based on the river centerline that is available from current geometry, in which case as you add new cross sections the new river stations will be computed harmoniously where the new section is inserted exactly as expected. This scenario, however, is not always the case. For the case where you want to specify the river station for the downstream cross section and have all others computed based and a distance from that cross section, there is the **Adjust River Stations** tool. This tool is especially useful when working in a team where each team has a specific portion of the river that will later be merged together into one long river model.

While Editing a model, right-click on the **Cross Sections** layer and select **Adjust River Stations**. The dialog that appears allows you to select the river to work on and specify the starting river station for the downstream cross section in the upper table (Figure 3-24). Select the units to work with and the number of decimal places and select the **Create New River Stations** button to create the new river stations. In the lower table, the new river station will be generated. The user can edit the values in the river station table at any time.

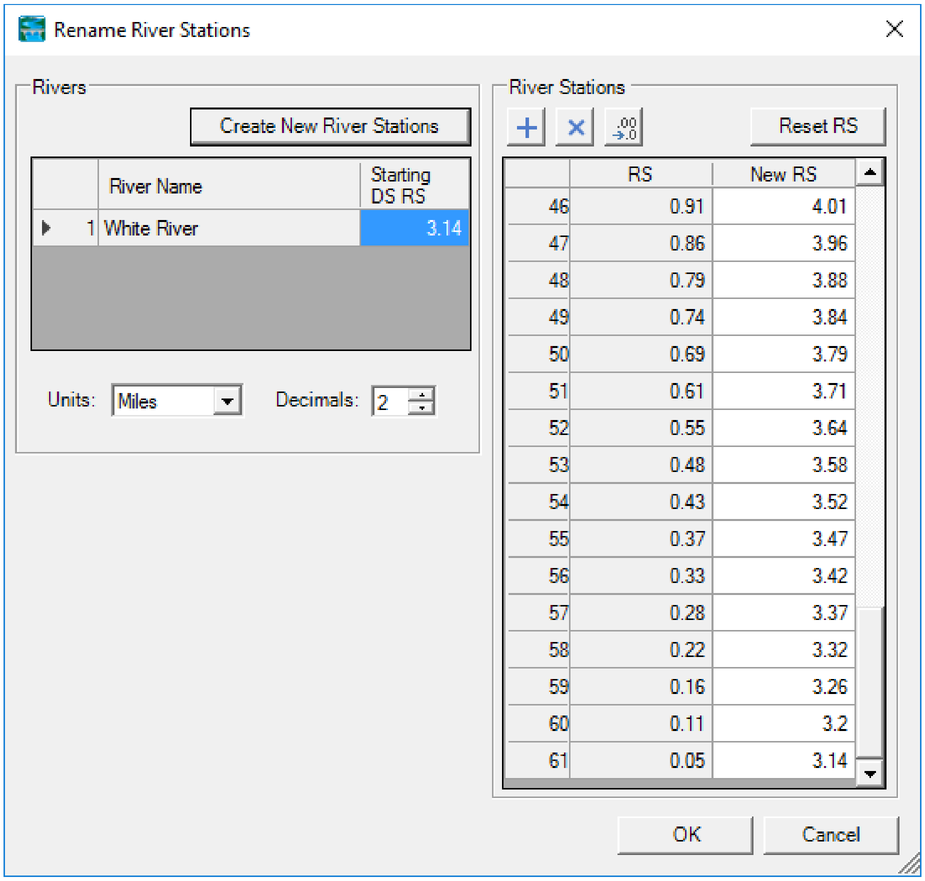


Figure 3-24. Adjust River Stations dialog.

Press **OK** to apply the new river stations to cross sections. After river stations have been set for the cross sections, as new cross sections are added to the model, the river station will be interpolated based on the proximity of the new cross section and the existing, bounding cross sections. This will ensure the correct computational order for all cross sections.

The Reset RS will reset the river station values to the river station values present prior to entering the table. The table has copy and paste functionality as well as some selected area editing options.

Edge Lines

Edge lines are used to limit the extents of inundation mapping and are typically the bounds of the cross sections. These are an editable feature. This allows users to map an area that is in backwater that is not included in the cross section layout. The edge lines should be modified prior to running your model because they will be copied to the results data and used in mapping. **[Note:** The user can only modify the edge lines between cross sections and may not move the point that is on a cross section.**]** Lastly, create the cross section interpolation surface once the edge lines have been completed to see what results.

Edge lines can be auto-computed by right-clicking on the **Edge Lines** layer and selecting **Compute Edge Lines at XS Limits**. The edge lines are computed based on the shape of the stream centerline and the XS Cut Lines layout. This will give the user a good starting point for modification.

XS Interpolation Surface
The XS Interpolation Surface is used for interpolation of hydraulic results. This layer cannot be edited directly. The interpolation surface is dependent on the cross sections, river lines, bank lines, and edge lines, which are all editable features. To see what the interpolation surface will look like, right-click on the **XS Interpolation Surface** layer and select **Compute XS Interpolation Surface**.

Errors
The Errors layer is used to visualize suspected problems with the RAS Geometry. The Errors list is updated each time that a new feature is added to a layer. RAS Mapper will evaluate the feature for missing data or bad intersections with other layers and create a feature and description of the message in the attribute table. The user can force an analysis of possible errors by right-clicking on the **Geometry Group** and choosing the **Validate Geometry** menu item.

The error messages can be categorized as critical, warnings, and notes. Critical errors are problems that are significantly wrong, such as a cross section not having a River Name, Reach Name, or River Station, which could result from the cross section intersecting the river more than once or more than one river line. The result is a geometry that will not run in HEC-RAS. A warning informs the user something most likely is wrong (high confidence) with the model geometry, such as a bank line that crosses the river centerline, which may lead to model errors (the interpolation surface will not be correct, which affects mapping, but the model will run and most likely give good results). A note indicates there may be something wrong (low confidence) with the data, such as a cross section that doesn't intersect with bank lines (RAS Mapper will automatically assign bank stations to the end points).

[Note: The **Errors** layer is currently being improved and developed to assist model creation.**]**

3.2.3 2D Model Layers

RAS Mapper will provide the framework for the geometric layers that need to be constructed for 2D Flow Areas. The Layers are listed in the order that the data should be created; however, the layers are rendered in an order that will allow for good visualization of the data (e.g., point data should be drawn on top of polygon data). **[Note:** In future versions of HEC-RAS, the user will have full control over rendering order.**]** A summary of layers used to create a 2D model is shown in Table 3-4.

Table 3-4. 2D Modeling Layers

Layer	Description	
2D Flow Areas	The 2D Flow Areas layer stores the computed mesh. This Layer also organizes information used to build a 2D Flow Area mesh.	
	Perimeters	A polygon is used to represent the boundary of each 2D Flow Area.
	Computation Points	The “cell centers” are stored in the Computation Points layer.
	Break Lines	Breaklines are line features used to enforce cell faces. Each breakline will have a Name and Cell Spacing information.
	Refinement Regions	Refinement Regions are used to modify the cell spacing in a 2D Area. The regions can be used to increase or decrease the density of the computation points.
Manning’s N (Group)	This group layer has a raster layer of *n* values used and a vector layer to override base values.	
	Override Regions	The Override Regions layers is a vector layer used to override the base Manning’s n value data. The user will specify a description and n value for each polygon in the layer.
	Final *n* values	This layer is a raster composite of base n values and override n values. These are the values used when developing hydraulic property tables for 2D modeling.
Errors	The Error Layer is designed to assist the user in identifying geometric mistakes. For instance, if a 2D cell has too many faces, an error will be produced.	

2D Flow Areas

The **2D Flow Areas** layer groups features that are used to create the 2D Area computational mesh: perimeters, computation points, breaklines, and refinement regions. This layer is not directly editable. A 2D flow area is developed by first adding a new **Perimeter** layer. Then the user can create a mesh by bringing up the **2D Flow Area Editor** (Figure 3-25), entering a base point spacing (DX and DY), and then generating the base cell points. After a base set of points are generated for a 2D Flow Area, users can refine the mesh by entering breaklines and Refinement Regions. When creating the 2D Mesh, the order in which RAS computes the mesh is to use the points from the computation points layer first; second, insert the refinement region points (overriding the perimeter points); and lastly, insert the breakline points (overriding points within a buffered area around the breakline). The perimeter of a refinement region is considered a breakline.

Perimeters

The **Perimeters** layer is used to define the polygon boundary for a 2D Flow Area. Upon completion of drawing a boundary polygon, the user will be prompted to name the 2D Flow Area, as shown in Figure 3-25.

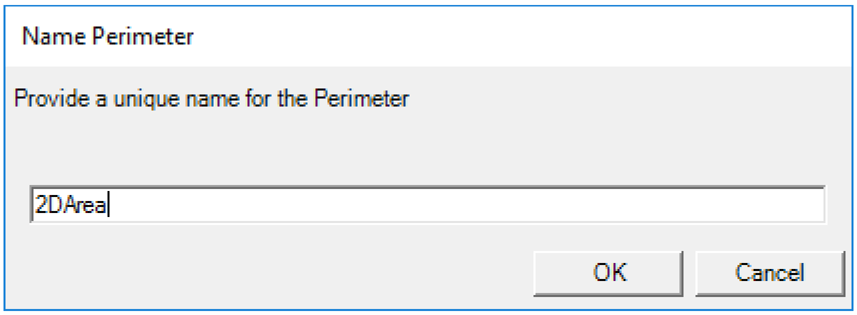


Figure 3-25. Providing a unique name for the 2D Flow Area.

The **2D Flow Area Editor**, shown in Figure 3-26, is then accessed either by right-clicking on the 2D Flow Area feature or right-clicking on the Perimeters layer and selecting the **Edit 2D Area Properties** menu item.

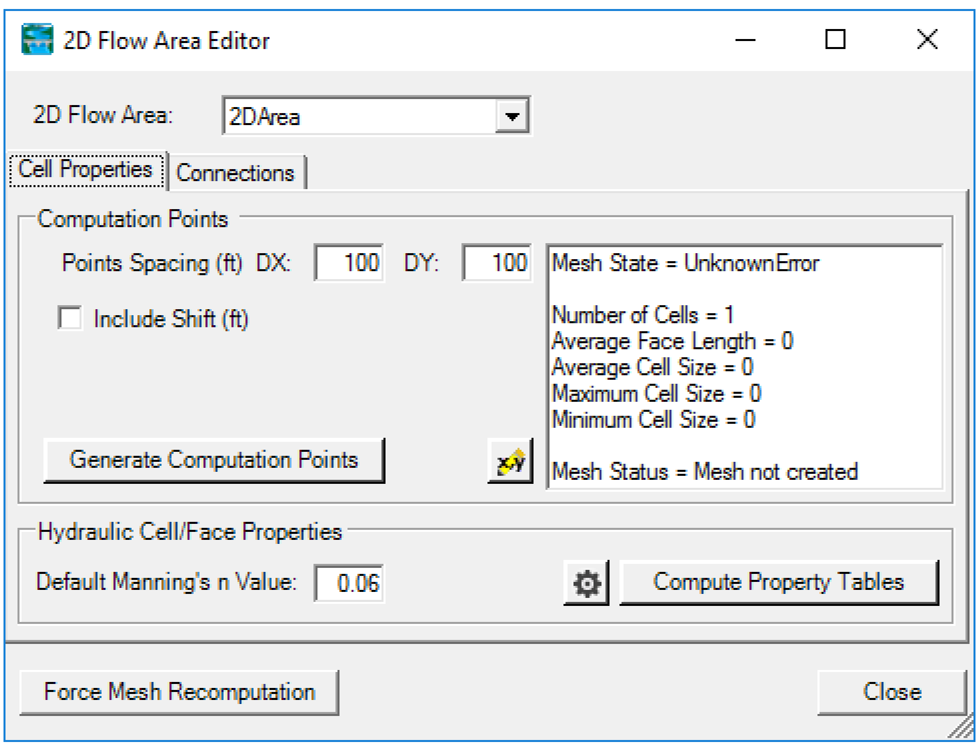


Figure 3-26. 2D Flow Area Editor.

The computational mesh is created based on computation points. Enter the nominal cell center **Points Spacing**, and press the **Generate Computational Points**. This will create a computational mesh based on an evenly distributed "grid" of points and generate cells that have faces orthogonal to the connections of each computation point.

After computing the mesh, the mesh status window will provide information of the success of the mesh generation and some statistics, such as the number of cells in the mesh, as shown in Figure 3-27. The computation points that were used to generate the mesh are stored in the **Computation Points** layer.

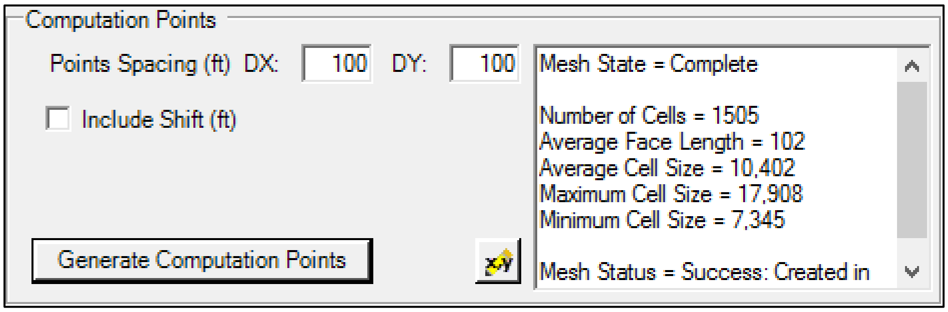


Figure 3-27. Status of the Mesh Generation is displayed on the 2D Flow Area Editor.

The computational mesh will be drawn to the Display Window (Figure 3-28). The visualization of the mesh is controlled by the 2D Flow Areas group (not by the Perimeters layer, which only controls the boundary).

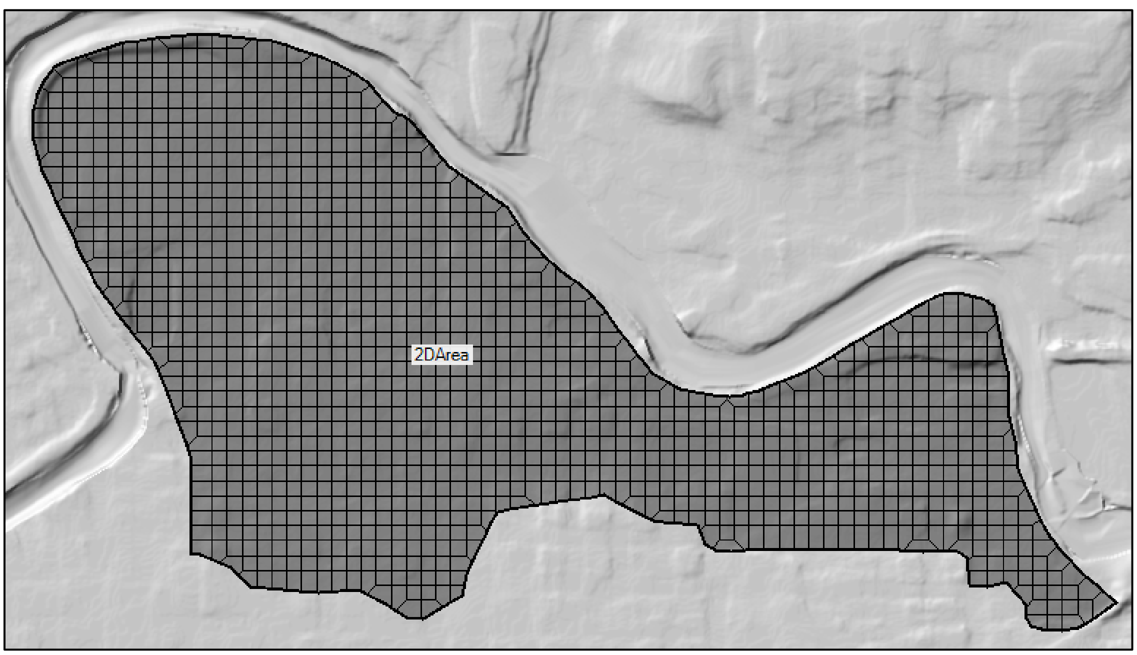


Figure 3-28. Example Computational Mesh.

The **2D Flow Area Editor** also provides access to the points used to generate the computational mesh, by pressing the **Edit/View Points** button. The computation points, however, are visualized by the **Computation Points** layer, discussed below. The **Edit Points** table is available many places in RAS Mapper and allows the user to **Add**, **Multiply**, **Set**, **Replace**, and **Adjust** decimal places for values within the selected area of the table (Figure 3-29). Users can also cut and paste points from and into this editor.

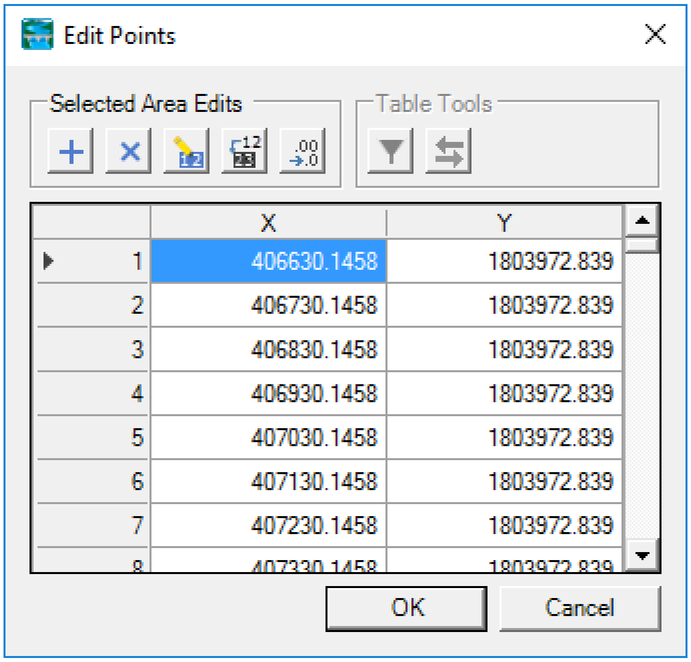


Figure 3-29. Edit Points Table Provides Functionality to adjust Point Values.

The **2D Flow Area Editor** also provides access to a default (base) Manning's n value (see in Figure 3-26) for the 2D area and filter tolerances for the computation of the **Hydraulic Property Table Tolerances** (shown in Figure 3-30).

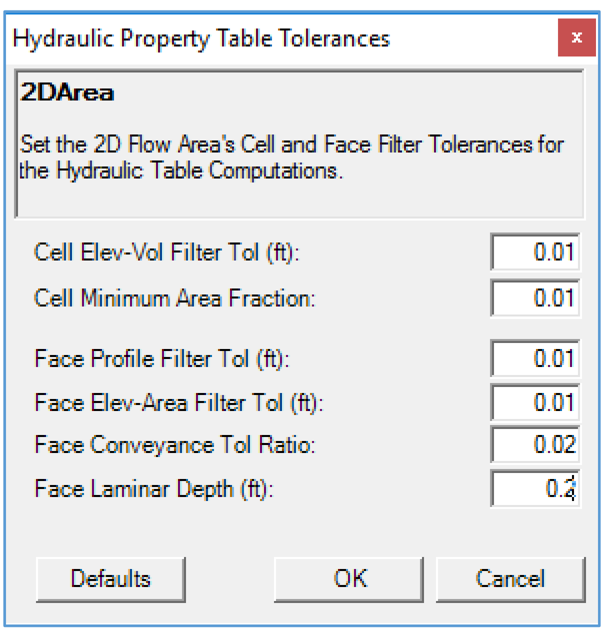


Figure 3-30. Hydraulic property table default values.

Computation Points

Points used to generate the 2D Flow Area meshes are stored on the Computation Points layer. All points for all of the 2D Flow Areas are stored in one multipoint feature in the Computation Points layer. Because there may be many, many mesh points, when in edit mode, the Computation Points layer must be the Edit Layer for the points to be drawn. Further, users must be zoomed in such that there are less than 15,000 points within the view area, for the points to be displayed. An example mesh with points displayed is shown in Figure 3-31.

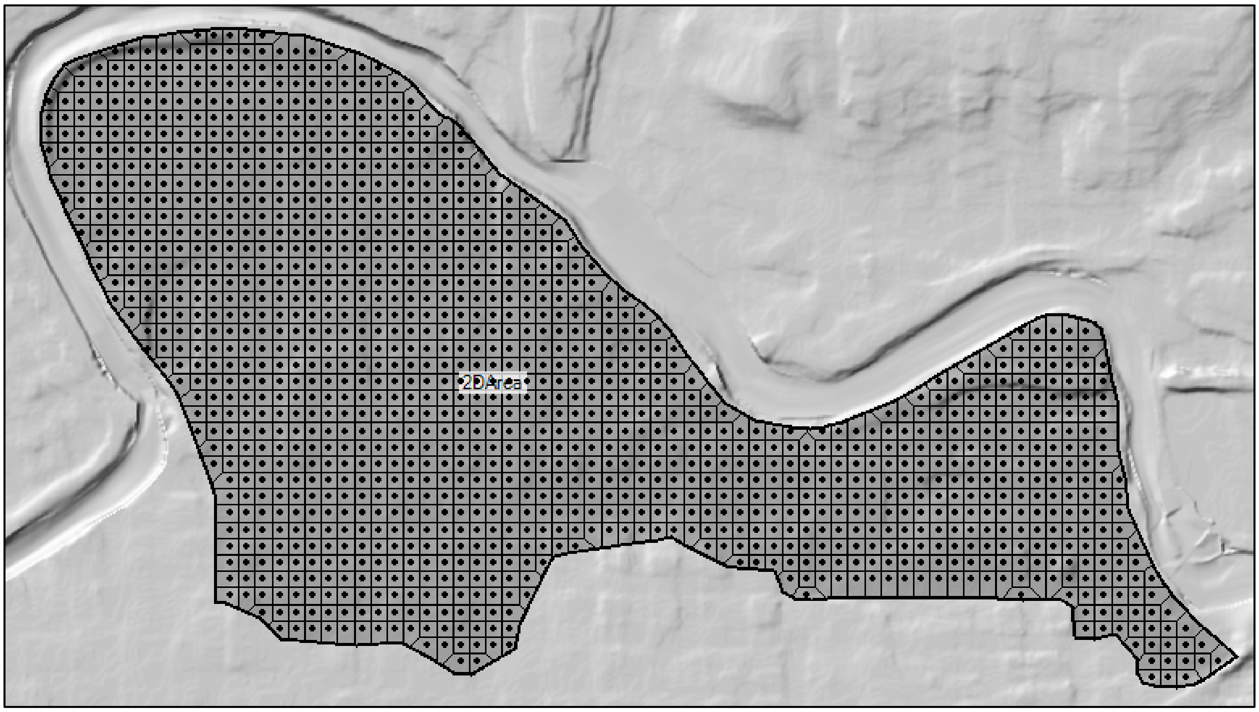


Figure 3-31. Example 2D Mesh with Computation Points.

As points are modified in the **Computation Points** layer, the visible portion of the mesh (with a surrounding buffer) will be recomputed on-the-fly. This will allow the user to see the impacts of modifying a point. An illustration of moving a point and mesh re-computation is shown in Figure 3-32.

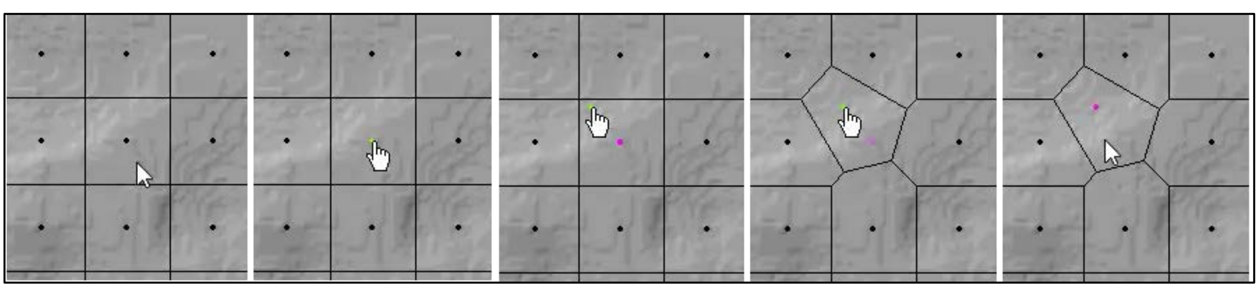


Figure 3-32. The local mesh is recomputed as points are moved in the Computation Points layer.

Break Lines

Breaklines are a hydraulic modeler's best friend. The **Break Lines** layer is a set of polylines used to enforce cell faces along linear features, such as high ground, to direct the movement of water through the 2D domain. Typically, users can start with a coarse 2D mesh and then begin refining and improving the mesh with the use of breaklines.

To add a breakline, draw a polyline where the cell faces should be aligned. When finished, a dialog, shown in Figure 3-33, will appear allowing users to provide a name for the breakline.

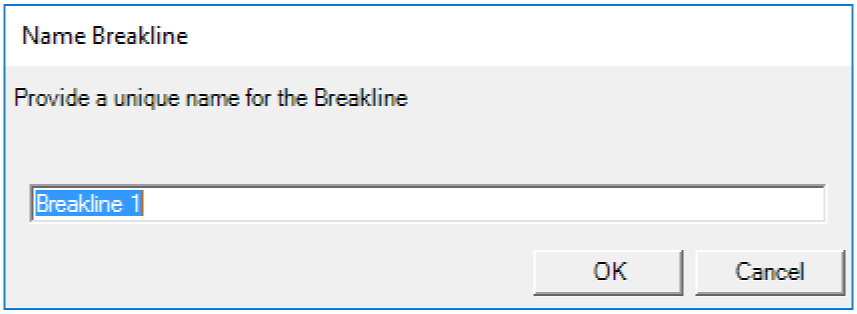


Figure 3-33. Breakline naming dialog.

Also, after finishing a breakline, the local mesh will update to show how cell faces have magnetized to the breakline. To enforce the breakline, right-click on the breakline feature and choose the **Enforce Breakline** menu item. Enforcing a breakline will clear out any cell points on either side of the line based on the 2D Flow Area cell point spacing value (default behavior). New points will then be added on either side of the line to produce cells at (approximately) the default cell point spacing. An example of breakline insertion and enforcement is show in Figure 3-34.

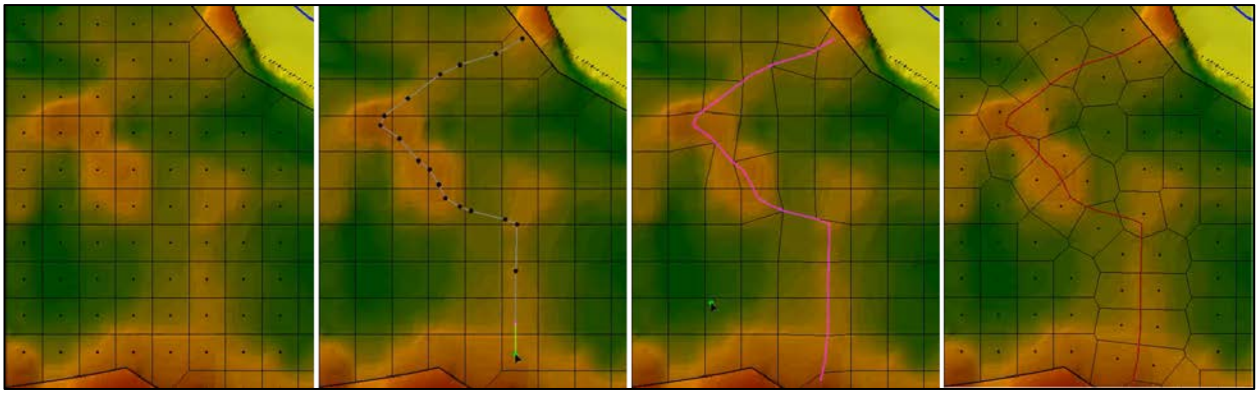


Figure 3-34. Example mesh with steps of breakline creation, face magnetization, and enforcement.

The breakline insertion algorithm is not a simple process. By default, the cell point spacing for the **2D Flow Area** layer is used to determine how often points are inserted. However, the breakline offers a much more robust option for removal of existing points and insertion of new points. The **Breakline Editor**, available by right-click on the feature or the **Break Line** layer and selecting the **Edit Breakline Properties** menu item, allows the user to modify the properties of each breakline.

As shown in Figure 3-35, the properties for each breakline include the Name, Near Spacing, Far Spacing, Near Repeats and Enforce 1 Cell Protection Radius. These properties (other than the Name) control how the breakline will be used to modify the mesh, when enforced.

Name – each breakline must have a unique name in the editor. **Breakline Editor** tools are: Add New, Rename, Delete and Import from a shapefile.

Near Spacing – The distance to add computation points along the breakline (i.e., how often are point added). The points are added next to the line at ½ the near spacing value. If not specified, the default value is the point spacing on the 2D Flow Area.

Far Spacing – How large a distance to go, when adding points, away from the line. Computation points will be added sequentially starting with the Near Spacing and transition (doubling the previous spacing) out until the Far Spacing values is achieved (approximately). If not specified, the default value is the point spacing on the 2D Flow Area.

Near Repeats – The number of times the Near Spacing will be used before the doubling of previous spacing begins. If not specified, the default value is zero. This property is used in cases where the breakline needs to provide detailed spacing along a feature such as a channel or on top of a hydraulic structure.

Enforce 1 Cell Protection Radius – A protective region buffered around the breakline that extends by the Near Spacing distance on each side. Within this protection region, cells can neither be added nor removed by the cell generation routines. This means that any previous hand-edits to those cells will remain, and any nearby breaklines cannot interfere with this already-enforced region.

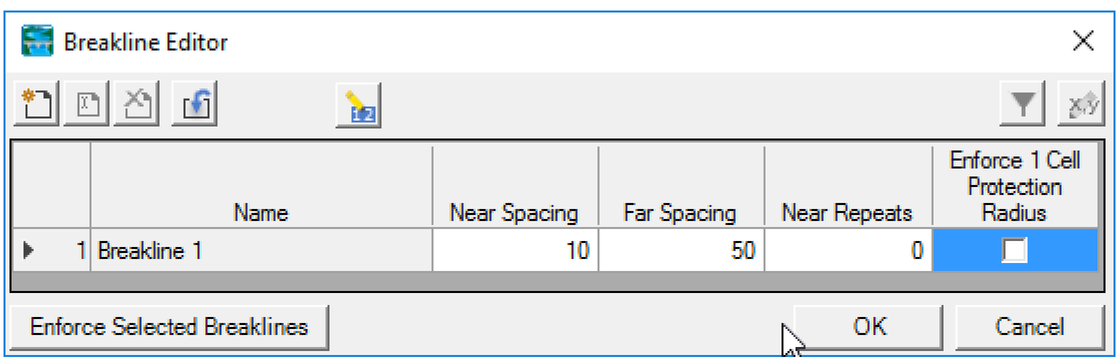


Figure 3-35. Breakline Editor is used to Control how the Breakline will be enforced.

When a breakline is enforced, it is added to the 2D Flow Area by first removing the computation points within a certain buffer region. This buffer region is computed from the Near Spacing, Far Spacing, and Near Repeats to make room for points that will be added as the spacing transitions from the Near Spacing to the Far Spacing values. As points are added back into the mesh along the breakline, the points are added such that the point spacing will double the cell size for each set of new cells that are added away from the breakline. An example of a breakline based on the data entered in Figure 3-35 is shown in Figure 3-36.

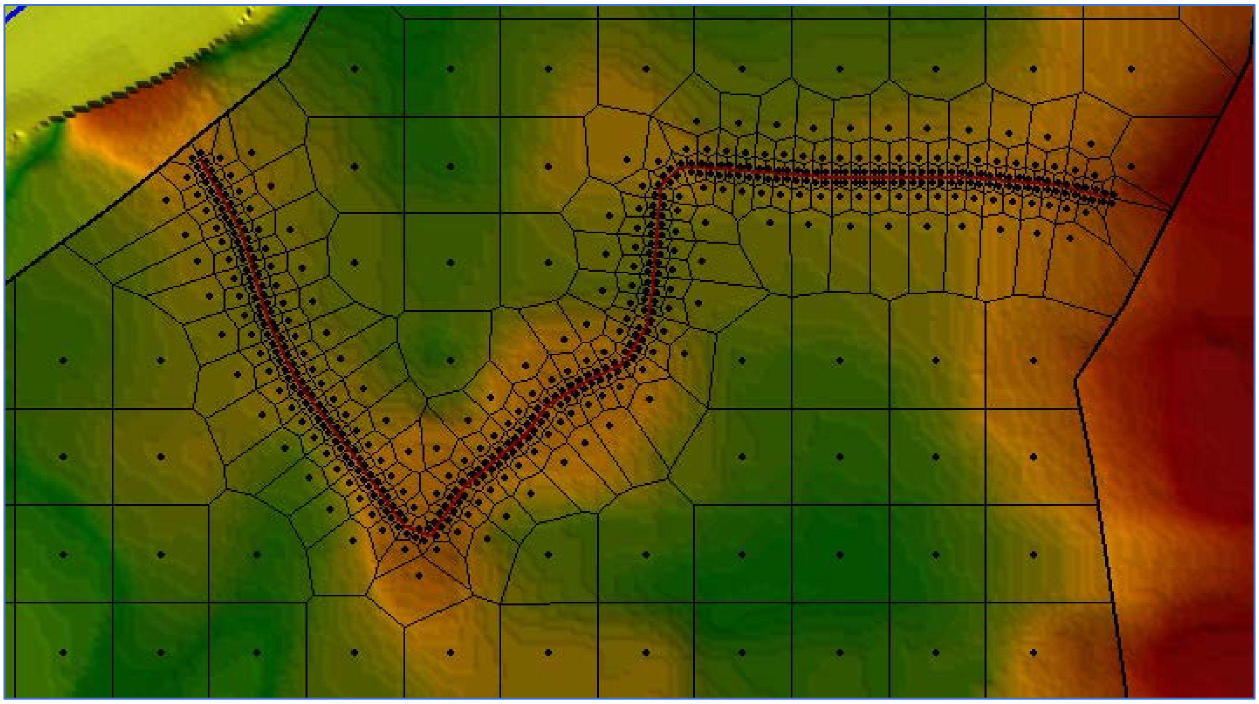


Figure 3-36. Example Breakline Enforcement – used to define Cell Faces along High Ground.

Refinement Regions

Refinement regions are used to change the cell point spacing for specific regions within a 2D Flow Area. A polygon is created to define the boundary of the refinement area. The interior of the area is given a cell spacing (just like the Perimeter layer) and the bounding polyline is given a point spacing (just like a breakline). Refinement regions can be used to densify an area where more detailed results are desired due to rapid changes in terrain or water surface elevation, or to simplify an area where the water surface elevation will not vary much and users want to reduce the number of computation points in the 2D Flow Area. After creating a refinement region polygon, users will be asked to name the feature, as shown in Figure 3-37.

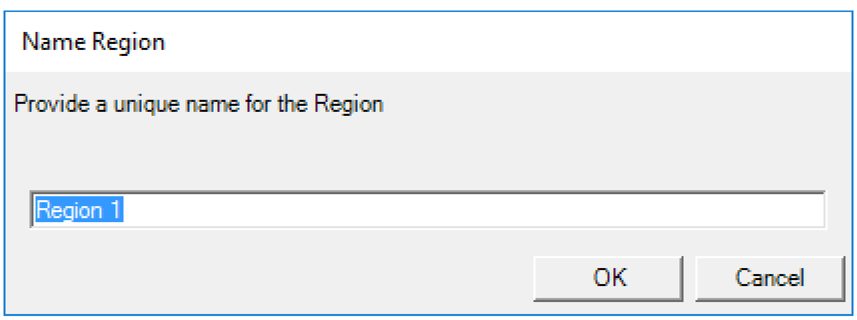


Figure 3-37. Naming dialog for a refinement region.

Additional properties for controlling how the regions affects the 2D Flow Area are accessed through the **Refinement Region Editor**. This editor is available by right-clicking on the feature or the **Refinement Region** layer and selecting the **Edit Refinement Region Properties** shortcut menu item. The **Refinement Region Editor** allows users to modify the properties of each region.

When a refinement polygon is enforced, the boundary is treated much like a breakline where the point spacing along the breakline grows larger farther away from the line. This transition of cell sizes happens both outside of the boundary and on the inside of the polygon. As shown in Figure 3-37, the properties for each refinement area include the Name, Cell Spacing X, Cell Spacing Y, Near Spacing, Far Spacing, and Near Repeats. These properties (described below) control how the refinement region will be used to modify the mesh, when enforced (e.g., Figure 3-39).

Name – Each region must have a unique name and the editor.

Cell Spacing X – The spacing distance in the X-direction for adding computation points inside of the refinement region.

Cell Spacing Y – [NOT IMPLEMENTED YET] The spacing distance in the Y-direction for adding computation points inside of the refinement region.

Perimeter Spacing – The distance to add computation points along the region boundary (i.e., how often points are added) just as done with the Near Spacing on the Breakline layer. The points are generally placed along the line offset by ½ of the spacing value. If not specified, the default value is the Cell Spacing X value.

Far Spacing – How large a distance to go when adding points away from the line. Computation points will be added sequentially starting with the Near Spacing and doubling the previous spacing until the Far Spacing values is achieved (approximately). If not specified, the default value is the point spacing on the 2D Flow Area.

Near Repeats – The number of times to duplicate the Perimeter Spacing on both sides of the perimeter before transitioning to the Far Spacing. If not specified, the default value is zero.

Enforce 1 Cell Protection Radius – A protective region buffered around the perimeter that extends by the Perimeter Spacing distance on each side. Within this protection region, cells can neither be added nor removed by the cell generation routines. This means that any previous hand-edits to those cells will remain, and any nearby breaklines cannot interfere with this already-enforced region.

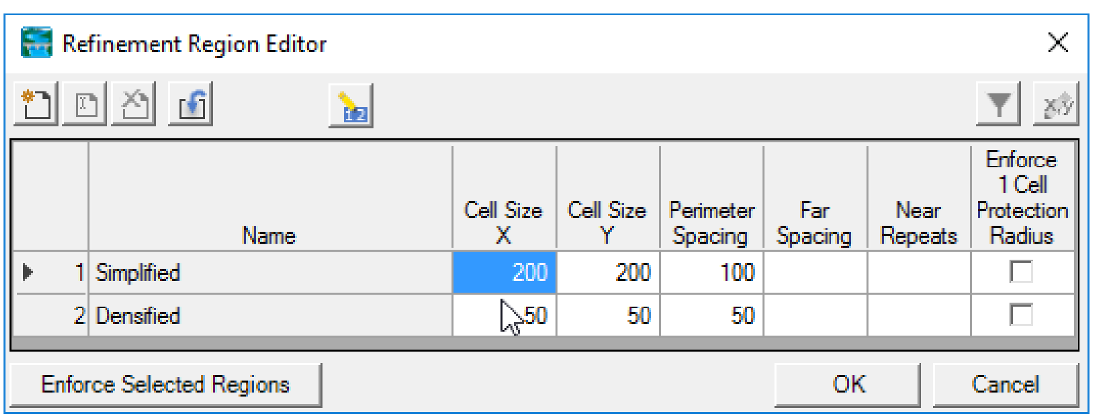


Figure 3-38. Refinement Region Editor.

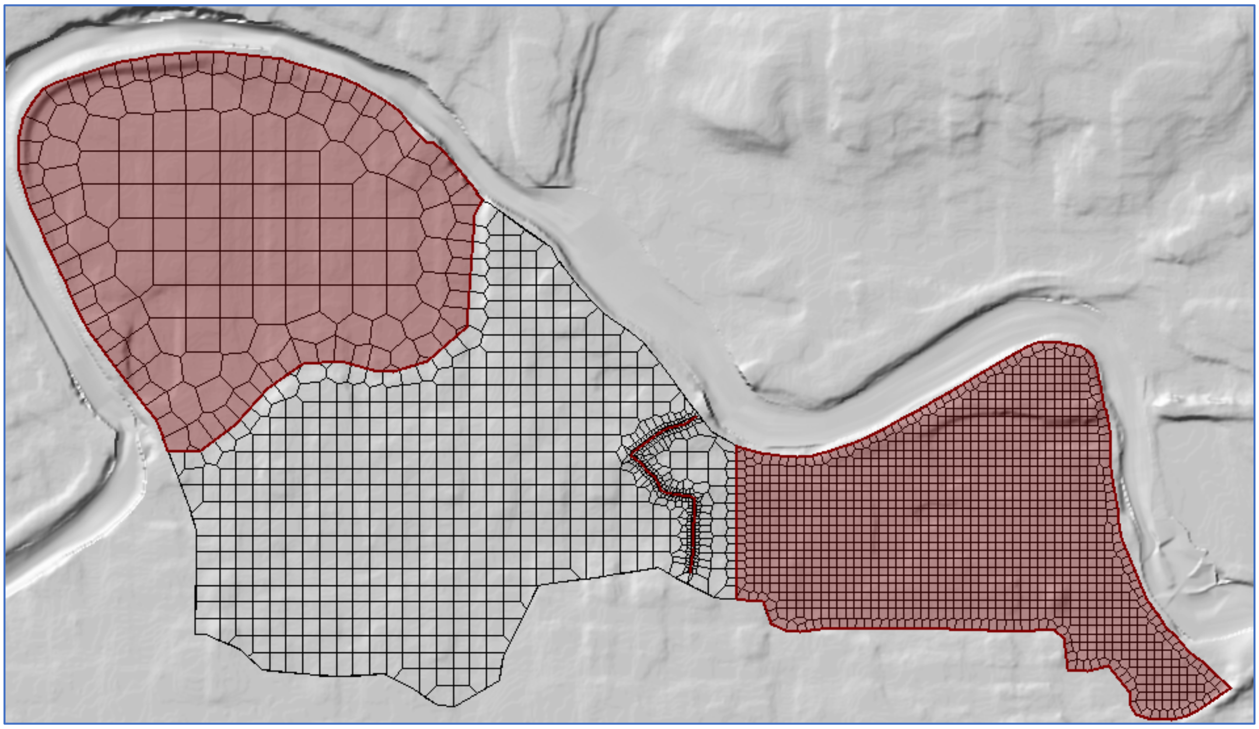


Figure 3-39. Example 2D Area with Refinement Regions – to simplify a portion of the Mesh and densify another portion of the Mesh.

Another use for the refinement region is around the main channel of a stream. Shown in Figure 3-40 is an example where a single refinement region was created for the entire main channel. By doing this the user can control the cell size inside of the channel and ensure that cell faces are aligned with the high ground at the main channel banks. This approach ensure that flow does not spill out of the channel until the water is high enough to cross over the outer cell faces representing the high ground of the main channel bank lines.

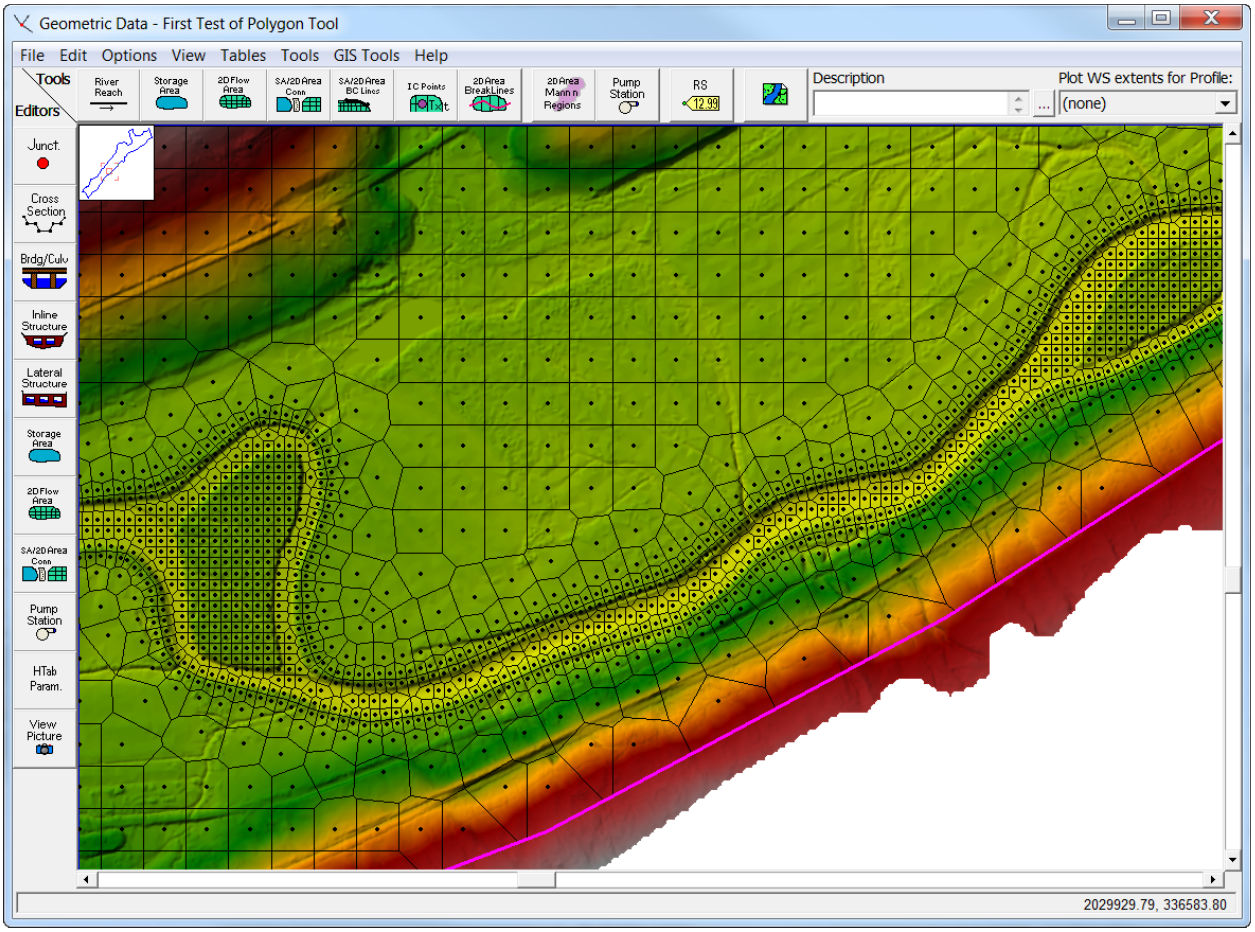


Figure 3-40. Example Refinement Region for Main Channel.

3.2.4 Manning's *n* Value Data

The creation of a Manning's *n* layer has been made much easier and more flexible. A Land Cover layer is no longer required to make a Manning's *n* layer. Users can now create their own polygon Manning's *n* shapefile in RAS Mapper. A Manning's *n* layer can be still be made from any combination of polygon layers and raster layers. So users can still use land cover data sets or they can create their own Manning's n shapefile layers. Also, Manning's *n* layers can be made from any combination of shapefile and raster layers (i.e., land cover, Manning's *n* shapefile, etc.).

Each 2D Flow Area has a base Manning's *n* value that can be specified on the **2D Flow Area Editor**; however, to provide spatially distributed roughness values the user must have the **2D Flow Area** (Geometry Data) layer associated with a Manning's *n* layer. If an existing dataset is available then users can add it to RAS Mapper by right-clicking on the **Map Layers** group and clicking **Add New Manning's n Layer** from the shortcut submenu to import the dataset. The Manning's *n* Value Layer is imported (from the **Manning's n Value Layer** import dialog as displayed in Figure 3-41) by clicking the **+** button to **Browse for Land Cover Files** (LC Data as ***.tif**, ***.img**, or ***.shp** files). The new layer must be associated to the land cover descriptions with

Manning's *n* values and imported to the grid format HEC-RAS uses. If a Manning's *n* layer dataset is not available yet, then the user can create one in RAS Mapper using the editing tools and then convert it to the gridded format that HEC-RAS uses.

To create a Manning n value dataset from the Layers Window, right-click on the **Map Layers** group and click **Map Data Layers | Create new RAS Layer | Manning's n Polygon Layer** from the shortcut submenu. Provide a name for the dataset in the dialog provided (Figure 3-41). The new layer (e.g., *Base n Values*) will be added to the Layers Window in the Map Layers group in edit mode. Select the created layer and start creating polygons.

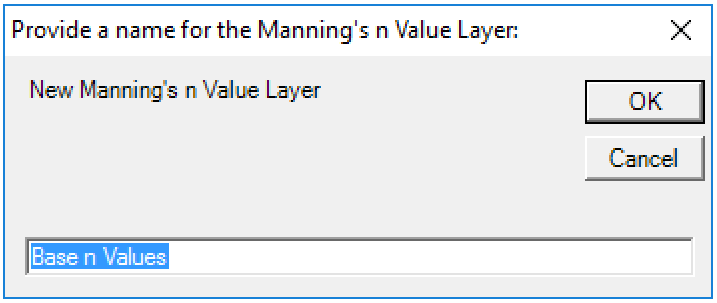


Figure 3-41. Provide a Name for the Created Manning's n Value Layer.

After creating a polygon for a specific area, a **Manning n Value** dialog will prompt the user to provide a description (name) for the area and a Manning's *n* value, as shown in Figure 3-42. A drop down list will contain the list of previously used names and associated Manning's *n* value. By default, the dialog will come populated with the last land cover description used (so create polygons of the same type sequentially) and the user can just click **OK** to accept the name and values in the dialog. As base *n* value polygons are created, they will be labeled with the description (name field in the table) in the Display Window. If polygons intersect, use the clipping tools to remove the overlapping portions (accessed by right-clicking the polygon and clicking the clipping option from the shortcut menu).

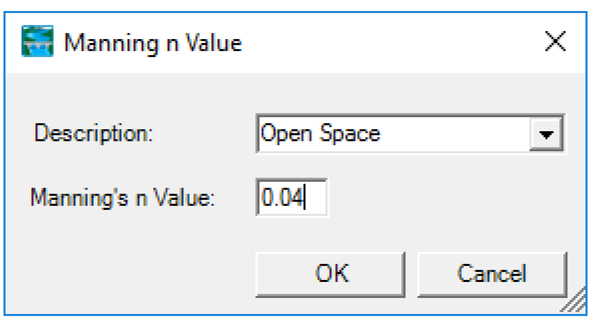


Figure 3-42. Enter a Description and n Value for Each Polygon Created in the Manning's n Value Layer.

When finished editing, a shapefile will be created and the **Manning's n Value Layer** import dialog (shown in Figure 3-43) will automatically appear, allowing the user to import the data to a base Manning's n value raster. (At this point, additional layers can be added to the created *n* value polygon.) This will be the dataset used by RAS Mapper for extracting n values when computing the Hydraulic Properties for each 2D cell face. The information for the importer will be properly filled out with the "Name" and "Manning N Value" fields selected. Users can also make adjustments to n values or description in the **Manning's n Value Layer** import dialog. Verify the raster cell size is appropriate for your dataset and press the **Create** button to generate the raster grid.

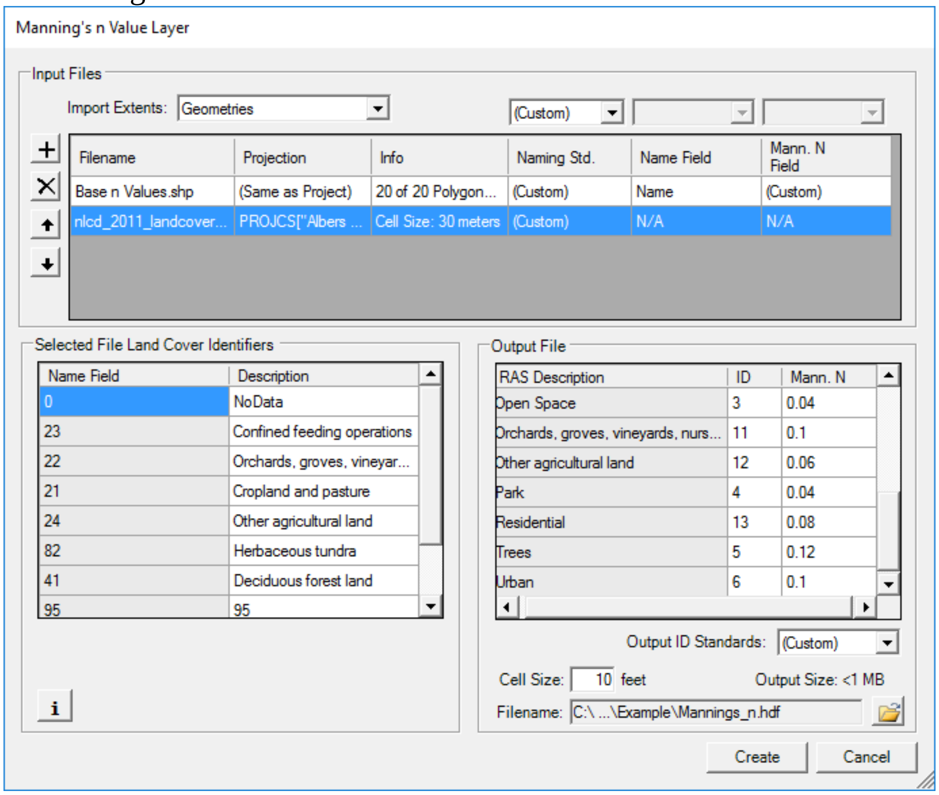


Figure 3-43. Importing a Base Manning's n Value Dataset to a Raster Dataset.

A gridded dataset is created ("*Mannings_n.tif*") with values from 0-255 and an associated lookup table ("*Mannings_n.hdf*") with the grid value, description, and n value. This new **Manning's n** dataset will be loaded to the **Map Layers** group. Next, the user must associate the **Manning's n** raster with the desired geometry by right-clicking the **Geometries** group and clicking **Associate Terrain Layer** from the shortcut menu.

This shortcut option opens the **Manage Geometry Associations** editor (Figure 3-44). In the editor, use the dropdown list to select the **Manning's n** raster to tie the land cover layer to the **Final n values** layer in the Geometry Group. The **Final n values** layer can show the user what values are used when computing the hydraulic properties for 2D flow areas (available by right-clicking the **2D Flow Areas** layer and clicking **Compute 2D Flow Areas Hydraulic Tables** from the shortcut menu). The user can select the **Final n values** layer from the Layers Window and move the cursor over the Display Window area of interest to view the values.

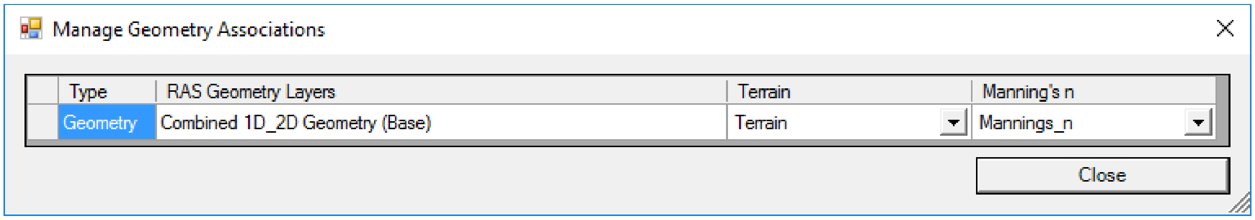


Figure 3-44. Associating a Land Cover layer with Manning's n Values.

Manning's n – Override Regions

The Manning's n – **Override Regions** layer is an editable polygon dataset used to modify Manning's n values in the **Mannings_n** value raster. Use this dataset during model calibration to adjust Manning's n values.

The **Final n Values** layer will reflect the combination of the Mannings_n value raster and the Override Regions polygon values.

Errors

At this time, the **Errors** layer is used predominantly for 1D feature layers. Errors when creating the 2D Mesh will be drawn to screen. Error messages will be drawn to the lower-left of the Display Window (Figure 3-33). The top error message line will be for the "Full" Mesh and the bottom line for the "Local" Mesh. The Full Mesh is the computational mesh for the entire 2D Flow Area. The Local Mesh is the mesh within the Display Window, which is recomputed on a more regular basis (for instance when a computation point or a breakline is added). Computation points for cells that are in error will be colored *red*. An example mesh with two errors is displayed in Figure 3-45 and partially resolved in Figure 3-46.

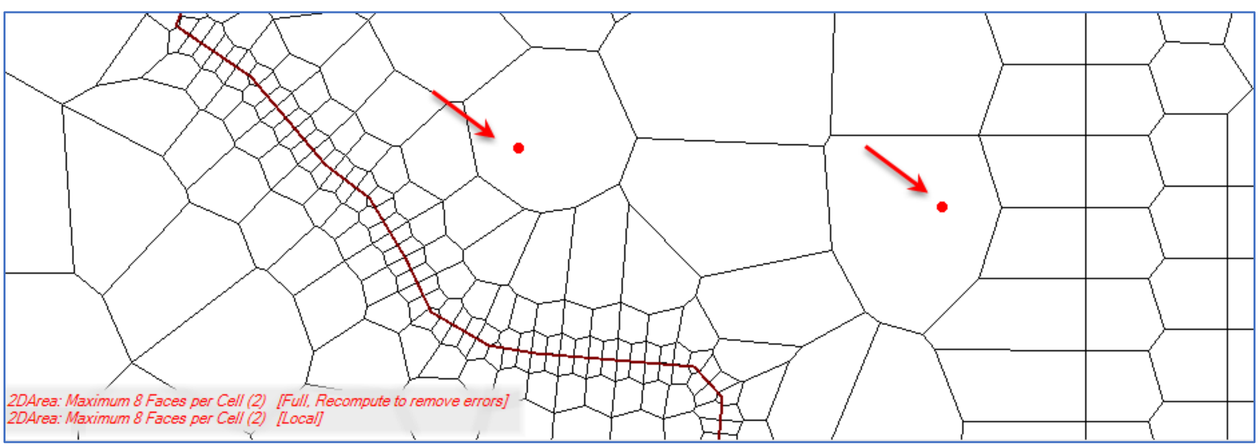


Figure 3-45. An Example Mesh with Two Errors (*red dots*) for Cells which have too Many Faces.

As the mesh is improved (a new cell center was added), the Local Mesh message is updated, as shown Figure 3-46. The Full Mesh status will not be updated until the mesh is recomputed.

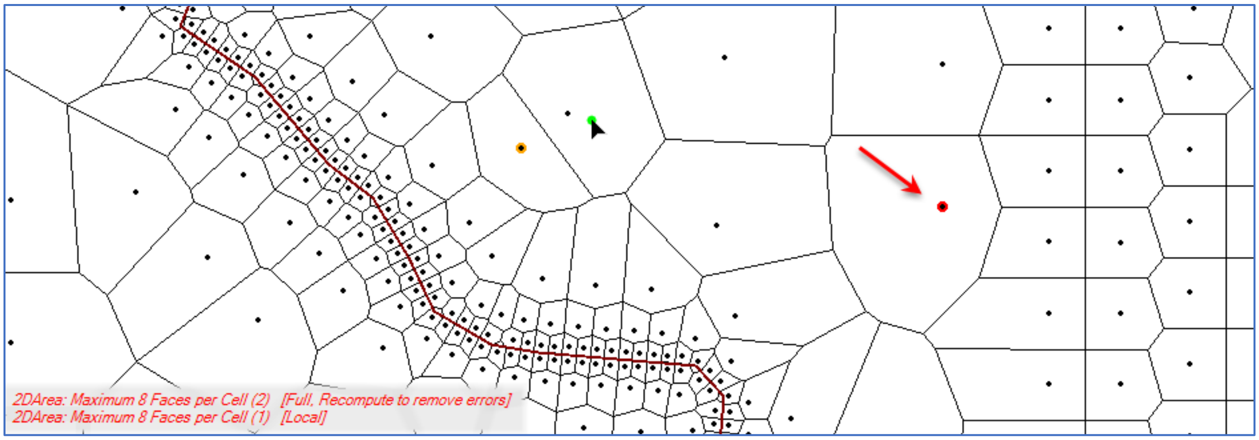


Figure 3-46. An example mesh with one error (a cell which has too many faces). The full mesh still is not updated until it is recomputed by the user.

3.3 Map Results

There are several new Map Types that have been added to HEC-RAS 5.0.4. To create a new Results Map, right-click on the plan name and select **Add New Results Map Layer**. Select the type of map and associated parameters from the map parameters dialog. A summary of the new Map Types available and a description of how they are computed are discussed in Table 3-5.

Table 3-5. Map Types available for RAS Results Maps.

Map Type	Description
Courant	The Courant Number (C_n) is computed for each 2D cell face as C_n = V*DT/DX. The courant number is computed from the face-normal velocity (V), the time step (DT), and the distance from the cell computation points perpendicular to the face (DX).
Froude	The Froude Number (F_r) is computed for each 2D cell face as $F_r = V_{ave} / (g*Y)^{1/2}$. The Froude Number is computed from the average face velocity (V_{ave}), gravitational constant (g), and the hydraulic depth (Y) which is the face area divided by the wetted top width.
Residence Time (2D Only)	Residence time is computed for each 2D cell as $T = Q_{out}$*DT/V. The residence time is computed from the flow out of the cell (Q_{out}), the time step (DT), and the cell volume based on the water surface for the corresponding time step (V).
Wet Cells	The Wet Cells inundation map plots the entire 2D cell wet if the any part of the cell is wet deeper than the specified Threshold depth. This dataset allows the user to identify all 2D cells that have received any water during the specified time step.

3.4 Filter Polygons

The ability to filter a line or polygon is available when editing a feature dataset. This option is also available when exporting a floodplain inundation boundary map. Select the Inundation Boundary map, right-click and select the **Export Layer | Filtered Polygons** menu item. The Export Filtered Polygons dialog (shown in Figure 3-47) has several options for simplifying the polygon dataset: you can remove polygons and filter the polygons in several ways. Part removal is optional.

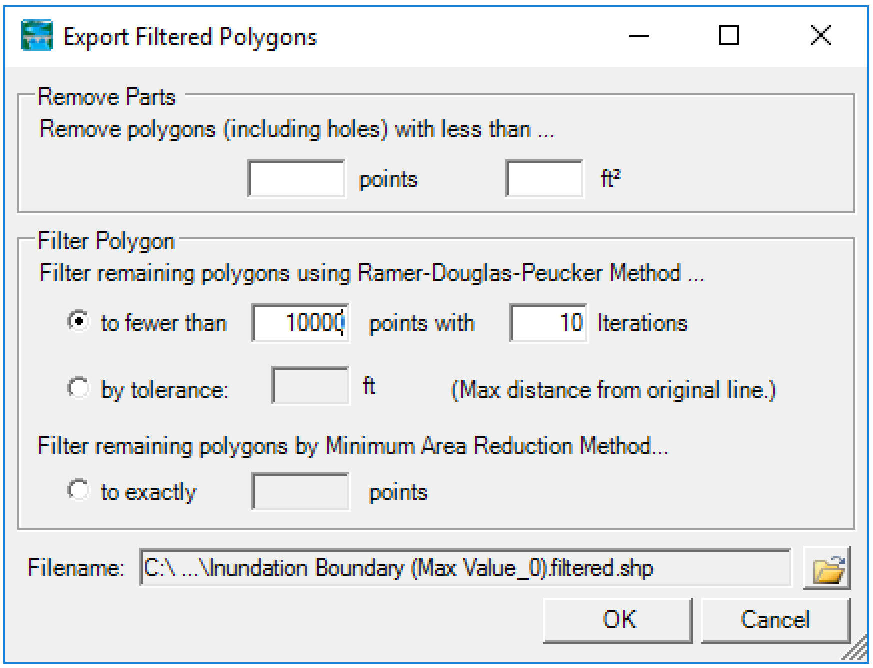


Figure 3-47. Export Filtered Polygon dialog.

The **Remove Parts** option allows you to remove small (and perhaps disconnected) polygons that that you do not wish to show as part of the floodplain or to remove holes in the larger floodplain that you wish to show wet. To collapse smaller holes and remove small polygons you can specify a number of points which each polygon must have to remain in the dataset or specify the smallest allowable area per polygon.

The **Filter Polygon** options allow you choose the method for the point removal using either the "Ramer-Douglas-Peucker Method" (RDP) or the "Minimum Area Reduction Method" (MAR). Each method has specific parameters that must be entered. The RDP Method tends to be faster than the MAR Method. Both methods were designed to work with points, so it is suggested that after filtration the user review the filtered result.

The **RDP Method** can be considered an insertion algorithm. The "line" is considered to be empty (other than the start and end points) and the most important points for describing the line are added one by one until the "new" point for insertion falls within a distance tolerance (effectively not changing the line). If the user chooses the *RDP Method by Tolerance,* the user-specified tolerance is used for considering point insertion. If the user chooses the *RDP Method* using a *specified number of points and iterations*, RAS will select the tolerance that will result in a polygon that has close to, but no larger than, the number of points specified. The final tolerance used is arrived upon through iteration trials up to the number of specified iterations.

The **MAR Method** is a point removal algorithm. The method will reduce the "line" to exactly the number of points specified by evaluating the contribution of each point to the shape of the line segment. Those points that minimally affect the shape when compared with the other points, based on the change in area under the line segment, are removed first. The algorithm will continue removing points until the specified number of points is achieved.

CHAPTER 4

Sediment Transport

This chapter describes the new and updated sediment transport capabilities. Topics include the new approach to calibrate or modify sediment transport functions, greater control over user defined grain classes, a new lateral structure capability, sediment stage boundary condition, and new rules capabilities for sediment. The information contained in this chapter is supplemental to Chapter 17 of the HEC-RAS 5.0 *User's Manual*.

4.1 New Transport Equation and Dialog for Calibration and Modification Options and Editor Dialog

The release of HEC-RAS 5.0.4 Beta includes a new approach to calibrate or modify sediment transport functions which is available from the updated **Transport Function Calibration and Modification** editor (Figure 4-1). Users can access the new approach available in the updated editor from the **Sediment Data** editor (from the **Options | Calibrate Transport Functions** menu option), which is opened from the HEC-RAS main window by clicking the button.

In the previous version of HEC-RAS (Version 5.0.3) the available approach was to define parameters and coefficients in the transport function. In HEC-RAS Version 5.0.4 this approach to hard code the transport functions is still available by selecting the **Parameters and Coefficients** radio button (Figure 4-1). However, when using the HEC-RAS Version 5.0.3 approach some of the definable mobility parameters are actually functions in the transport equations. Overwriting such parameters can potentially remove functionality and sensitivity of the transport function. Even overwriting these parameters with the default parameter values (displayed in Figure 4-1) can lead to situations where results diverge from the un-modified transport function.

Therefore, in HEC-RAS Version 5.0.4 a second approach was added to calibrate sediment transport functions based on two ratios: one for the total computed "transport" capacity and one for the "mobility parameter." This new approach is enabled by selecting the **Scaling Factors** radio button (Figure 4-1). HEC-RAS Version 5.0.4 uses the **Transport Function Scaling Factor** to scale the computed transport capacity, e.g., a factor of 0.9 returns 90% of the computed capacity and 1.1 computes an additional 10%. This calibration is a simple, linear scaling factor which is applied to any transport function selected.

Selecting the **Scaling Factors** option also requires a **Mobility Parameter Scaling Factor** (Figure 4-1). This factor only applies to the four transport functions in the original editor: Ackers-White, Laursen-Copeland, Wilcock-Crowe, and Meyer-Peter Muller (as well as the MPM component of the blended MPM-Toffaleti algorithm). These "excess mobility" equations compute capacity as a function of the difference between a driving force/stress/power and a "mobility factor" (usually raised to a power). The mobility factor quantifies the threshold of motion (the competence threshold). The Mobility Parameter Scaling Factor scales the mobility parameter in each equation (A in AW, τ_{rm} in WC, τ_c in LC and MPM). Therefore, the effects of this ratio are usually non-linear and "inverse", i.e., increasing the mobility parameter increases the transport *threshold* which makes sediment less mobile and decreases transport (e.g., a Mobility Parameter Scaling Factor of 1.1 will show up in MPM as $\mathbf{Gs \sim (\tau^{*} - (1.1)\tau^{*}_{c})^{1.5}}$, decreasing transport).

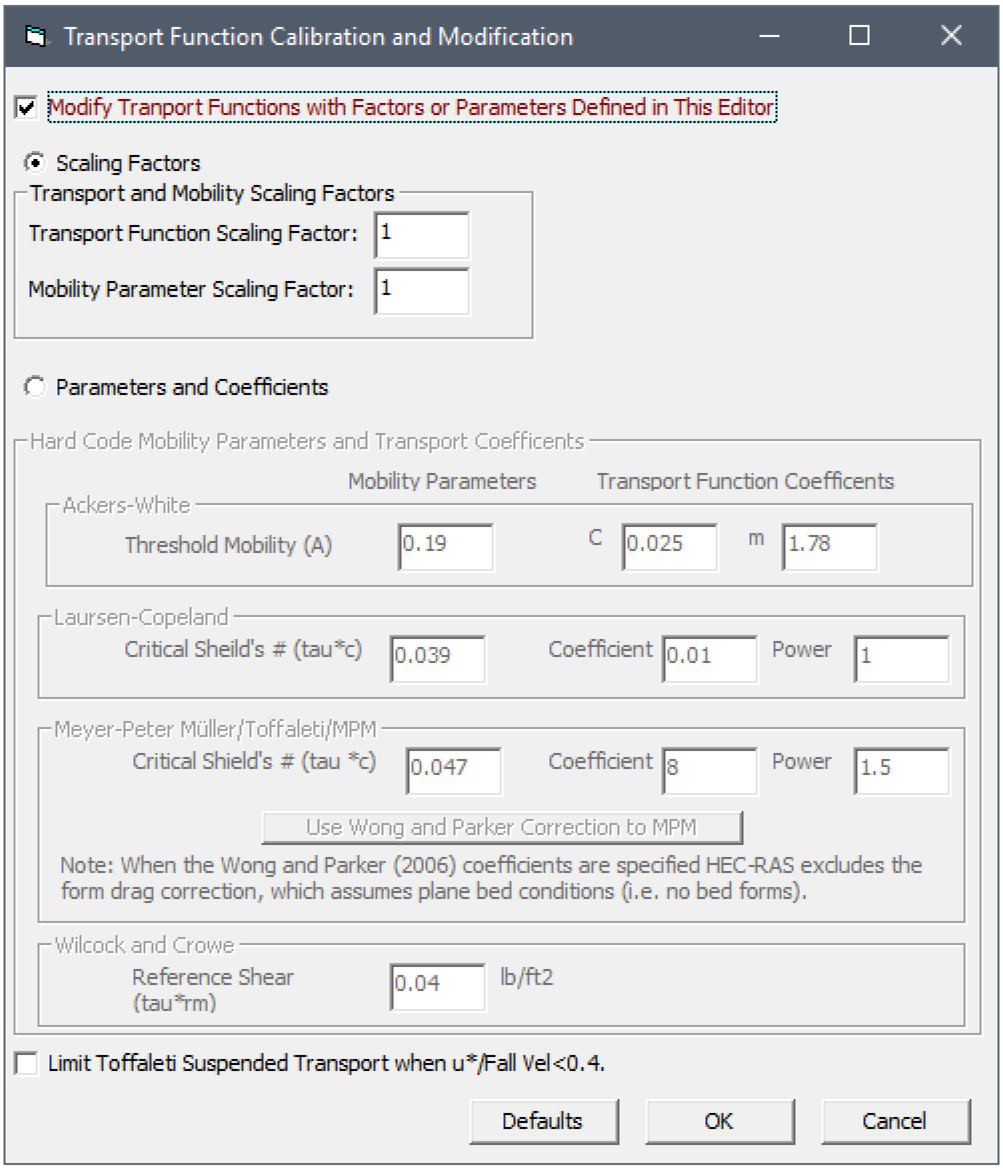
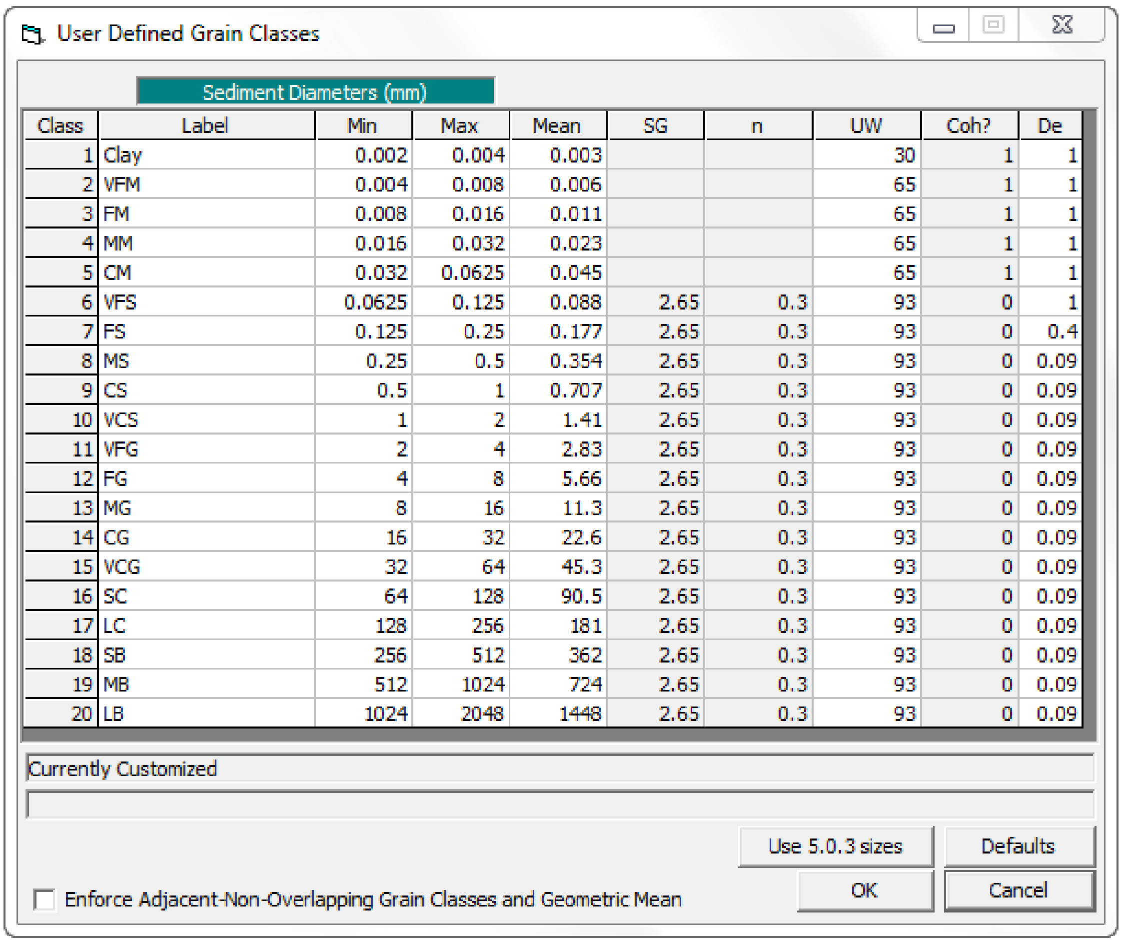


Figure 4-1: New transport equation calibration editor.

Modifying transport functions can be controversial: This is because sediment data are uncertain and noisy. Nevertheless, there is a tradition of selecting an "appropriate" transport function and then adjusting input data (inflowing load and gradations) and/or bed gradations, within the range of variability that reproduces observed bed change, downstream concentration, or other calibration metrics. This traditional selection is a valid calibration approach. However, transport functions are also based on noisy and uncertain data, including data collected in a range of lab and field conditions that may differ from the modeled system. Modelers often have greater confidence in their data than in the transport function, and making reasonable and defensible adjustments to the transport function is preferable in such cases.

4.2 Variable Sediment Properties

The **User Defined Grain Class** editor (Figure 4-2) has been expanded to include other sediment properties, making them editable by grain class. Some of these are disabled (greyed out), and will become available in future versions of HEC-RAS. In HEC-RAS Version 5.0.4, sediment unit weight is editable by grain class. This supersedes the **Sediment Properties** editor that used to define unit weight by combined Clay, Silt, and Sand+ grain classes.



Class	Label	Min	Max	Mean	SG	n	UW	Coh?	De
1	Clay	0.002	0.004	0.003			30	1	1
2	VFM	0.004	0.008	0.006			65	1	1
3	FM	0.008	0.016	0.011			65	1	1
4	MM	0.016	0.032	0.023			65	1	1
5	CM	0.032	0.0625	0.045			65	1	1
6	VFS	0.0625	0.125	0.088	2.65	0.3	93	0	1
7	FS	0.125	0.25	0.177	2.65	0.3	93	0	0.4
8	MS	0.25	0.5	0.354	2.65	0.3	93	0	0.09
9	CS	0.5	1	0.707	2.65	0.3	93	0	0.09
10	VCS	1	2	1.41	2.65	0.3	93	0	0.09
11	VFG	2	4	2.83	2.65	0.3	93	0	0.09
12	FG	4	8	5.66	2.65	0.3	93	0	0.09
13	MG	8	16	11.3	2.65	0.3	93	0	0.09
14	CG	16	32	22.6	2.65	0.3	93	0	0.09
15	VCG	32	64	45.3	2.65	0.3	93	0	0.09
16	SC	64	128	90.5	2.65	0.3	93	0	0.09
17	LC	128	256	181	2.65	0.3	93	0	0.09
18	SB	256	512	362	2.65	0.3	93	0	0.09
19	MB	512	1024	724	2.65	0.3	93	0	0.09
20	LB	1024	2048	1448	2.65	0.3	93	0	0.09

Figure 4-2: User defined sediment properties by grain class.

HEC-RAS Version 5.0.4 includes a modified **User Defined Grain Class** editor to allow users to turn off the "adjacent and non-overlapping" requirements for user defined grain classes. This is useful for users who want gap-graded sediment without including intermediate grain classes, or who want distinct grain classes that are the same size. The **Enforce Adjacent-Non-Overlapping Grain Classes and Geometric Mean** check box makes this feature available. Previous versions of HEC-RAS required that grain classes define all grain sizes without gaps or overlap, and enforced this requirement by setting the downstream limit of each grain class to the upstream limit of the finer class. Note, specifying grain classes that increase in size is still strongly recommended; most of the sorting and armoring routines use the grain class order to determine armoring (based on the percentage of coarser particles) and assume that the grain class size increases.

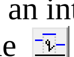
Finally, HEC-RAS Version 5.0.4 converted the sediment write files to HDF5. In this process, the model converts grain size from feet to millimeters. This conversion can cause rounding errors which generate small differences between HEC-RAS Version 5.0.3 and Version 5.0.4 results. The HEC-RAS Version 5.0.4 results are more precise. However, if users want to reconstruct HEC-RAS Version 5.0.3 results, the **User Defined Grain Classes** editor includes a **Use 5.0.3 Sizes** button that updates the representative grain class sizes to reproduce the HEC-RAS Version 5.0.3 rounding error.

4.3 Lateral Weir Sediment Connection to a 2D Area

HEC-RAS Version 5.0.4 can pass sediment from an unsteady, 1D sediment model into a 2D area with a lateral structure. Previous versions of HEC-RAS did not support 1D-2D connections in sediment models. HEC-RAS version 5.0.4 does not compute 2D sediment, so the 2D area simply becomes a sediment sink. The information contained in this section is supplemental to Chapter 17 of the HEC-RAS 5.0 *User's Manual*.

4.4 Quasi-Unsteady Internal Stage Boundary Conditions

This section discusses the HEC-RAS Version 5.0.4 enhancements made to the quasi-unsteady flow analysis, which is only utilized in sediment transport modeling. Information provided in this section is supplemental to Chapter 17 in the HEC-RAS 5.0 *User's Manual*. In general, quasi-unsteady flow assumptions can pose problems in reservoir simulations. If a quasi-unsteady HEC-RAS model simulates a dam as an internal inline structure, the quasi-steady flow analysis will simply compute a backwater through the gates to compute a reservoir stage. This approach does not conserve volume and can compute highly variable reservoir stages that do not reproduce historical or projected conditions (see Gibson *et al* (2017) for an extended discussion). Modelers generally work around this limitation by either modeling reservoirs in unsteady sediment or making the reservoir pool the downstream boundary condition in a quasi-unsteady model. In previous versions of HEC-RAS it was difficult to model the reservoir pool and downstream reach or a multi-reservoir cascade in quasi-unsteady flow.

HEC-RAS Version 5.0.4 addresses this quasi-unsteady model limitation by including an internal stage boundary condition in the **Quasi Unsteady Flow Editor** (opened by clicking the  button, from the HEC-RAS main window). This new feature is available from the **Quasi Unsteady Flow Editor** by clicking the **Internal Stage BC** button (*blue box* in Figure 4-3), which opens the **Internal Stage Boundary Condition** dialog (e.g., Figure 4-4).

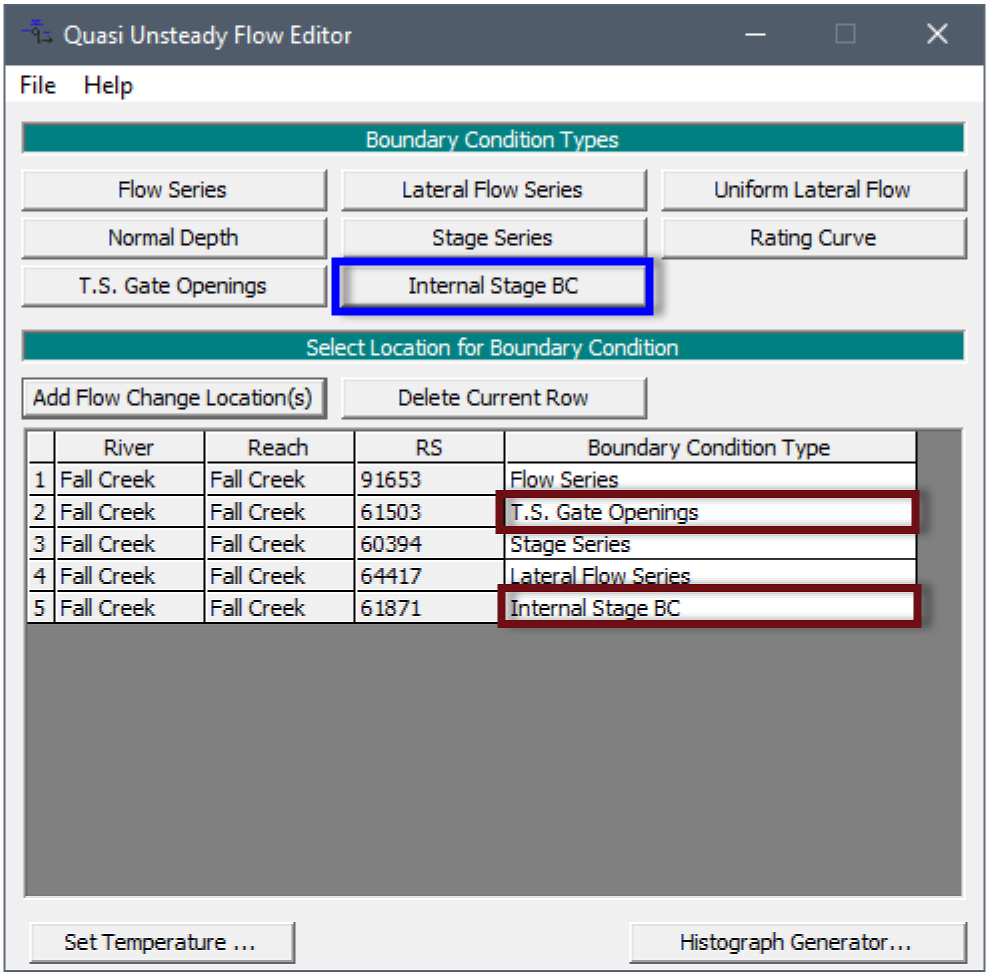


Figure 4-3. New Internal Stage Boundary Condition Option (*blue box*) in Quasi-Unsteady Flow.

From the **Internal Stage Boundary Condition** dialog (Figure 4-4) users can specify a time series of fixed stages at any cross section in an HEC-RAS model. This feature is generally applied at the cross section upstream of an inline structure, to override the backwater stage through the dam and set the reservoir stage to historic levels or to levels computed with a hydrologic computation that conserves mass.

The inline structure still requires a time series of gate openings (**T.S. Gate Openings** *red box* in Figure 4-3) but if an **Internal Stage BC** is entered then the gate openings data is overridden by the internal stage boundary condition (BC). If users define a partial internal stage BC time series, then HEC-RAS will use the specified stages when one exists, and will switch to the backwater though the specified gate openings if the **Internal Stage BC** is blank (which can be useful for dam removal and reservoir flushing simulations).

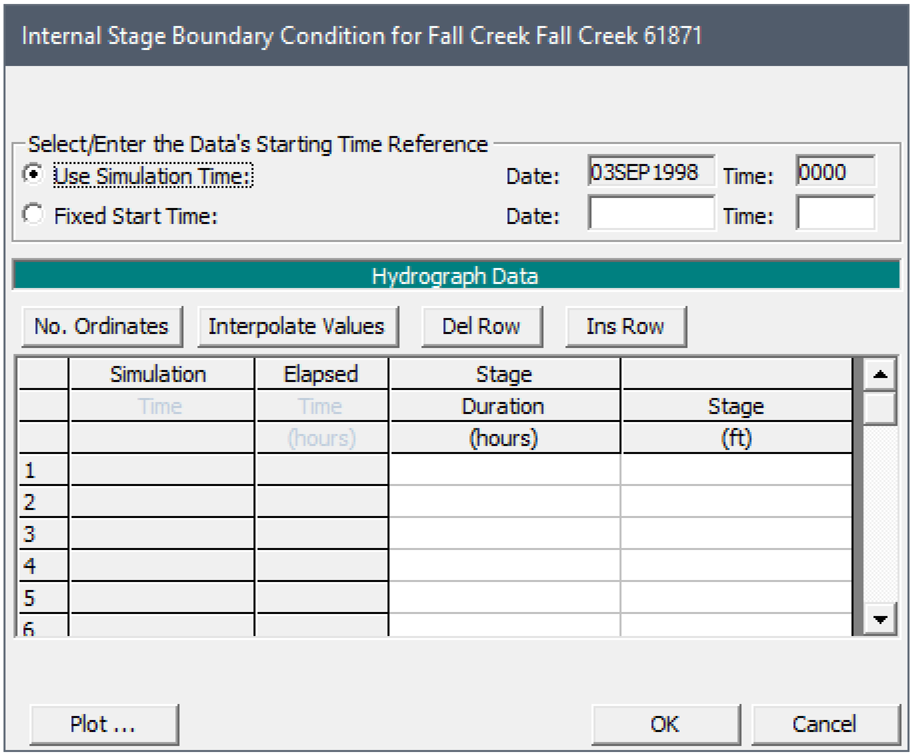


Figure 4-4. Defining an Internal Stage Boundary Condition.

4.5 Sediment Parameters in the HEC-RAS Rules Editor

HEC-RAS Version 5.0.4 now includes sediment parameters for unsteady flow in the **Operational Rules** editor (Figure 4-5). The **Operational Rules** editor is accessed from the **Unsteady Flow Data** editor (opened from the HEC-RAS main window by clicking the  button) by clicking the **Rules** button located in the "Boundary Condition Types" panel (e.g., Figure 2-9).

This new ability means that an unsteady HEC-RAS sediment model can operate structures based on bed change at a cross section or sediment concentration (e.g., downstream total maximum daily loads or TMDLs). Gibson and Boyd (2016a,b) document an example of this new feature, using sediment rules to simulate a reservoir flush on the Niobrara River (as displayed in Figure 4-5).

The HEC-RAS Version 5.0.4 (and previous versions) provides users with example projects (from the HEC-RAS main window's **Help | Install Example Projects** menu option) and the unsteady sediment example project, "***SedRuleLat.prj***" in the ***Unsteady Sediment with Concentration Rules*** installed data folder, includes a simple example of sediment concentration rules.

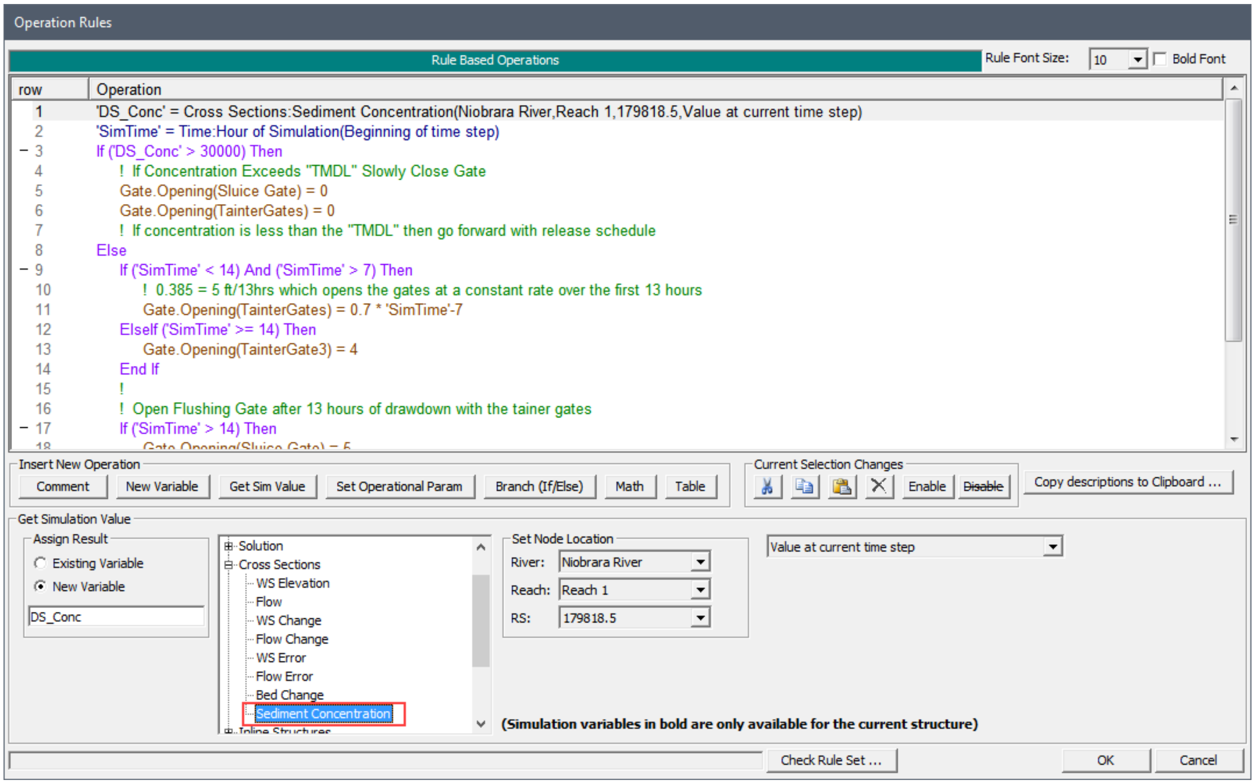


Figure 4-5. Gibson and Boyd (2016a,b) Example – Sediment Parameters (*red box*) in the HEC-RAS Operational Rules Editor.

CHAPTER 5

References

Gibson, S., A. Sánchez, S. Piper, and G. Brunner, (2017) “New One-Dimensional Sediment Features in HEC-RAS 5.0 and 5.1,” Proceedings, ASCE EWRI World Environmental & Water Resource Congress.

Gibson, S. and Boyd, P. (2016a) “Monitoring, Measuring, and Modeling a Reservoir Flush on the Niobrara River in the Sandhills of Nebraska,” Proceedings, River Flow 2016, ed Constantinescu *et al*., 1448-1455.

Gibson, S. and Boyd, P. (2016b) “Designing Reservoir Sediment Management Alternatives with Automated Concentration Constraints in a 1D Sediment Model,” Proceedings International Symposium on River Sedimentation.

Hydrologic Engineering Center, February 2016. HEC-RAS, *River Analysis System*, User's Manual, CPD-68, U.S. Army Corps of Engineers, Davis, CA.
http://www.hec.usace.army.mil/software/hec-ras/documentation/HEC-RAS%205.0%20Users%20Manual.pdf

Hydrologic Engineering Center, February 2016. HEC-RAS, *River Analysis System*, 2D Modeling User's Manual, CPD-68a, U.S. Army Corps of Engineers, Davis, CA.
http://www.hec.usace.army.mil/software/hec-ras/documentation/HEC-RAS%205.0%202D%20Modeling%20Users%20Manual.pdf