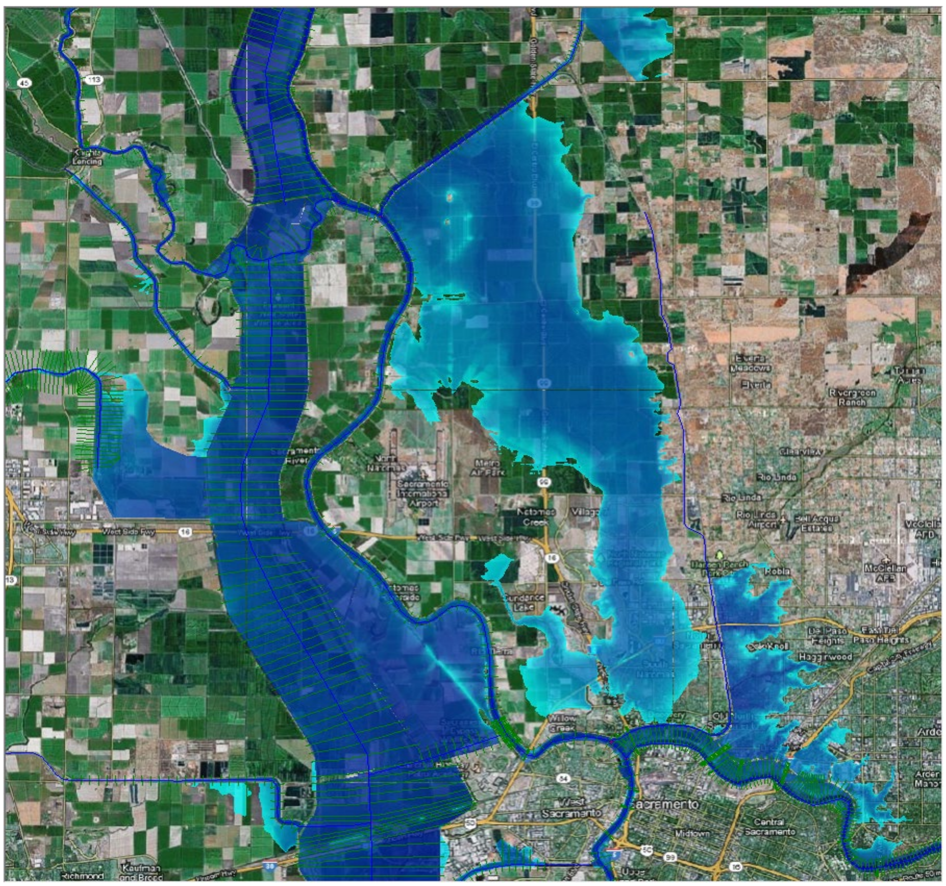
US Army Corps
of Engineers
Hydrologic Engineering Center

HEC-RAS
River Analysis System



2D Modeling User's Manual

Version 6.0 Beta
December 2020

Approved for Public Release. Distribution Unlimited.

CPD-68A

REPORT DOCUMENTATION PAGE		*Form Approved OMB No. 0704-0188*

1. REPORT DATE *(DD-MM-YYYY)*	**2. REPORT TYPE**	**3. DATES COVERED** *(From - To)*
December 2020	Computer Program Documentation	

4. TITLE AND SUBTITLE	**5a. CONTRACT NUMBER**
HEC-RAS River Analysis System 2D Modeling User's Manual Version 6.0 Beta	**5b. GRANT NUMBER**
	5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S)	**5d. PROJECT NUMBER**
Gary W. Brunner, CEIWR-HEC	**5e. TASK NUMBER**
	5F. WORK UNIT NUMBER

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)	**8. PERFORMING ORGANIZATION REPORT NUMBER**
US Army Corps of Engineers Institute for Water Resources Hydrologic Engineering Center (HEC) 609 Second Street Davis, CA 95616-4687	CPD-68A
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 and 1D and two-dimensional (2D) unsteady flow river hydraulics calculations. HEC-RAS is an integrated system of software. The system is comprised of a graphical user interface (GUI), separate hydraulic analysis components, data storage and management capabilities, graphics, mapping (HEC-RAS Mapper) and reporting facilities.

The HEC-RAS system contains four hydraulic analysis components for: (1) steady flow water surface profile computations; (2) 1D and 2D unsteady flow simulations; (3) movable boundary sediment transport computations (cohesive and non-cohesive sediments); 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 software also contains tools for performing inundation mapping directly inside the software.

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**			
	U	U	UU	283	**19b. TELEPHONE NUMBER**

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

HEC-RAS
River Analysis System

2D Modeling

User's Manual

Version 6.0 Beta
December 2020

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-68A

River Analysis System, HEC-RAS

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

Terms and Conditions of Use of HEC-RAS:

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 the HEC Meteorological Visualization Utility Engine (HEC-RAS) "the Software" (whether 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-MEVUE \"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-RAS 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-RAS 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, delete all copies, and cease using the program.

Table of Contents

Foreword

This manual was written by Mr. Gary W. Brunner.

The U.S. Army Corps of Engineers' River Analysis System (HEC-RAS) is software that allows you to perform one-dimensional steady flow hydraulics; one and two-dimensional unsteady flow river hydraulics calculations; quasi Unsteady and full unsteady flow sediment transport-mobile bed modeling; water temperature analysis; and generalized water quality modeling (nutrient fate and transport).

The first version of HEC-RAS (version 1.0) was released in July of 1995. Since that time there have been several major releases of this software package, including versions: 1.1; 1.2; 2.0; 2.1; 2.2; 3.0, 3.1, 3.1.1, 3.1.2, 3.1.3, 4.0, 4.1, 5.0 and now version 6.0 in 2020.

The HEC-RAS software was developed at the Hydrologic Engineering Center (HEC), which is a division of the Institute for Water Resources (IWR), U.S. Army Corps of Engineers.

The software was designed by Mr. Gary W. Brunner, leader of the HEC-RAS development team. The user interface and graphics were programmed by Mr. Mark R. Jensen, Alex Kennedy, Anton Rotter-Sieren, Cameron Ackerman, and Stanford Gibson. The steady flow water surface profiles computational module and the majority of the one-dimensional unsteady flow computations modules was programmed by Mr. Steven S. Piper. The One-dimensional unsteady flow matrix solution algorithm was developed by Dr. Robert L. Barkau (Author of UNET and HEC-UNET).

The two-dimensional unsteady flow modeling capabilities were developed by Gary W. Brunner, Ben Chacon (Resource Management Consultants, RMA), Steve S. Piper, Mark R. Jensen, Alex J. Kennedy, and Alex Sanchez.

The sediment transport interface module was programmed by Mr. Stanford A. Gibson. The quasi unsteady flow computational sediment transport capabilities were developed by Stanford A. Gibson and Steven S. Piper. The Unsteady flow sediment transport modules were developed by Stanford A. Gibson, Steven S. Piper, and Ben Chacon (RMA). Special thanks to Mr. Tony Thomas (Author of HEC-6 and HEC-6T) for his assistance in developing the quasi-unsteady flow sediment transport routines used in HEC-RAS. The two-dimensional sediment transport modules were developed by Alex Sanchez and Stanford Gibson. The Debris flow capabilities in HEC-RAS (1D and 2D) were developed by Stanford Gibson and Alex Sanchez. Most of the sediment output was designed by Stanford Gibson and Alex Sanchez and programmed by Anton Rotter-Sieren.

The new 2D plotting library and plots (Breach Plot, Hydrographs, and DSS viewer) were developed by Mark R. Jensen, Anton Rotter-Sieren, and Ryan Miles (RMA).

The new 3D visualization tool was developed by Anton Rotter-Sieren and Alex Kennedy.

The water quality computational modules were designed and developed by Mr. Mark R. Jensen, Dr. Cindy Lowney and Zhonglong Zhang (ERDC-RDE-EL-MS).

The spatial data and mapping tools (RAS Mapper) were developed by Mark R. Jensen, Cameron T. Ackerman, Alex J. Kennedy, and Anton Rotter-Sieren. Special thanks to Mr. Will Breikreutz for his assistance in developing the RAS Tile server.

The interface for channel design/modifications was designed and developed by Mr. Cameron T. Ackerman and Mr. Mark R. Jensen. The stable channel design functions were programmed by Mr. Chris R. Goodell.

The routines that import HEC-2 and UNET data were developed by Ms. Joan Klipsch. The routines for modeling ice cover and wide river ice jams were developed by Mr. Steven F. Daly of the Cold Regions Research and Engineering Laboratory (CRREL).

Many other HEC staff members have made contributions in the development of this software, including: Vern R. Bonner, Richard Hayes, John Peters, Al Montalvo, and Michael Gee. Mr. Matt Fleming was the Chief of the H&H Division, and Mr. Chris Dunn was the director during the development of this version of the software.

HEC-RAS uses the following third party libraries:

1. Hierarchical Data Format (HDF) – HEC-RAS uses the HDF5 libraries in both the User Interface and the Computational engines for writing and reading data to binary files that follow the HDF5 standards. The HDF Group: http://www.hdfgroup.org/HDF5/
2. Geospatial Data Abstraction Library (GDAL) – HEC-RAS uses the GDAL libraries in the HEC-RAS Mapper tool. These libraries are used for all Geospatial data rendering, coordinate transformations, etc… GDAL: http://www.gdal.org/
3. Bitmiracle LibTiff .Net. LibTiff.Net provides support for the Tag Image File Format (TIFF), a widely used format for storing image data. Bitmiricle: http://bitmiracle.com/libtiff/
4. Oxyplot – 2 dimensional X-Y plots in HEC-RAS Mapper. Oxyplot: http://oxyplot.org/
5. SciChart – 2 dimensional X-Y plots. http://www.scichart.com
6. SQLite – Reading and writing database files. SQLite: https://www.sqlite.org/
7. cURL - HTTP support for GDAL http://curl.haxx.se/
8. Clipper – an open source freeware library for clipping and offsetting lines and polygons. http://www.angusj.com/delphi/clipper.php
9. SharpDX – .Net DirectX graphics library for 2D and 3D support. https://www.sharpdx.org
10. ZeroFormatter – Use to read and write HEC-RAS objects to disk very quickly. https://github.com/neuecc/ZeroFormatter

CHAPTER 1

Introduction

HEC-RAS can perform two-dimensional (2D) hydrodynamic routing within the unsteady flow analysis portion of the software. Users can now perform one-dimensional (1D) unsteady-flow modeling, two-dimensional (2D) unsteady-flow modeling (Shallow Water equations (SWE) or Diffusion Wave equations (DWE)), as well as combined 1D and 2D unsteady-flow routing. The 2D flow areas in HEC-RAS can be used in number of ways. The following are some examples of how the 2D flow areas can be used to support modeling with HEC-RAS:

- Detailed 2D channel modeling
- Detailed 2D channel and floodplain modeling
- Combined 1D channels with 2D floodplain areas
- Combined 1D channels/floodplains with 2D flow areas behind levees
- Directly connect 1D reaches into and out of 2D flow areas
- Directly connect a 2D flow area to 1D Storage Area with a hydraulic structure
- Multiple 2D flow areas in the same geometry
- Directly connect multiple 2D flow areas with hydraulic structures
- Simplified to very detailed Dam Breach analyses
- Simplified to very detailed Levee Breach analyses
- Mixed flow regime. The 2D capability (as well as the 1D) can handle supercritical and subcritical flow, as well as the flow transitions from subcritical to super critical and super critical to subcritical (hydraulic jumps).

2D flow modeling is accomplished by adding 2D flow area elements into the model in the same manner as adding a storage area. A 2D flow area is added by drawing a 2D flow area polygon; developing the 2D computational mesh; then linking the 2D flow areas to 1D model elements and/or directly connecting boundary conditions to the 2D areas.

Note: This document assumes that the reader already knows how to use HEC-RAS to perform 1D unsteady flow modeling. This document focuses on how to use the new 2D modeling capabilities and the new HEC-RAS Mapper features. For assistance with 1D unsteady flow modeling, and how to use the user interface, please review the main HEC-RAS User's Manual and Hydraulic Reference Manual.

HEC-RAS 2D Modeling Advantages/Capabilities

The 2D flow routing capabilities in HEC-RAS have been developed to allow the user to perform 2D or combined 1D/2D modeling. The 2D flow modeling algorithm in HEC-RAS has the following capabilities:

1. **Can perform 1D, 2D, and combined 1D and 2D modeling.** HEC-RAS can perform 1D modeling, 2D modeling (no 1D elements), and combined 1D and 2D modeling. The ability to perform combined 1D/2D modeling within the same unsteady flow model will allow users to work on larger river systems, utilizing 1D modeling where appropriate (for example: the main river system), and 2D modeling in areas that require a higher level of hydrodynamic fidelity.

2. **Shallow Water Equations (SWE) or Diffusion Wave Equations (DWE) in 2D.** The program solves either the 2D Shallow Water equations (with optional momentum additions for turbulence, wind forces, mud and debris flows, and Coriolis effects) or the 2D Diffusion Wave equations. This is user selectable, giving modelers more flexibility. In general, the 2D Diffusion Wave equations allow the software to run faster and have greater stability properties. The 2D Shallow Water equations are applicable to a wider range of problems. However, many modeling situations can be accurately modeled with the 2D Diffusion Wave equations. Because users can easily switch between equation sets, each can be tried for any given problem to see if the use of the 2D Shallow Water equations is warranted over the Diffusion Wave equations.

3. **Implicit Finite Volume Solution Algorithm.** The 2D unsteady flow equations solver uses an Implicit Finite Volume algorithm. The implicit solution algorithm allows for larger computational time steps than explicit methods. The Finite Volume Method provides an increment of improved stability and robustness over traditional finite difference and finite element techniques. The wetting and drying of 2D cells is very robust. 2D flow areas can start completely dry, and handle a sudden rush of water into the area. Additionally, the algorithm can handle subcritical, supercritical, and mixed flow regimes (flow passing through critical depth, such as a hydraulic jump) without any special options to turn on.

4. **1D and 2D Coupled Solution Algorithm.** The 1D and 2D solution algorithms are tightly coupled on a time step by time step basis with an option to iterate between 1D and 2D flow transfers within a time step. This allows for direct feedback each time step between the 1D and 2D flow elements. For example,

consider a river that is modeled in 1D with the area behind a levee is modeled in 2D (connected hydraulically with a Lateral Structure). Flow over the levee (Lateral Structure) and/or through any levee breach is computed with a headwater from the 1D river and a tailwater from the 2D flow area to which it is connected. The weir equation is used to compute flow over the levee and through the breach. Each time step the weir equation uses the 1D and the 2D results to compute the flow allowing for accurate accounting of weir submergence, at each time step, as the interior area fills up. Additionally, flow can go back out of the breach (from the 2D area to the 1D reach), once the river stages subside.

5. **Unstructured or Structured Computational Meshes.** The software was designed to use unstructured computational meshes but can also handle structured meshes. A structured mesh is treated the same as an unstructured mesh. The software assumes that cells are orthogonal to each other (the face between two cells is perpendicular to a line connecting the two cell centers). Assuming orthogonality simplifies some of the computations required and improves computational speed. This means that computational cells can be triangles, squares, rectangles, or even five and six-sided elements (**the model is limited to elements with up to eight sides**). The mesh can be a mixture of cell shapes and sizes. The outer boundary of the computational mesh is defined with a polygon. The computational cells that form the outer boundary of the mesh can have very detailed multi-point lines that represent the outer face(s) of each cell.

6. **Detailed Hydraulic Property Tables for 2D Computational Cells and Cell Faces.** Within HEC-RAS, computational cells do not have to have a flat bottom, and cell faces/edges do not have to be a straight line, with a single elevation. Instead, each Computational cell and cell face is based on the details of the underlying terrain. This type of model is often referred to in the literature as a "high resolution subgrid model" (Casulli, 2008). The term "subgrid" means it uses the detailed underlying terrain (subgrid) to develop the geometric and hydraulic property tables that represent the cells and the cell faces. HEC-RAS has a 2D flow area pre-processor that processes the cells and cell faces into detailed hydraulic property tables based on the underlying terrain used in the modeling process. For an example, consider a model built from a detailed terrain model (2ft grid-cell resolution) with a computation cell size of 200x200 ft. The 2D flow area pre-processor computes an elevation-volume relationship, based on the detailed terrain data (2ft grid), within each cell. Therefore, a cell can be partially wet with the correct water volume for the given water surface elevation (WSEL) based on the 2ft grid data. Additionally, each computational cell face is evaluated similar to a cross section and is pre-processed into detailed hydraulic

property tables (elevation versus - wetted perimeter, area, roughness, etc…). The flow moving across the face (between cells) is based on this detailed data. This allows the modeler to use larger computational cells, without losing too much of the details of the underlying terrain that govern the movement of the flow. Additionally, the placement of cell faces along the top of controlling terrain features (roads, high ground, walls, etc…) can further improve the hydraulic calculations using fewer cells overall. The net effect of larger cells is fewer computations, which means much faster run times. An example computational mesh overlaid on detailed terrain is illustrated in Figure 1-1.

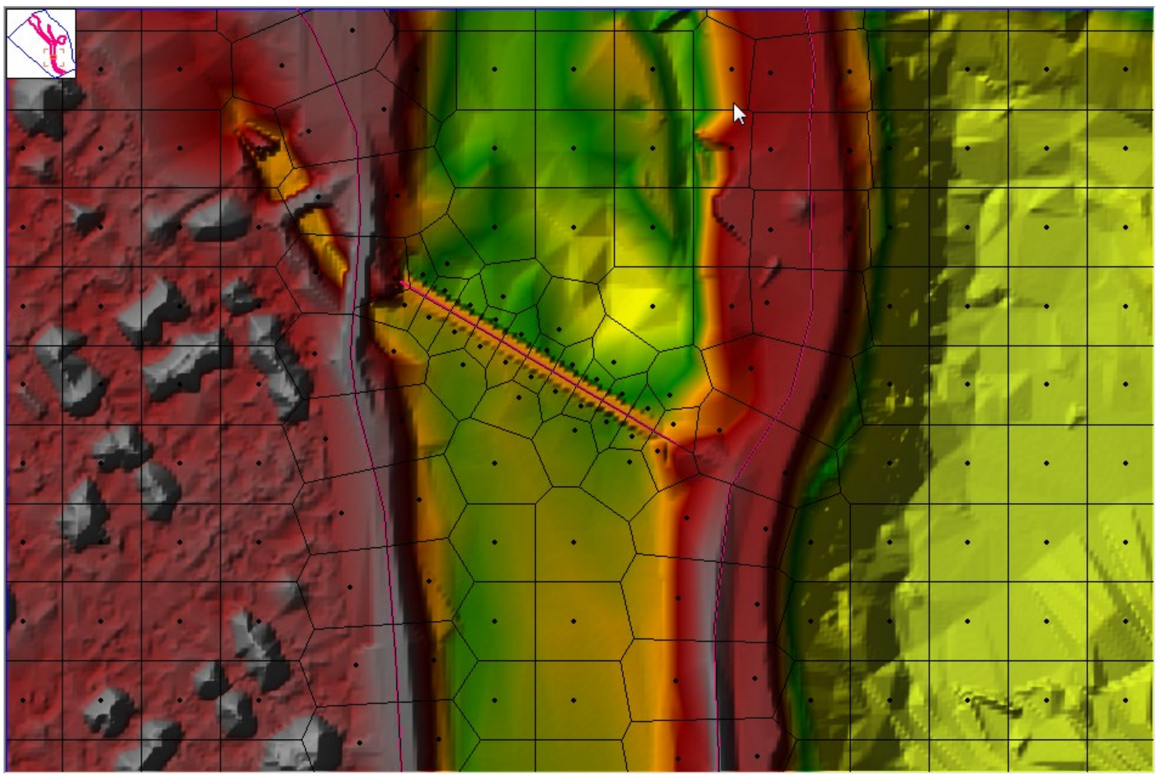


Figure 1-1. Unstructured computational mesh with detailed sub-grid terrain data.

Shown in Figure 1-1 is an example computational mesh over terrain data depicted with color shaded elevations. The computational cells are represented by the thick black lines. The cell computational centers are represented by the black dots and are the locations where the water surface elevation is computed for each cell. The elevation-volume relationship for each cell is based on the details of the underlying terrain. Each cell face is a detailed cross section based on the underlying terrain below the line that represents the cell face. This process allows for water to move between cells based on the details of the underlying terrain, as it is represented by the cell faces and the volume contained within that cell.

Therefore, a small channel that cuts through a cell, and is much smaller than the cell size, is still represented by the cell's elevation volume relationship, and the hydraulic properties of the cell faces. This means water can run through larger cells, but still be represented with its normal channel properties. An example of a small channel running through much larger grid cells is shown in Figure 1-2. The example shown in Figure 1-2 has several canals that are much smaller than the average cell size used to model the area (cell size was 500 x 500 ft, where the canals are less than 100 ft wide). However, as shown in Figure 1-2, flow is able to travel through the smaller canals based on the canal's hydraulic properties. Flow remains in the canals until the stage is higher than the bank elevation of the canal, then it spills out into the overbank areas.

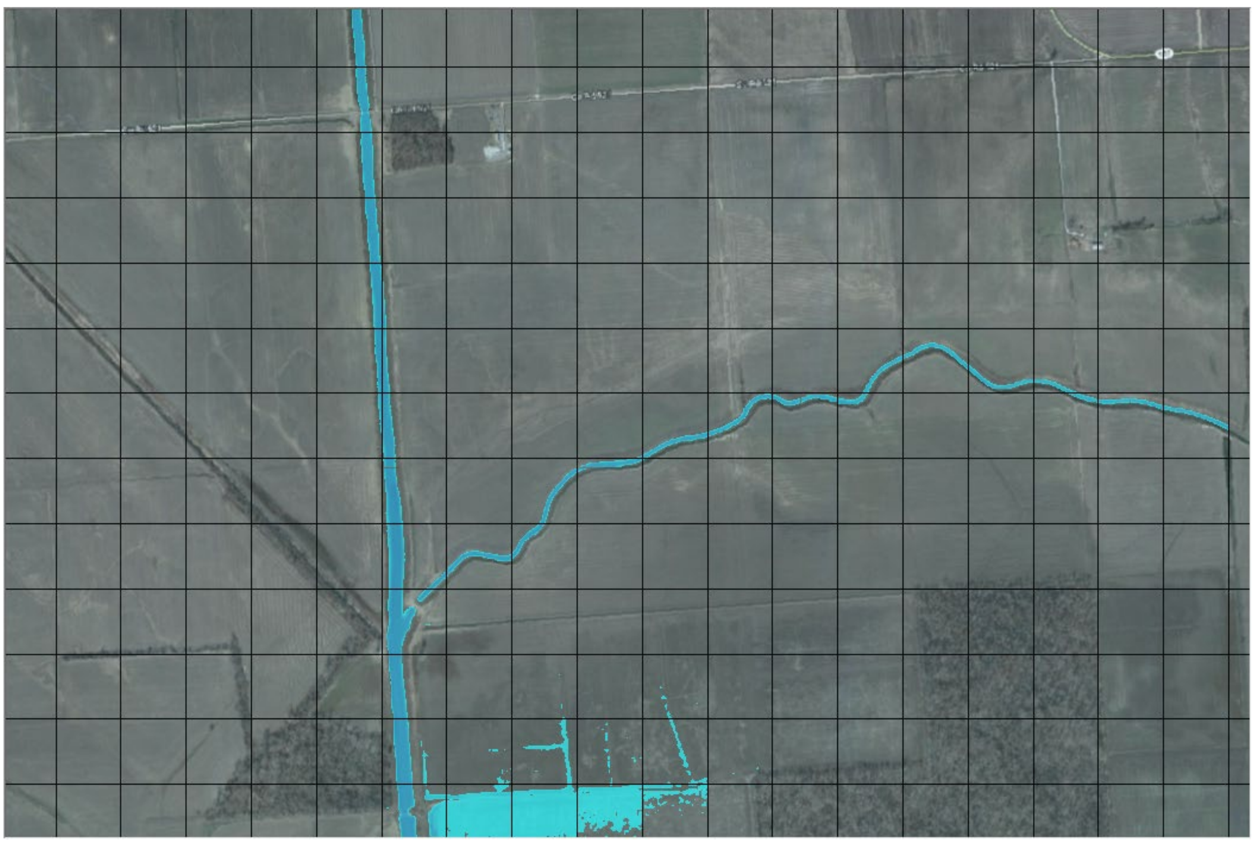


Figure 1-2. Example showing the benefits of using the detailed subgrid terrain for the cell and face hydraulic properties.

7. **Detailed Flood Mapping and Flood Animations.** Mapping of the inundated area, as well as animations of the flooding can be done inside of HEC-RAS using the RAS Mapper features. The mapping of the 2D flow areas is based on the detailed underlying terrain. This means that the wetted area will be based on the

details of the underlying terrain, and not the computational mesh cell size. Computationally, cells can be partially wet/dry (this is how they are computed in the computational algorithm). Mapping of the results will reflect those details, rather than being limited to showing a computational cell as either all wet or all dry.

8. **Multi-Processor Based Solution Algorithm (Parallel Computing).** The 2D flow area computational solution has been programmed to take advantage of multiple processors on a computer (referred to as parallelization), allowing it to run much faster than on a single processor.

9. **64-Bit Computational Engines.** HEC-RAS now comes with 64-bit computational engines. The 64-bit computational engines run faster than the previous 32-bit and can handle much larger data sets. 32-bit computations are no longer supported, and user's must be running a 64-bit operating system.

Develop a 2D or 1D/2D Unsteady Flow Model

Overview

Using HEC-RAS to perform 2D modeling or combined 1D/2D modeling is very easy and straight forward. The following are the basic steps for performing 2D (or combined 1D/2D) modeling within HEC-RAS:

1. Establish a Horizontal Coordinate Projection to use for your model, from within HEC-RAS Mapper. This is normally done by selecting an existing projection file from an ESRI shapefile or another mapping layer.
2. Develop a terrain model in HEC-RAS Mapper. The terrain model is a requirement for 2D modeling, as it is used to establish the geometric and hydraulic properties of the 2D cells and cell faces. A terrain model is also need in order to perform any inundation mapping in HEC-RAS Mapper.
3. Build a Manning's n layer data set within HEC-RAS Mapper, using land cover data layers and polygon layers, in order to establish Manning's n values within the 2D Flow Areas. Additionally, HEC-RAS has an option for user defined polygons that can be used to override the base Manning's n values. These user defined polygons could be used as calibration zones.
4. Add any additional mapping layers that may be needed for visualization, such as aerial photography, levee locations, road networks, etc.

5. From within the HEC-RAS Mapper draw a boundary polygon for each of the 2D Flow Areas to be modeled. Or you can import the X, Y boundary coordinates from another source.
6. Layout any break lines within the 2D flow area to represent significant barriers to flow, such as: levees, roads, natural embankments, high ground between main channel and overbank areas, hydraulic structures, etc.
7. Using the 2D Flow Area editor, create the 2D computational mesh for each 2D Flow Area.
8. Edit the 2D Flow Area mesh in order to improve it, such as: add additional break lines; use the refinement regions option to increase or decrease cell density as needed; Add, Move, or Delete cell centers where needed. Use the mesh refinement region tool to make a nice channel mesh.
9. Add any internal hydraulic structures or bridges inside of the 2D flow area(s) using the SA/2D Area Hydraulic Connection feature.
10. Run the 2D geometric pre-processor from RAS Mapper in order to create the cell and face hydraulic property tables.
11. Connect the 2D Flow Areas to 1D Hydraulic elements (river reaches, Lateral structures, storage area/2D flow area hydraulic connections) as needed.
12. From the RAS Mapper or the Geometric Data editor, draw any external boundary condition lines along the perimeter of the 2D flow areas.
13. Enter all of the necessary boundary and initial condition data for the 2D flow areas in the Unsteady Flow data editor.
14. From the Unsteady Flow Simulation window, set any necessary computational options and settings needed for the 2D flow areas.
15. Run the Unsteady flow simulation.
16. Review the combined 1D/2D output in RAS Mapper, as well as using the existing output capabilities for the 1D portions of the model.

Current Limitations of the 2D modeling Capabilities

The following is a list of the current limitations of the HEC-RAS 2D flow modeling software. These are items actively being worked on to improve the software, and will be available in future versions:

1. Cannot perform water quality modeling in 2D flow areas.

CHAPTER 2

Developing a Terrain Model and Geospatial Layers

It is essential to have a detailed and accurate terrain model in order to create a detailed and accurate hydraulics model. The quality of the terrain data can be a limiting factor in the quality of the hydraulics model the user can create. Terrain data comes from many different sources, formats, and levels of detail. Currently HEC-RAS uses gridded data for terrain modeling. It is up to the user to gather data from multiple sources, then convert/export it into a gridded data format that can be read in by HEC-RAS.

It is necessary to create a terrain model in HEC-RAS Mapper before the user can perform any model computations that contain 2D flow areas, or before the user can visualize any 1D, 2D, or combine 1D/2D mapping results. This section of the document describes how to create a terrain model in RAS Mapper, as well as other optional geospatial layers. The user can develop one or more terrain models that can then be associated with a specific geometry input file, or a specific results output file. Additionally, users have the option to create a Landcover data layer; Soils layer, and an Infiltration layer. Once a landcover layer is created, user can associate Manning's n values with each of the landcover types. Optionally users can specify a percent impervious for each landcover type (% Impervious is only necessary if the user is modeling precipitation and infiltration from within HEC-RAS). Soils data can be imported from GSSURGO database files, or the information can be imported from existing shape files. Soils data can be specified as either SCS Hydrologic Soil Groups or more detailed soil texture classes. An Infiltration layer can be derived from a landcover and Soils layer; just landcover; or just a soils layer. Users have three infiltration methods to choose from: Deficit-Constant; SCS Curve Number; or Green and Ampt.

For more details on creating terrain models and other geospatial layers with HEC-RAS Mapper, please review the HEC-RAS Mapper User's manual.

Opening RAS Mapper

The first step in developing a terrain data set is to open RAS Mapper. This is accomplished by selecting **GIS Tools** from the HEC-RAS main window, then selecting **RAS Mapper**, or by pressing the **RAS Mapper** button  on the HEC-RAS main window. When this is done, the window shown in Figure 2-1 will appear (the users may be blank at first).

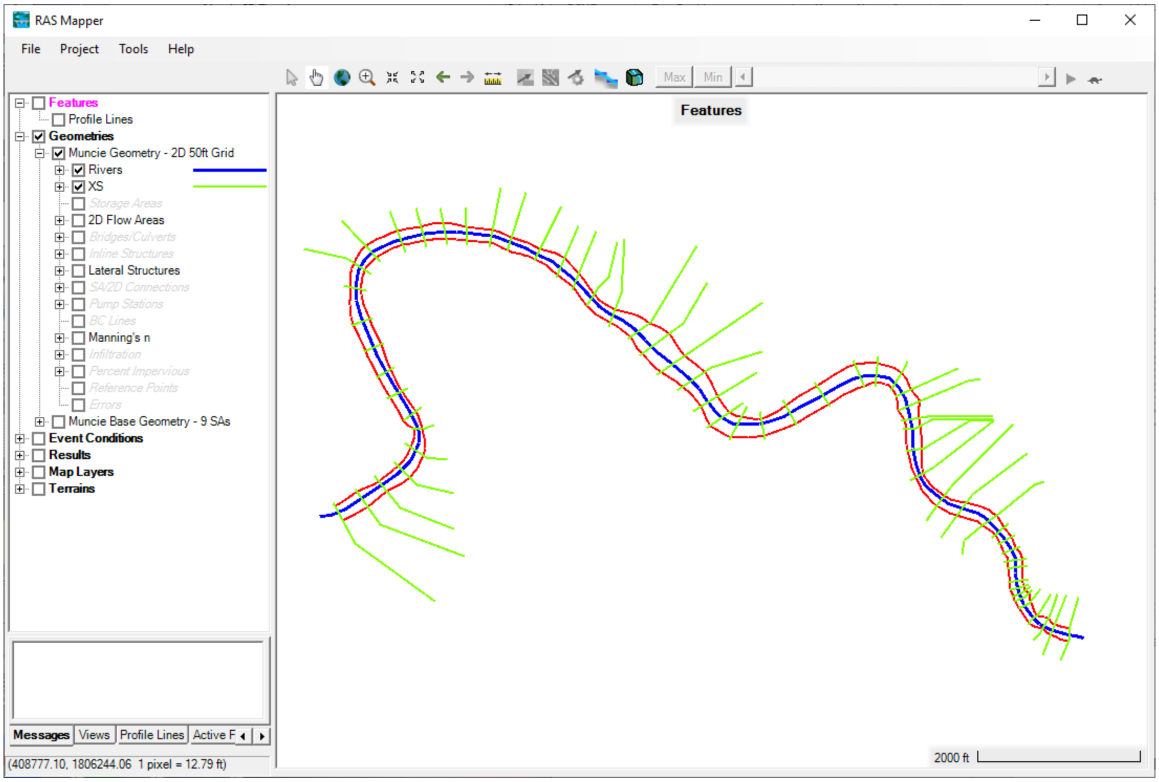


Figure 2-1. RAS Mapper with no terrain or other map layers loaded.

Setting the Spatial Reference Projection

Once RAS Mapper is open, if the data is in a specific spatial coordinate projection, that projection should be set in RAS Mapper. Setting a spatial coordinate system is not required (i.e., maybe the user is just doing some testing of hypothetical data), but using one has many advantages in HEC-RAS and HEC-RAS Mapper.   To set the spatial reference system for the project, select the **Project | Set Projection** menu item from the RAS Mapper menu bar. When the **Set Projection** option is selected the window shown below will appear (Figure 2-2).

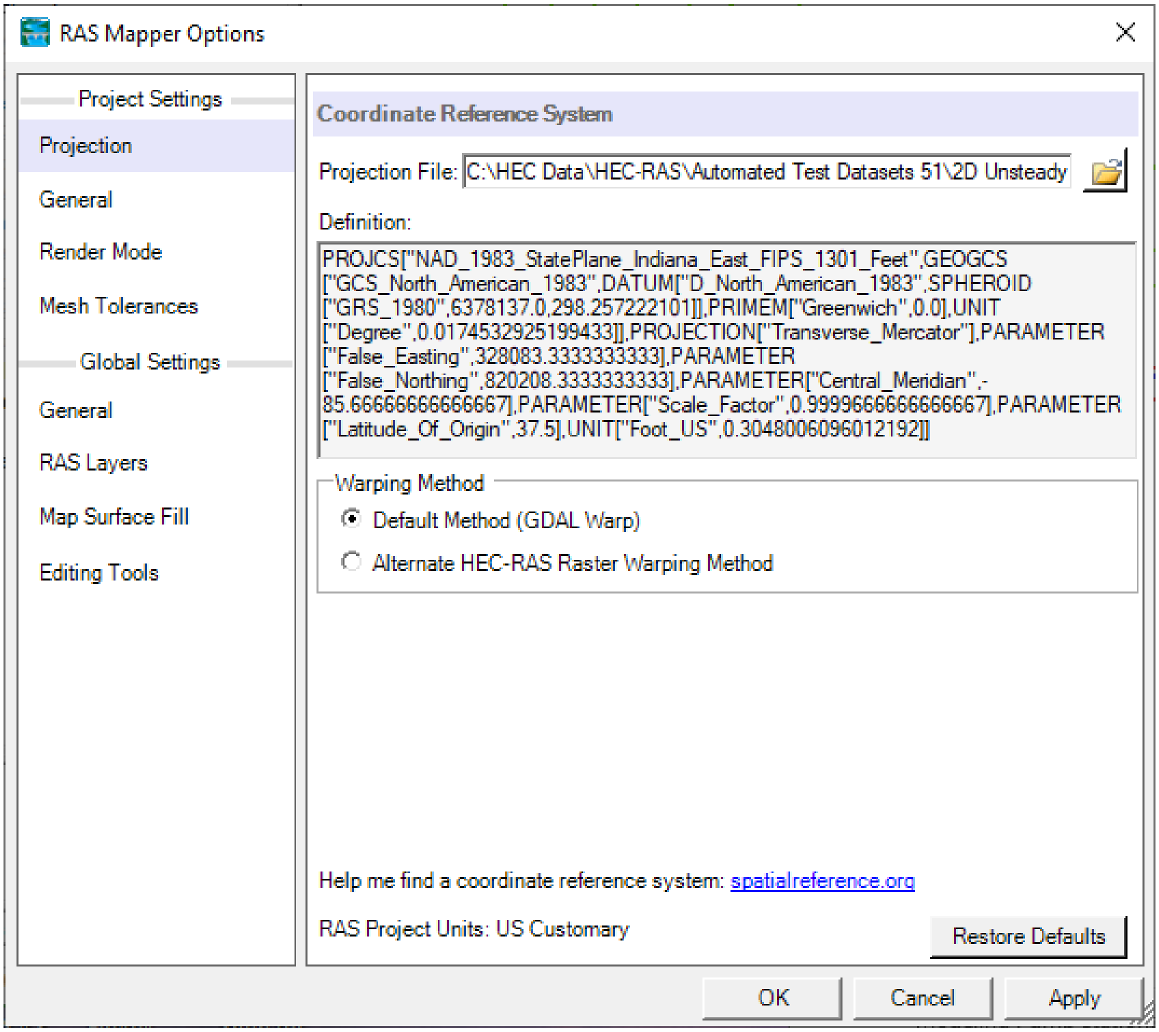


Figure 2-2. Editor to set the RAS project's spatial reference system.

To set the spatial reference system (coordinate system), browse and select an existing ".prj" file (ESRI projection file) that contains the correct coordinate system. If ArcGIS Version 10.0 or earlier) is installed on the computer, the user can browse to the ArcGIS directory that contains a listing of all the available coordinate systems and select the appropriate one. Otherwise, find an ArcGIS projection file (*.prj) within one of the GIS project directories (look for a shapefile that has a projection file defined). Unfortunately, the directory of coordinate systems was removed at ArcGIS version 10.1 and newer. So, users will have to create one with ArcGIS or search on the internet for an ArcGIS projection file. For this example, "NAD_1983_StatePlane_Indiana_East.prj" was selected.

Loading Terrain Data and Making the Terrain Model

The next step is to load the terrain data that will be used in creating the terrain model for HEC-RAS. To develop a new terrain data set (terrain model), right click on the Terrains layer and select **Create New RAS Terrain** menu item from the **RAS Mapper** pop up window. The **New Terrain Layer** dialog will appear (Figure 2-3). This dialog allows the user to provide a name for the new Terrain Layer (**Filename** field, the default name is "Terrain"); select a directory for storing the terrain (**Folder** button); define the elevation precision of the new terrain data layer (**Rounding (Precision)** field, 1/32 is the default for English units); and select the files to be used in building the new terrain layer (**Plus (+)** button).

At this time, RAS Mapper can import terrain data that is in the floating-point grid format (*.flt); GeoTIFF (*.tif) format; ESRI grid files; and several other formats (for example a USGS DEM file). A list of file formats supported by the RAS Mapper software is contained in **Appendix B** of this manual. We have not tested all of these file formats, but the library we are using says it supports these file formats. Whatever format you use, the data must be in a gridded format, in order to be used to make a terrain model

Floating point grids consist of a main file with the *.flt file extension, and they also have a *.hdr file, and possibly a *.prj file that goes along with it. **Note: if the *.flt file is not in the same projection as what has been set in RAS Mapper, then the user must have a *.prj file that describes the projection of the *.flt file**). ESRI grid files will have *.adf file extensions. **Note: there are several *.adf files that make up an ESRI grid. Pick any one of them and the program will process all of them as needed.** Use the **Plus (+)** button to get a file chooser, then select the terrain grid or grids (more than one grid can be used simultaneously to form a tiled terrain model), then select the **Open** button to use the selected files.

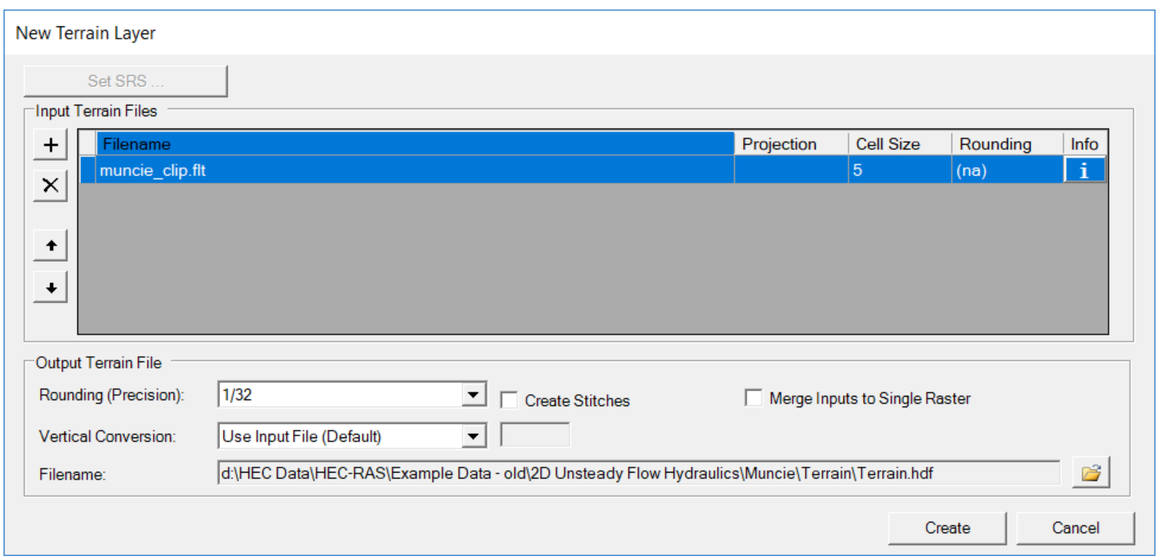


Figure 2-3. Example New Terrain Layer dialog.

NOTE: If more than one grid file is loaded, use the up and down arrow buttons to the left of the table to set the priority of the grid layers. If one grid has more detail (finer resolution) than others, the user will want to give it a higher priority for creating the combined Terrain Layer. If there is only one Terrain layer, which will be the case for many studies, the name may be left as “Terrain”, or another name can be given.

Once the grid files are selected, and placed in the appropriate priority order, press the **Create** button to create the new Terrain Layer. Once the **Create** button is pressed, RAS Mapper will convert the grids into the GeoTIFF (*.tif) file format. The GeoTIFF file structure supports tiled and pyramided data. Tiled data uses less area of the terrain by removing the “NoData” values, while pyramided data stores multiple terrain layers of varying resolutions. Additionally, the GeoTIFF files are automatically stored in a compressed form (using the zip format), which makes the file storage much smaller. In general, the GeoTIFF files will be 2 to 20 times smaller than the original FLT or ESRI grid files. The GeoTIFF file format allows for smaller storage space, faster computational speed (in generating flood maps), as well as “dynamic mapping” of the results (depth grids that are created on the fly in memory, as the user zooms in/out, pans, or animates the flood maps).

Once the GeoTIFF files are created, RAS Mapper also creates a *.hdf file and a *.vrt file. The *.hdf (Hierarchical Data Format) file contains information on how the multiple GeoTIFF files are stitched together. The *.vrt (Virtual Raster Translator) file is an XML file that contains information about all of the raster files (*.tif). The user can drag and drop the *.vrt file onto an ArcGIS project and it will then know about all of the raster files that make up the terrain layer. Additionally, they will have the same scale and color ramp when they are plotted. Once RAS Mapper has completed the conversion of the files to GeoTIFF, and then created the HDF and VRT file, the new terrain layer will be visible in the window. See the example shown below in Figure 2-4.

Figure 2-4. RAS Mapper with a Terrain Data Layer added.

Once the terrain model is created the user can enhance the look of the terrain data by right clicking on the terrain layer and selecting **Image Display Properties**. The **Image Display Properties** window (Figure 2-5) allows the user to: select and control the Surface Color Ramp; Transparency; Create and plot Contour Lines; and shade the terrain using a Hill Shading algorithm (Hill Shading makes the visualization of the terrain much more realistic and semi 3D).

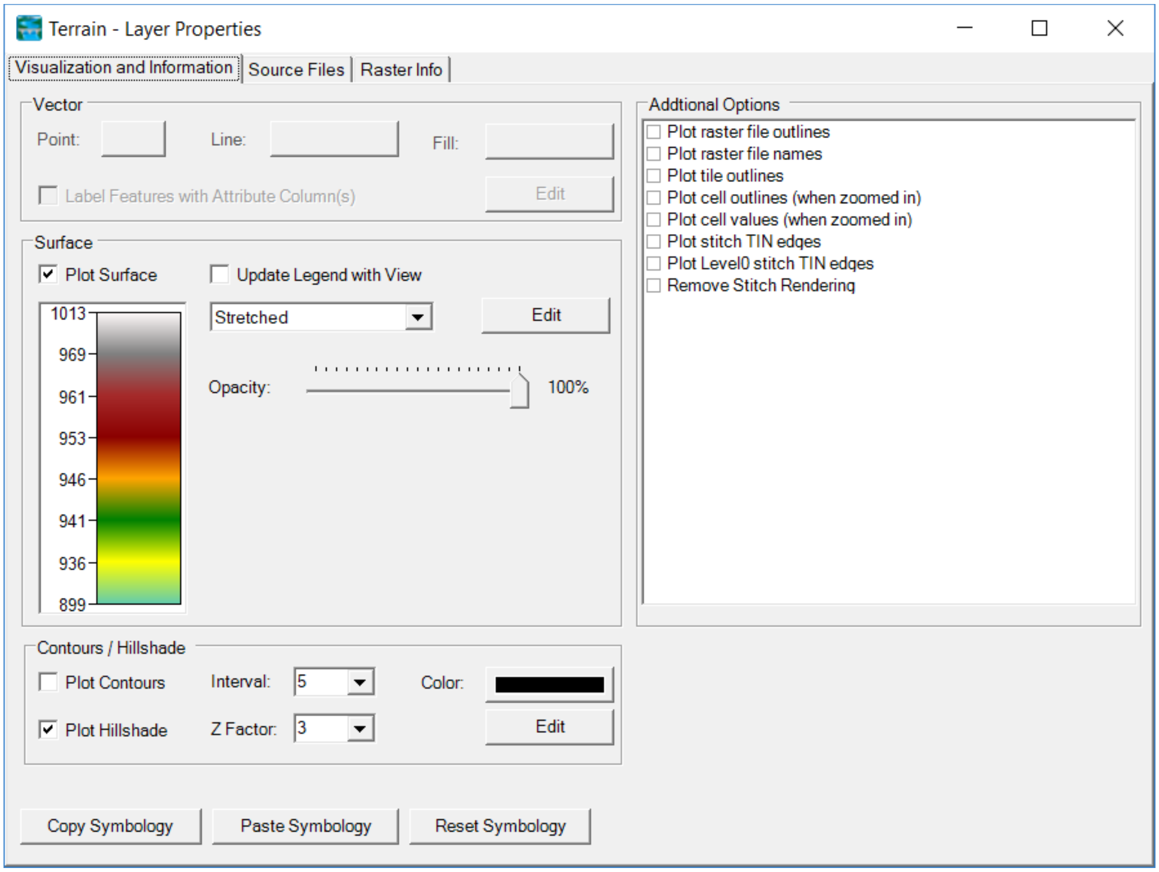


Figure 2-5. Layer Properties Window for the Terrain Data Layer.

An example of terrain data with some of the layer properties enhancements (Hill Shading and Contour Lines) turned on is shown in Figure 2-6.

Note: After a Terrain data set is created, the user will be able to display this terrain layer as a background image in the HEC-RAS geometry editor. Terrain layers, and any other Map Layers developed in RAS Mapper are available for display in the HEC-RAS Geometry editor.

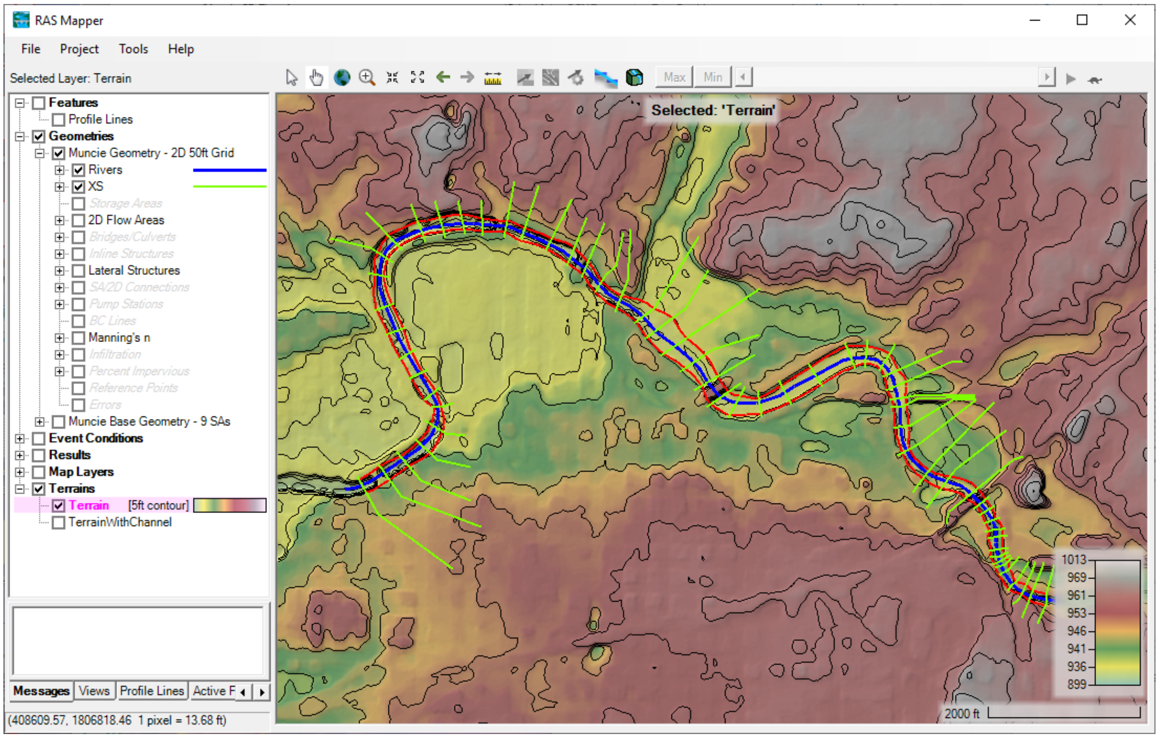


Figure 2-6. Terrain Data with Hill Shading and Contour Lines Turned On.

Using Cross Section Data to Modify/Improve the Terrain Model

One of the major problems in hydraulic modeling is that terrain data does not often include the actual terrain underneath the water surface in the channel region. HEC-RAS Mapper can be used to create a terrain model of the channel region from the HEC-RAS cross sections and the Cross-Section Interpolation Surface. This terrain model can then be combined with the general surface terrain model (that does not accurately depict the terrain below the water surface) to create an improved terrain model for hydraulic modeling and mapping.

NOTE: HEC-RAS Mapper now has a wide set of vector-based terrain modification tools that can be applied to any HEC-RAS terrain model. To learn about the vector-based terrain modification tools, and how to use them, please see the HEC-RAS Mapper User's manual.

The steps to include a channel in a terrain model using HEC-RAS cross sections are the following:

Creating a Terrain Model of the Channel

From HEC-RAS Mapper, turn on the Geometry layer for the 1D cross section geometry data to be used in creating the channel terrain model. Also turn on the following sublayers: **River (stream centerline)**; **River - Bank Line**; **XS** (cross sections); and **XS - Interpolation Surface**. Review the stream centerline (River); Bank Lines, XS (cross sections); and the XS Interpolation Surface to ensure they are correct, and what the user wants for a new channel terrain model.

NOTE: it is generally a good idea to make a copy of the 1D cross sections geometry that will be used to make the channel terrain. Then move the main channel bank stations inside of the channel down to the level at which the terrain model information is not correct. By doing this, when you export the 1D channel data as a terrain model, it will only create a terrain model of the area that is not defined well in the original terrain model. This approach will also prevent good terrain data from being overwritten by terrain data created from the interpolation of 1D cross sections. See an example with the desired geometry sublayers selected in Figure 2-7 below.

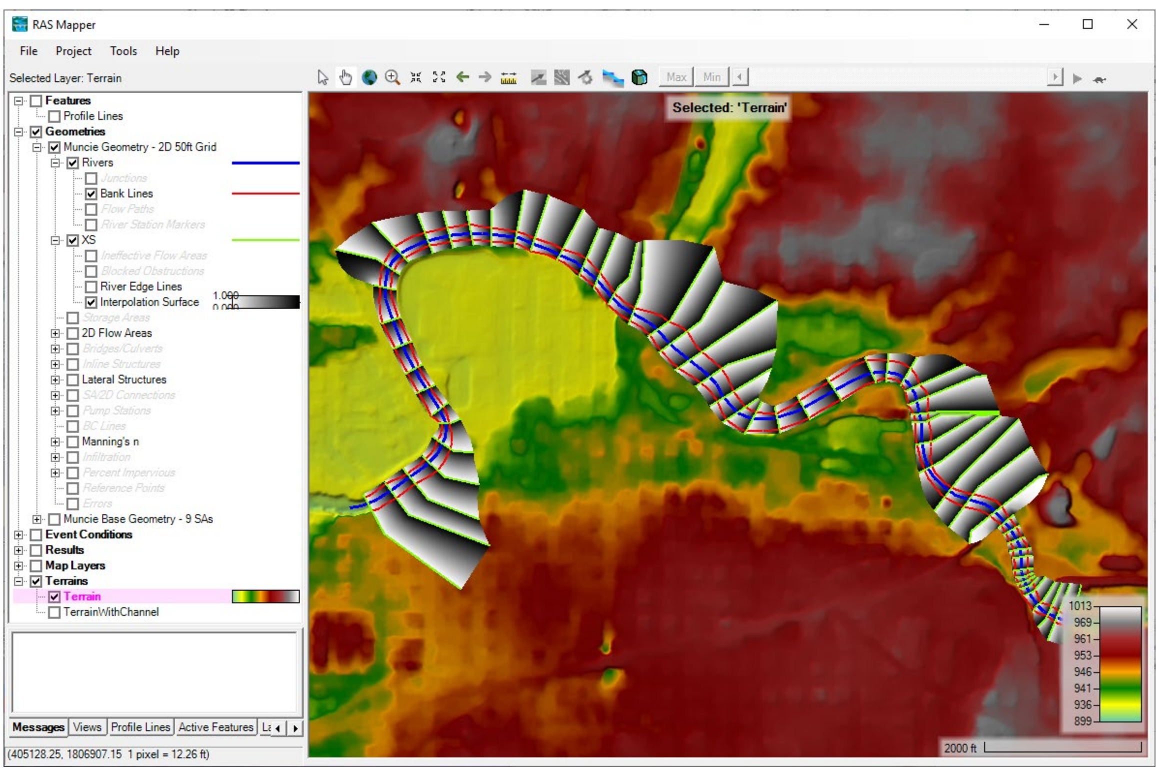


Figure 2-7. RAS Mapper with base terrain and Geometry Layers Displayed.

Once the geometry layers are completed, the channel terrain model is created by right clicking on the Geometry layer (parent name) and selecting **Export Layer**, then **Create Terrain GeoTIFF from XS's (channel only)**. Alternatively, the user can make a terrain model out of the entire cross section region (channel and overbank area), but if the base terrain model has good overbank terrain information, the user will not want to do that. Once the **Export** option is selected, a file selector will appear, in which the user will need to give the new terrain model a name, and choose a directory to put it in. See the file selector in Figure 2-8.

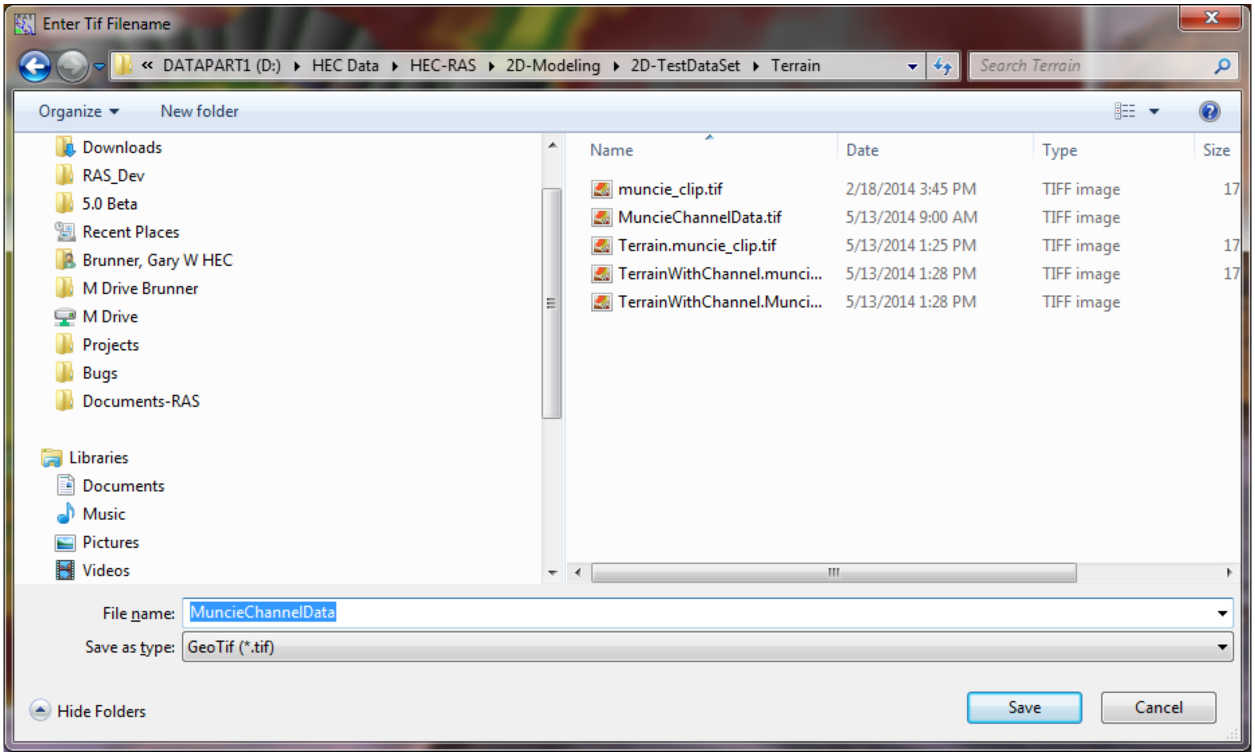


Figure 2-8. Terrain Export File Choose shown with Example Name and Directory.

Once a directory is chosen and a filename is entered, press the **Save** button to create the channel terrain model. The program will then ask the user for the raster cell size to use for this new terrain model. For example, if the user enters "2.0", then the new terrain model will have grids that are 2 x 2 ft. The terrain model is created by taking the elevation data from the cross sections and using the interpolation surface to interpolate an elevation for each grid cell between two cross sections. This new surface is clipped at the main channel bank stations (if the user selected to make a terrain of the channel only), and then written as a terrain grid in the GeoTIFF file format.

Making a Combined Channel and Overbank Terrain Model

Once the user has a terrain model from the channel data, a new combined terrain model can be made from the base terrain model (the terrain with the overbank/floodplain data) and the newly created channel only terrain model. To make the new combined terrain model, select the **Tools** menu from RAS Mapper, and then select **New Terrain**. This step is the same as previously described for creating a terrain model in RAS Mapper, however, the files used to create this terrain model will be the previously created GeoTIFF of the base terrain data, and the newly created GeoTIFF of the channel only data. Once the **New Terrain** option is selected, the window to make a new terrain layer will appear (Figure 2-3). Enter a new Name for the new terrain model ("TerrainWithChannel" was used in this example). Select the folder for the new terrain model to be written to. Select the precision of the new terrain model, keeping in mind the precision should not be finer than the terrain files used to create this new terrain model. Then press the **Plus** button and select the base terrain models GeoTIFF file, and the channel only terrain models GeoTIFF file. Make sure that the new channel-only terrain model has a higher priority than the base terrain model (i.e. make sure it is first in the list of the added terrain files). Then press **Create** button and a new combined terrain model will be created and added to the RAS Mapper project. See the original (terrain model without cross section data included) and the new (terrain model with cross section data included) terrain models in Figure 2-9 below.

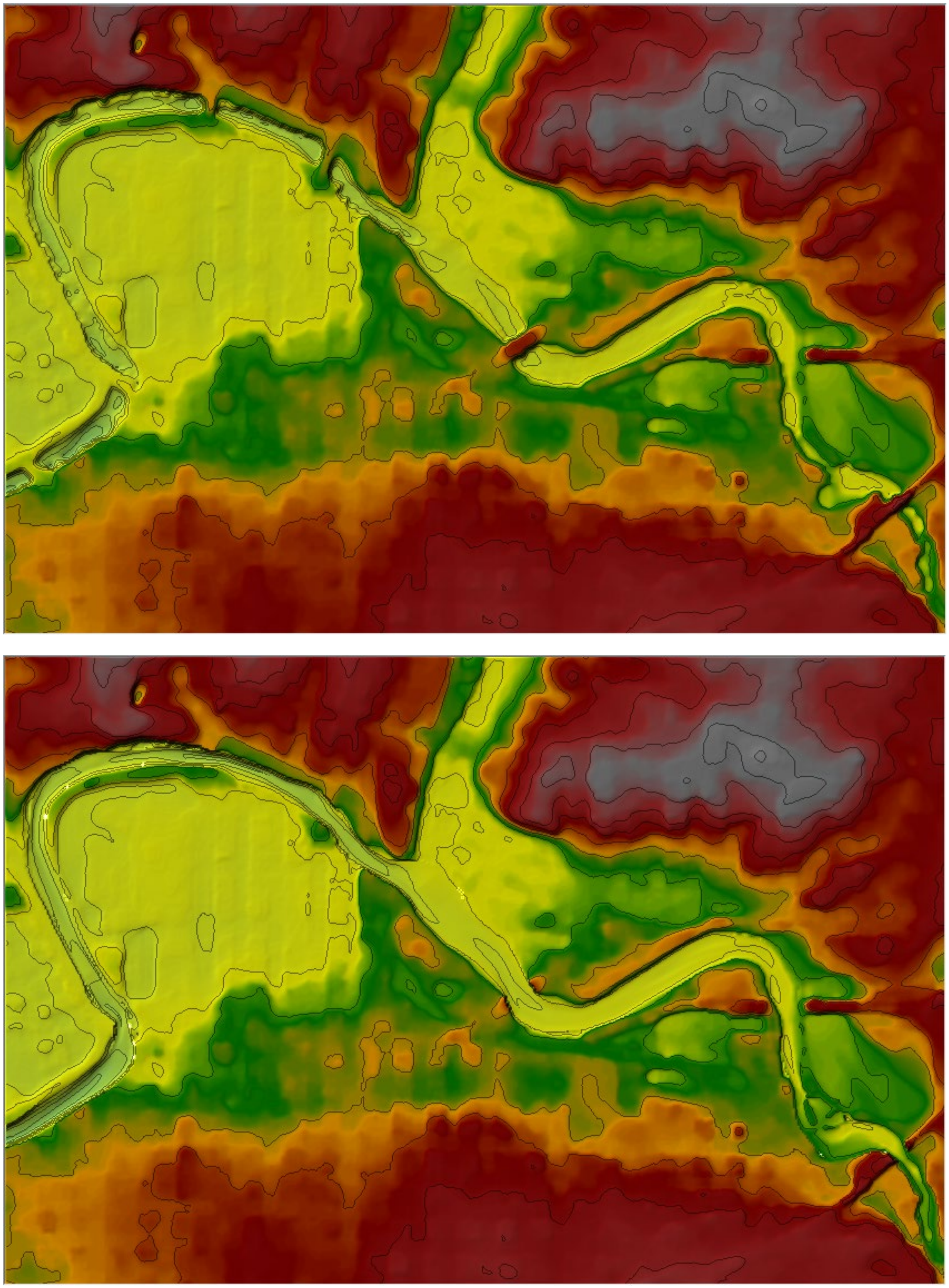


Figure 2-9. Original Terrain model (Top) and New Terrain model with Channel Data (Bottom).

Creating Land Cover, Manning's n values, and % Impervious Layers

A spatially varying land cover layer can be created in RAS Mapper, and then associated with a specific geometry data set. Once a land cover data set is created, the user can specify Manning's n values to be used for each land cover type. Additionally, the user can create their own land cover classification polygons (user defined polygons), in which they can override the base land cover layers within that polygon and define a new land cover type. User defined classification polygons are often used for channel areas, as national land cover data sets do not adequately define the correct area for the main channels. The creation of user defined landcover classifications will allow users to create a good set of base Manning's n values for these user defined classification polygons. More than one polygon can be drawn and given the same name. However, only one Manning's n value can be set for each user defined land cover classification polygon. So, if you want the Manning's n values to be different, then the polygons must have different names. Users can also specify percent impervious for each of the land cover types. Percent impervious is optional, and only necessary if the user is modeling precipitation and infiltration.

Once a Land cover/Manning's n layer is created and associated with a specific Geometry data set, the user can create Calibration regions (polygons) in which they can use to override all of the Manning's n values associated with land cover within that polygon. These Calibration regions only apply to that specific geometry, and do not change the base land cover/Manning's n value layer. This will allow user to come up with different Manning's n values for different calibration events, if necessary.

This portion of the document will discuss creating a Land cover data set; Manning's n Values and Percent Impervious; User defined land cover classification polygons; Associating Land Cover/Manning's n with Geometry Data; and Manning's n Calibration Regions.

NOTE: User's must have a land cover data set in order utilize spatially varying Manning's n values within 2D Flow Areas, and to also utilize the capability of specifying User defined Land Cover Classification Polygons.

Creating a Land Cover Data Set

In the current version of HEC-RAS, users can import land use information in both polygon (shapefile) and gridded formats. Custom user defined Shapefile layers can be created by users in HEC-RAS Mapper. Gridded landcover data can be obtained from USGS websites (NLCD 2016 and USGS LULC), as well as other sources. RAS Mapper allows the user to use multiple land use data files and types, to create a single land use coverage layer in HEC-RAS. For example, a user may want to use the USGS NLCD 2016 gridded land used data (which is available for the whole USA) as their base land use coverage data. However, they may want to also find or generate a polygon coverage (shapefile) that is more accurate for many of the areas within their study region (i.e. the main channel regions, buildings, roads, etc…). By setting the more accurate shapefile as the higher priority, the land use from the shape file will be used unless it does not cover portions of the area, then the USGS gridded data will be used for those areas. RAS Mapper ingests the various land use data types and creates a combined land cover layer and stores it as a GeoTIFF file (there is also a companion *.hdf file generated).

NOTE: The two example 2D data sets that come with HEC-RAS (Muncie.prj and BaldEagle.prj) contain land use information for defining Manning's n values. Please open one of these data sets and use it as a guide along with the discussion in this manual.

To create a spatially varying land cover layer within HEC-RAS, go to RAS Mapper, then right click on **Map Layers** in the tree on the left, then select **Create New RAS Layer| Land Cover Layer**. This selection will bring up the window shown in Figure 3-16. The Land Cover Layer window is broken into three sections: **Input Files; Unique Classification Names for Selected File;** and **Output File**. The Input Files section is for selecting the grid and shapefiles to be used as input, as well as setting their priority. The Unique Classification Names for Selected File section is used to display the numeric value (Integer) and the text label of the land cover data for the file currently selected (highlighted) in the Input Files section. The Output File section is used to show what HEC-RAS will use for the RAS Classification name and their numeric ID.

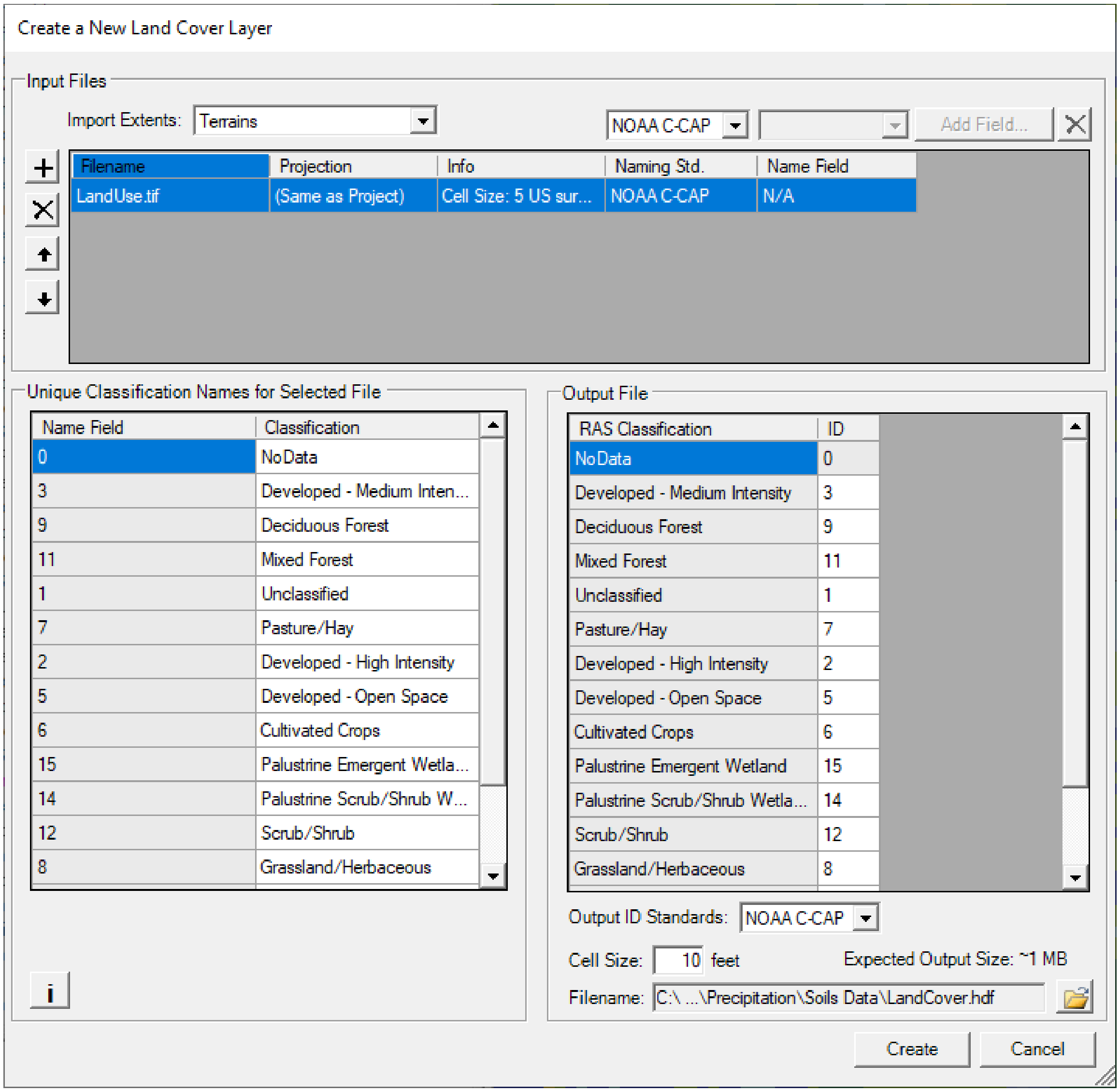


Figure 2-10. RAS Mapper New Land Cover Classification Layer Editor.

The window shown in Figure 2-10 allows the user to select one or more land use files of varying type. This is accomplished under the **Input** Files area by selecting the **Plus (+)** button. Once the plus button is selected a file chooser will come up allowing you to select a land use coverage file. If more than one land use file is selected, use the up and down arrow to set the priority of the files. The file at the top of the list has the highest priority, and so on. Because HEC-RAS supports multiple land use files and types, the user will have to either select one of the established naming conventions or enter their own naming convention for each land use type. Currently there are three options for defining the names of the land use types: NLCD 2016 (which is from the USGS 2011 land use data coverage); Anderson II (developed by James R. Anderson, et al, from the USGS in 1976); and NOAA C-CAP.

From the Input File section of the editor, if the user selects (highlights) a shapefile, then the **Name Field** column will be active, and the user can select a column contained in the shapefile to be used for that field. This is accomplished by selecting from the drop-down menu chooser above each of the fields. If the user selects (highlights) a gridded land cover data set, then only the **Naming Std.** field will be available for identifying that column within the data.

When you select a land cover layer in the Input File section of the editor**,** the "Unique Classification Names for Selected File" section of the editor will show what is contained within the file for the **Name Field (**integer value or text label**)**, and also what HEC-RAS will use as an identifier for that specific land cover type. Because different land cover data sources use different naming conventions, if the user has more than one input file, the software must come up with a single naming convention to use for all of the data. The combined naming convention that HEC-RAS will use is shown in the **Output File** section of the window. The Output File section shows the final naming convention it will use, along with the integer ID's. Manning's n values could have been selected from a Shapefile, or the user can edit/enter them directly into the Land cover data table.

The user can choose an **Output ID Standard** from the drop down below the Output File table. This is generally used when you have a Shapefile, and you want to apply one of the USGS naming standards to that file. If you have more than one Input land cover file, then normally, the option labeled **Custom** will be the only option, as a single standard will generally not work for multiple land cover types. Also, when using more than one land cover layer type (Shapefile and grid), there will be different naming conventions within the two files for the same land cover type. The USGS naming conventions use specific integer ID's for their associated land cover type, while integer ID's get assigned to land cover types for Shapefiles. If the same ID gets assigned to two different land cover types, the software will display an error in red below the output table with the label "**Duplicate IDs"**. The user must change one or more of the duplicate ID's to a unique integer identifier, currently not used in the table.

The filename and output directory for the new land classification layer is shown at the bottom of the window. The user should select a directory to be used for the HEC-RAS land cover layer, and also enter a name for this new layer. RAS Mapper takes all of the input layers and creates a single land cover output layer in the *.tif file format. The last step before creating a Land Classification dataset, is to enter a cell size (i.e. 1, 2, 5, 10 ft) that makes sense with the computational cell size being used, and the spatial accuracy needed for the land classification (and therefore roughness) values. After all the data is entered press the **Create** button, RAS Mapper will read the input file layers and convert them into a single GeoTIFF file in the user define output directory. A window will appear telling the user the progress and when it has finished creating the file. When the user presses the **OK** button, both windows will close.

Shown in Figure 2-11 below is an example Land Classification layer in RAS Mapper. The user can control the color of each land use category, and the transparency used to display the polygons. The display of the land use classification is controlled by right clicking on the layer and selecting **Image display properties**. This will bring up a window allowing the user to control the colors and transparency of the polygons.

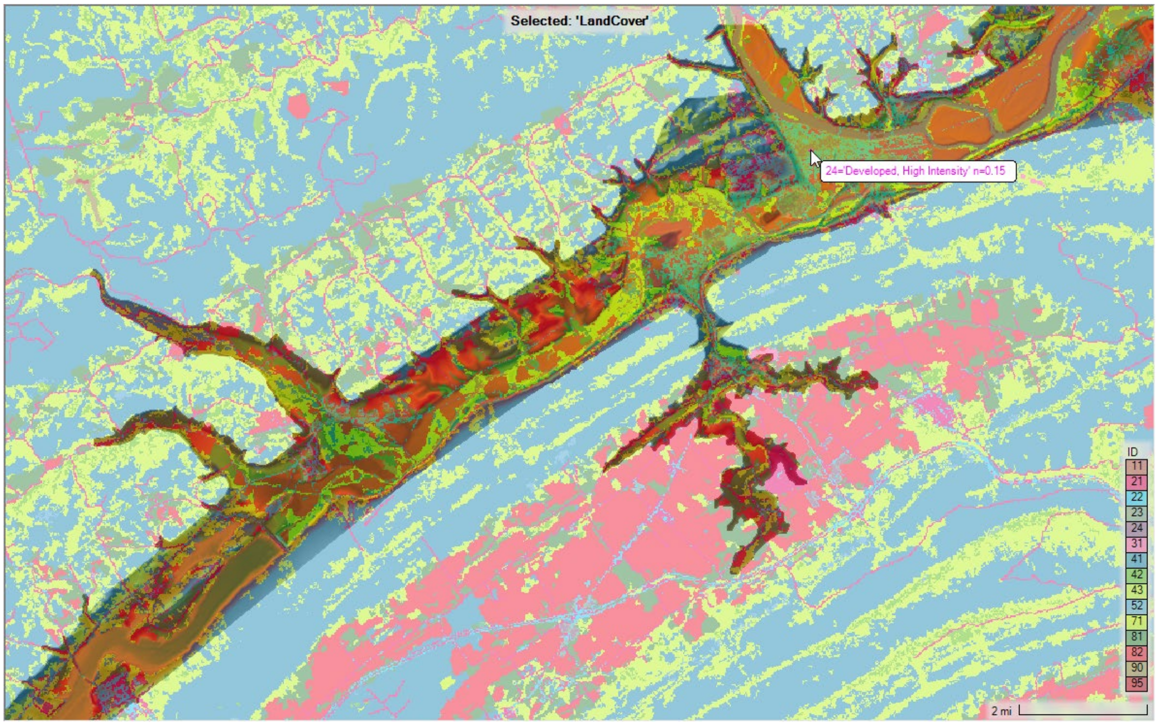
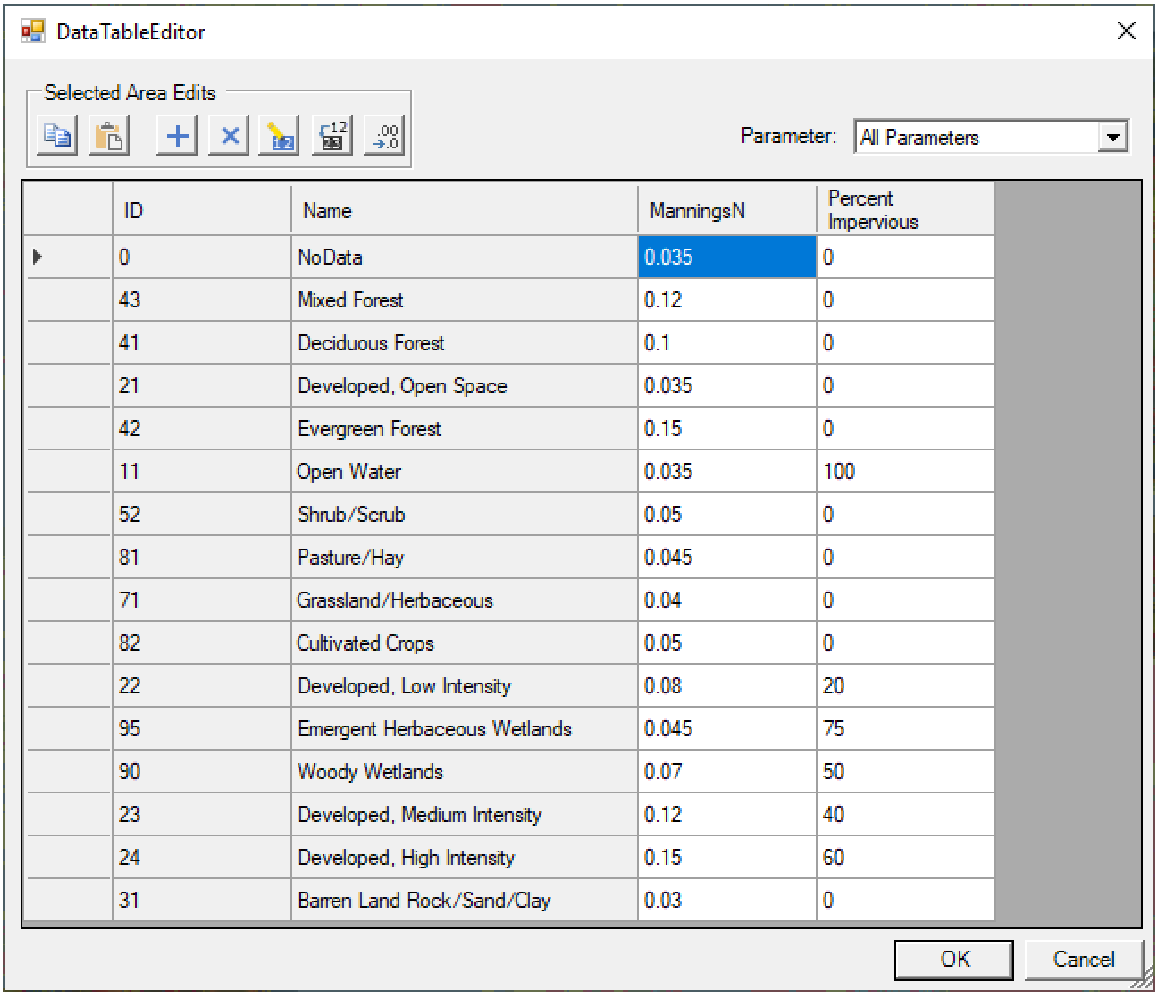


Figure 2-11. Example Land Cover Layer containing Land Classification values.

Manning’s n Values and Percent Impervious

Once a Land Cover layer has been created, the user can then build a table of Land Cover versus Manning’s n values, which can then be used in defining roughness values for 2D flow areas and 1D river reaches. This Land Cover versus roughness table is developed from within the HEC-RAS Mapper, directly within the Land cover layer. To create the Manning’s n vs Land Cover table, right click on the Land Cover Layer of choice under the Map Layers, the select **Edit Land Cover Data**. When this option is selected, the table in Figure 2-12 will appear.



ID	Name	ManningsN	Percent Impervious
0	NoData	0.035	0
43	Mixed Forest	0.12	0
41	Deciduous Forest	0.1	0
21	Developed, Open Space	0.035	0
42	Evergreen Forest	0.15	0
11	Open Water	0.035	100
52	Shrub/Scrub	0.05	0
81	Pasture/Hay	0.045	0
71	Grassland/Herbaceous	0.04	0
82	Cultivated Crops	0.05	0
22	Developed, Low Intensity	0.08	20
95	Emergent Herbaceous Wetlands	0.045	75
90	Woody Wetlands	0.07	50
23	Developed, Medium Intensity	0.12	40
24	Developed, High Intensity	0.15	60
31	Barren Land Rock/Sand/Clay	0.03	0

Figure 2-12. Example Land Cover Layer with Manning's n values and Percent Impervious.

As shown in Figure 2-12, the user must define Manning’s n values for all of the land classification types, including the NoData field. The user entered Manning’s n values will be considered to be the base Manning’s n values for this land cover layer. Additionally the user can define **Percent Impervious** for each Land Cover Classification type. Percent Impervious is only needed in the user intends to use precipitation and infiltration features within HEC-RAS.

User Defined Land Cover Classification Polygons

In addition to establishing Manning’s n values for each of the land cover classifications within the land cover layer, users have to option to override these values by creating user defined **Classification Polygons**. For example, the national based land cover data sets are not well defined for the main channel areas of a river system. Users will need to create their own main channel polygons to ensure they can establish a good set of base Manning’s n values for the entire channel. Typically, a river system will be broken up into many user defined polygons, one polygon for each Manning’s n value region.

To create user defined land cover classification polygons, right click on the **Classification Polygons** layer underneath the Land Cover layer that you want to edit, then select **Edit Layer**. This puts RAS Mapper into edit mode and will allow you to create new or edit existing classification polygons. An example showing a single user defined land cover Classification Polygon, for the "Main Channel", is shown in Figure 2-13. This user defined classification was given a Manning's n value of 0.035 and a percent impervious of 100. Generally, several user defined land cover classifications will be developed for a land cover data layer in order to reclassify main channel areas, roads, parking lots, buildings, or anything else that is not well defined in the base land cover data layer.

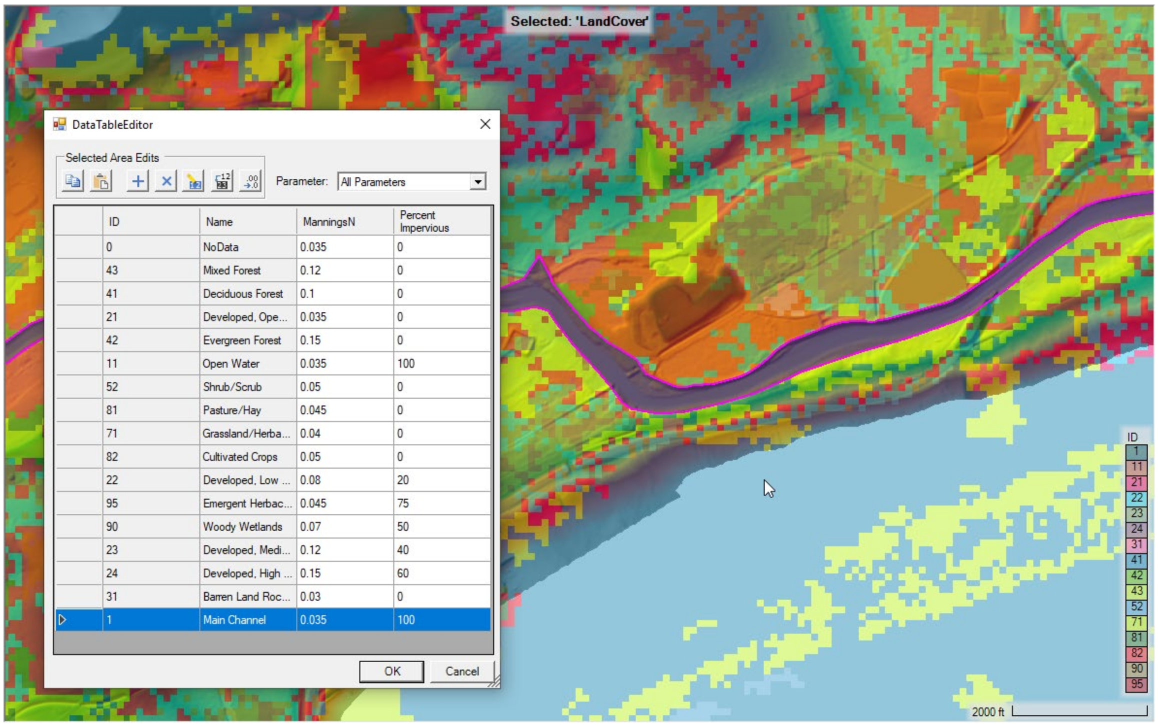


Figure 2-13. Example User Defined Land Cover Classification Polygon.

Associating Land Cover/Manning's n with Geometry Data

Once the user has created a Land Cover layer and added some of their own user defined classification regions, then they need to associate that land cover layer with the geometry file(s) they want to use it with. To associate the Land Cover layer with a geometry file, right click on the **Geometries** layer (on the left side of the RAS Mapper window) and select **Manage Geometry Associations**. This will bring up a window that will allow the user to select the desired layers to associate with each of the Geometry files in the project (Figure 2-14).

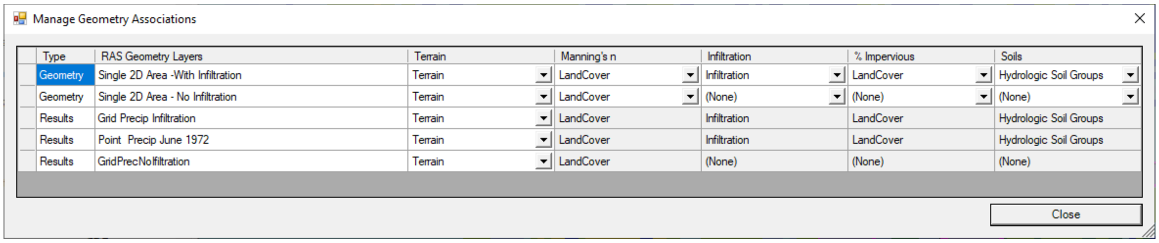


Figure 2-14. Editor for managing all Geometry Associations to Spatial Layers.

As shown in Figure 2-14, the Geometry Associations table allows the user to associate the following information:

Terrain: This must be a Terrain data set created from within HEC-RAS Mapper. This layer association is required.

Manning's n: Land Cover layer containing Manning's n values to use as base Manning's n values for the associated Geometry. **This layer is required** in order to have spatially varying Manning's n values within 2D Flow Areas.

Infiltration: Infiltration Layer containing land cover, soils, and infiltration parameters for one of the available infiltration methods. This layer is Optional depending upon modeling approaches.

% Impervious: Land Cover layer containing % Impervious values to use with associated Geometry. This layer is Optional depending upon modeling approaches.

Soils: Soils Layer to be associated with Geometry. This Layer is Option, and only required for 2D Sediment Transport modeling.

Manning's n Calibration Regions

Once a Land Cover/Manning's n layer has been developed and associated with a specific Geometry data set, the user has the option to create Manning's n Calibration Regions that will only be applied to that geometry data set. In order to calibrate a hydraulic model, it may be necessary to increase or decrease Manning's n values on a reach by reach basis for a specific geometry/event. This can be accomplished by developing Manning's n calibration regions and using those regions to raise or lower all the Manning's n values contained within that region/polygon.

Manning's n Calibration regions/polygons are created within the Manning's n layer of a specific geometry data set. To create Manning's n calibration regions, open HEC-RAS Mapper and expand the Geometry layer to be edited. Then expand the Manning's n layer for that Geometry data set. To create/edit Manning's n Calibration regions, right click on **Calibration Regions** and select **Edit Geometry**. This option allows the user to create new regions or edit existing regions. Once a region is created user can redefine all the Manning's n values within that region for each land cover type. An example of a model with Calibration regions for the main channel areas is shown in Figure 2-15.

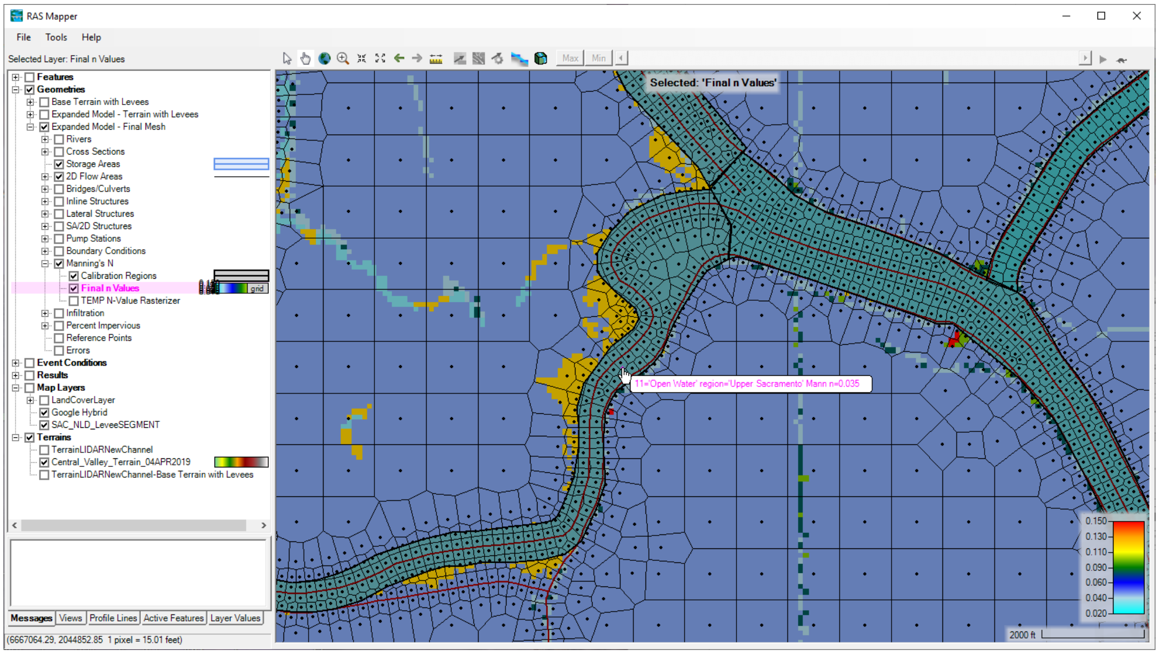


Figure 2-15. Example Land Cover data set and User Defined 2D Area Manning's n value Calibration Regions.

Once the user has drawn the Manning's n Calibration regions for a geometry, the Manning's n values can be edited in a table by right clicking on the **Calibration Regions** layer and selecting **Edit Manning's n Values**. When this is done, a table will appear that allows you to edit the Manning's n values for all the calibration regions (Figure 2-16).

Layer Parameter Values

Selected Area Edits — Show Base Overrides — Region: All Regions

ID	Name	ManningsN	Upper Sacramento - ManningsN	Feather River - ManningsN	Sac Feather-American - ManningsN	Sac Lower - ManningsN	American - ManningsN	NEMDC - ManningsN
0	NoData		NaN	0.035	0.033	0.03	0.038	0.04
31	Barren Land Rock/Sand/Clay	0.04	0.035	0.035	0.033	0.03	0.038	0.04
82	Cultivated Crops	0.06	0.035	0.035	0.033	0.03	0.038	0.04
41	Deciduous Forest	0.1	0.035	0.035	0.033	0.03	0.038	0.04
24	Developed, High Intensity	0.15	0.035	0.035	0.033	0.03	0.038	0.04
22	Developed, Low Intensity	0.08	0.035	0.035	0.033	0.03	0.038	0.04
23	Developed, Medium Intensity	0.1	0.035	0.035	0.033	0.03	0.038	0.04
21	Developed, Open Space	0.04	0.035	0.035	0.033	0.03	0.038	0.04
95	Emergent Herbaceous Wetlan...	0.08	0.035	0.035	0.033	0.03	0.038	0.04
42	Evergreen Forest	0.12	0.035	0.035	0.033	0.03	0.038	0.04
71	Grassland/Herbaceous	0.045	0.035	0.035	0.033	0.03	0.038	0.04
43	Mixed Forest	0.08	0.035	0.035	0.033	0.03	0.038	0.04

OK Cancel

Figure 2-16. Manning's n Calibration Regions Table.

This Manning's n by Land Cover and the Calibration Regions will be used during the 2D flow area pre-processing stage (i.e. the process where the software creates the cell and cell face table properties). In order to get these Manning's n values into the 2D flow area property tables, the 2D flow area Hydraulic Property tables must be recomputed. When the cell faces are processed, the Manning's n value selected will be based on finding the cell face center, then the corresponding Manning's n value from the land cover layer. If there is no Land Cover layer defined for a specific cell face, then the default Manning's n value entered into the 2D flow area editor will be used for that cell face. For this version of HEC-RAS, the program will select only one Manning's n value for the entire cell face. Future versions of HEC-RAS will allow for multiple Manning's n values across each cell face. So, this is a limitation right now when using large cell sizes.

Examples of typical Manning's n value ranges for the various NLCD Land Cover types is shown in Table 2-1 below. These n values are for appreciable depths of flow, and are not meant for shallow overland flow. Shallow, overland flow Manning's n values are generally much higher, due to the relative roughness compared to the flow depth.

Table 2-1. Example Manning's n values for various NLCD Land Cover Types.

NLCD Value	**n Value Range**	**Description**
11	0.025 - 0.05	**Open Water**- areas of open water, generally with less than 25% cover of vegetation or soil. This is for natural streams on mild to moderate slopes.
12	n/a	**Perennial Ice/Snow**- areas characterized by a perennial cover of ice and/or snow, generally greater than 25% of total cover.
21	0.03 - 0.05	**Developed, Open Space**- areas with a mixture of some constructed materials, but mostly vegetation in the form of lawn grasses. Impervious surfaces account for less than 20% of total cover. These areas most commonly include large-lot single-family housing units, parks, golf courses, and vegetation planted in developed settings for recreation, erosion control, or aesthetic purposes.
22	0.06 - 0.12	**Developed, Low Intensity**- areas with a mixture of constructed materials and vegetation. Impervious surfaces account for 20% to 49% percent of total cover. These areas most commonly include single-family housing units.
23	0.08 - 0.16	**Developed, Medium Intensity** -areas with a mixture of constructed materials and vegetation. Impervious surfaces account for 50% to 79% of the total cover. These areas most commonly include single-family housing units.
24	0.12 - 0.20	**Developed High Intensity**-highly developed areas where people reside or work in high numbers. Examples include apartment complexes, row houses and commercial/industrial. Impervious surfaces account for 80% to 100% of the total cover.
31	0.023 - 0.030	**Barren Land (Rock/Sand/Clay)** - areas of bedrock, desert pavement, scarps, talus, slides, volcanic material, glacial debris, sand dunes, strip mines, gravel pits and other accumulations of earthen material. Generally, vegetation accounts for less than 15% of total cover.
41	0.10 - 0.20	**Deciduous Forest**- areas dominated by trees generally greater than 5 meters tall, and greater than 20% of total vegetation cover. More than 75% of the tree species shed foliage simultaneously in response to seasonal change.

42	0.08 - 0.16	**Evergreen Forest**- areas dominated by trees generally greater than 5 meters tall, and greater than 20% of total vegetation cover. More than 75% of the tree species maintain their leaves all year. Canopy is never without green foliage.
43	0.08 - 0.20	**Mixed Forest**- areas dominated by trees generally greater than 5 meters tall, and greater than 20% of total vegetation cover. Neither deciduous nor evergreen species are greater than 75% of total tree cover.
51	0.025 - 0.05	**Dwarf Scrub**- Alaska only areas dominated by shrubs less than 20 centimeters tall with shrub canopy typically greater than 20% of total vegetation. This type is often co-associated with grasses, sedges, herbs, and non-vascular vegetation.
52	0.07 - 0.16	**Shrub/Scrub**- areas dominated by shrubs; less than 5 meters tall with shrub canopy typically greater than 20% of total vegetation. This class includes true shrubs, young trees in an early successional stage or trees stunted from environmental conditions.
71	0.025 - 0.50	**Grassland/Herbaceous**- areas dominated by gramanoid or herbaceous vegetation, generally greater than 80% of total vegetation. These areas are not subject to intensive management such as tilling, but can be utilized for grazing.
72	0.025 - 0.50	**Sedge/Herbaceous**- Alaska only areas dominated by sedges and forbs, generally greater than 80% of total vegetation. This type can occur with significant other grasses or other grass like plants, and includes sedge tundra, and sedge tussock tundra.
73	n/a	**Lichens**- Alaska only areas dominated by fruticose or foliose lichens generally greater than 80% of total vegetation.
74	n/a	**Moss**- Alaska only areas dominated by mosses, generally greater than 80% of total vegetation.
81	0.025 - 0.50	**Pasture/Hay**-areas of grasses, legumes, or grass-legume mixtures planted for livestock grazing or the production of seed or hay crops, typically on a perennial cycle. Pasture/hay vegetation accounts for greater than 20% of total vegetation.
82	0.020 - 0.50	**Cultivated Crops** -areas used for the production of annual crops, such as corn, soybeans, vegetables, tobacco, and cotton, and also perennial woody crops such as orchards and vineyards. Crop vegetation accounts for greater than 20% of total vegetation. This class also includes all land being actively tilled.
90	0.045 - 0.15	**Woody Wetlands**- areas where forest or shrubland vegetation accounts for greater than 20% of vegetative cover and the soil or substrate is periodically saturated with or covered with water.
95	0.05 - 0.085	**Emergent Herbaceous Wetlands**- Areas where perennial herbaceous vegetation accounts for greater than 80% of vegetative cover and the soil or substrate is periodically saturated with or covered with water.

Creating a Soils Data Layer

The use of soils data in HEC-RAS is optional. Soils data can be used to define infiltration parameters for rainfall runoff modeling within HEC-RAS. Soils data can also be used to define spatial sediment transport data for 2D sediment transport modeling (see the Sediment Transport user's manual for information on soils data).

Importing Data from a Shapefile

Soils data information can be quite complex. Vector (shapefile) data from the SSURGO database can be downloaded from the NRCS:

https://websoilsurvey.sc.egov.usda.gov/App/WebSoilSurvey.aspx

The data comes as a shapefile with an abbreviated soils name and unique key. Numerous tables are also included, however, to use the tabular data you will need to have an understanding of the tables (metadata and table columns), information can be accessed here:
https://www.nrcs.usda.gov/wps/portal/nrcs/detail/soils/survey/geo/?cid=nrcs142p2_05363

The user will need to join the data from the tables using other software like a GIS. Data are also available in a geodatabase format from the GSSURGO database:

https://datagateway.nrcs.usda.gov/GDGOrder.aspx

RAS Mapper has the capability to import the GSSURGO data.

Creating a Soils Layer is similar to the Land Cover Layer. Select the **Map Layers | Create New RAS Layer | Soils Layer** to invoke the Create New Soils Layer dialog (Figure 2-17). If you are adding soils in shapefile format, select the field name that provides a unique name convention for the soils layer, such as "Hydrologic Group" or "Texture Class". Adding soils data from a standard coordinate system will take a moment to load the data as all of the polygons must be reprojected to the coordinate system used for the project. If using the GSSURGO geodatabase (discussed below), other field choices will be provided for you.

Pressing the Create button will create a “Soils” grid ("Soils.hdf" and "Soils.tif") will be created, using the specified Cell Size.

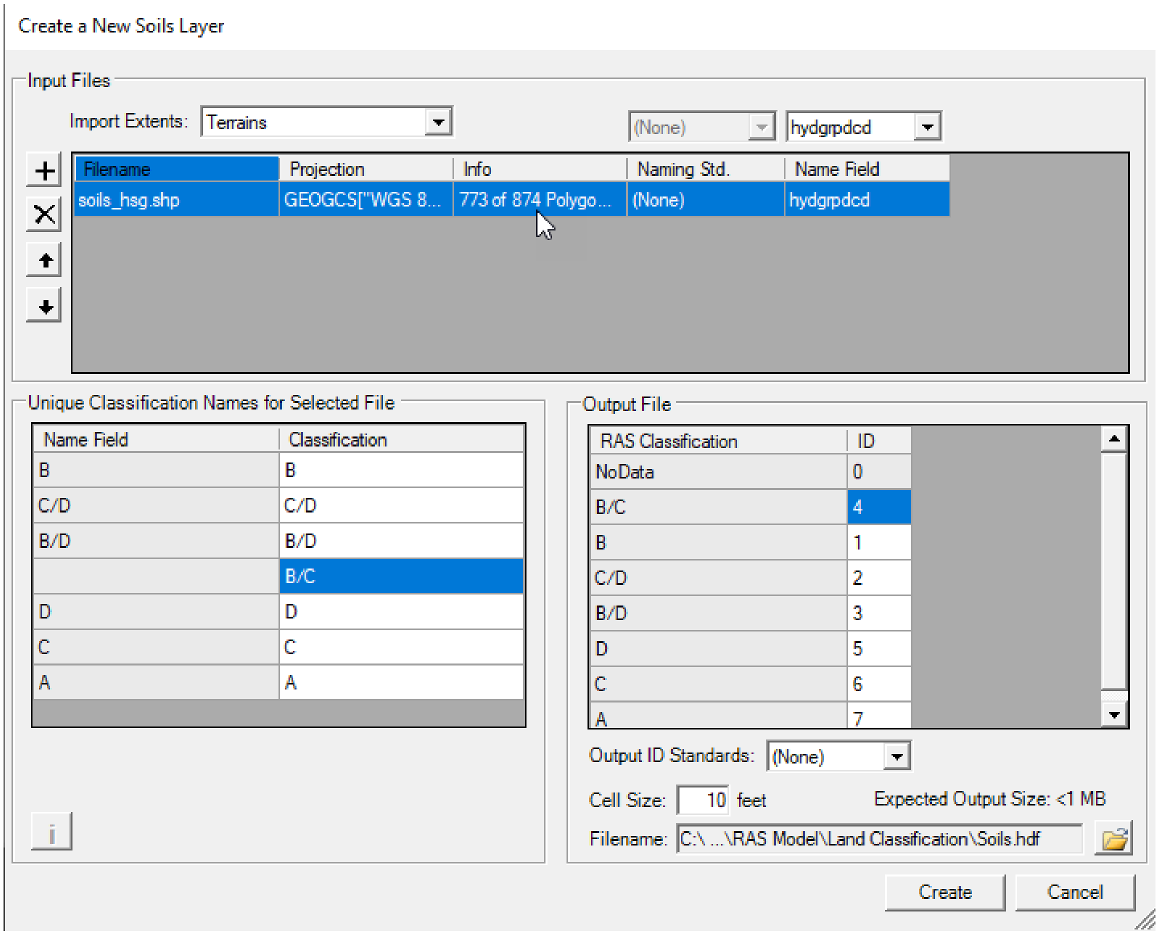


Figure 2-17. Example of Importing Soils Data from a Shapefile.

Importing GSSURGO Data

If you are adding GSSURGO data, you will need to select the "GSSURGO" name from the file type list. Navigate to the geodatabase (which is a folder). Pick a file inside of the database ("gdb" is a good choice), choose the **Open** button. RAS Mapper will recognize and interpret the database, preparing it for import.

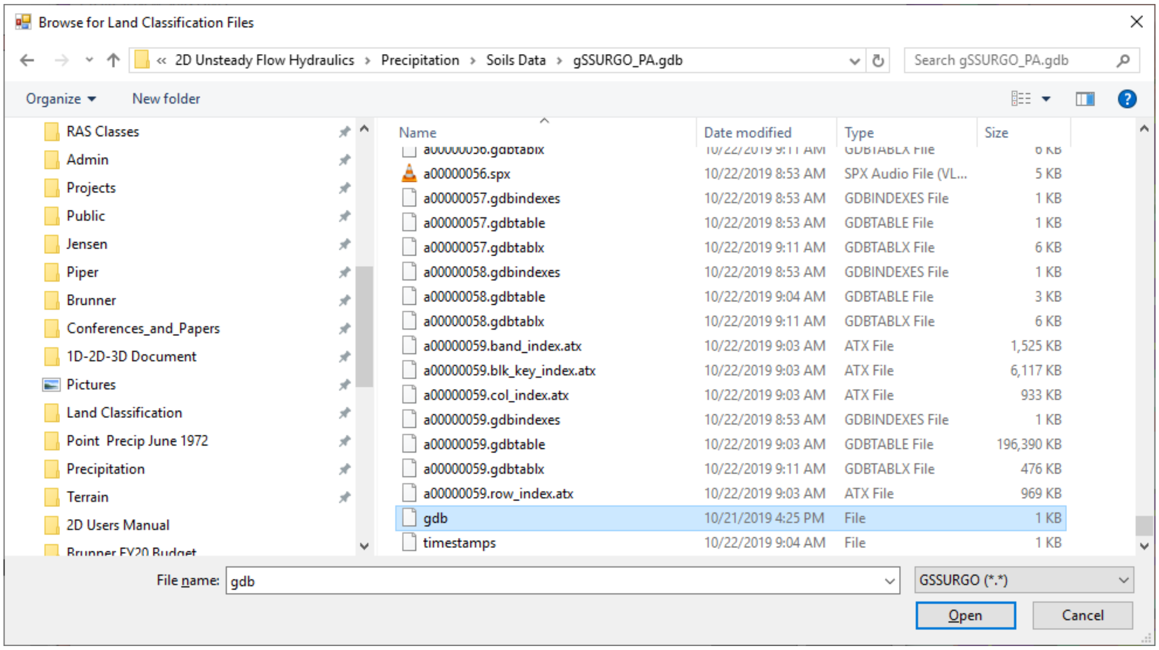


Figure 2-18. Selecting the GSSURGO Database File.

The data will then be imported, converted from the geodatabase to raster and projected to the RAS Mapper coordinate system. As the data are loaded, the user will be informed with a progress bar.

By default, RAS Mapper will choose to bring the data in classified on the "Hydrologic Group" field. Other options include the "Texture Group", "Map Unit Key", "Map Unit Symbol", or a the combination of hydrologic group and texture group "Merged Hydr : Texture". Select the **Name Field** to classify the data. It is likely that some of the GSSURGO data will not be classified; in this case, RAS Mapper will classify those areas with the "none" keyword. It is the user's responsibility to provide a classification for that data.

Verify the output file **Cell Size** and **Filename**. Press **Create** to create the soils layer.

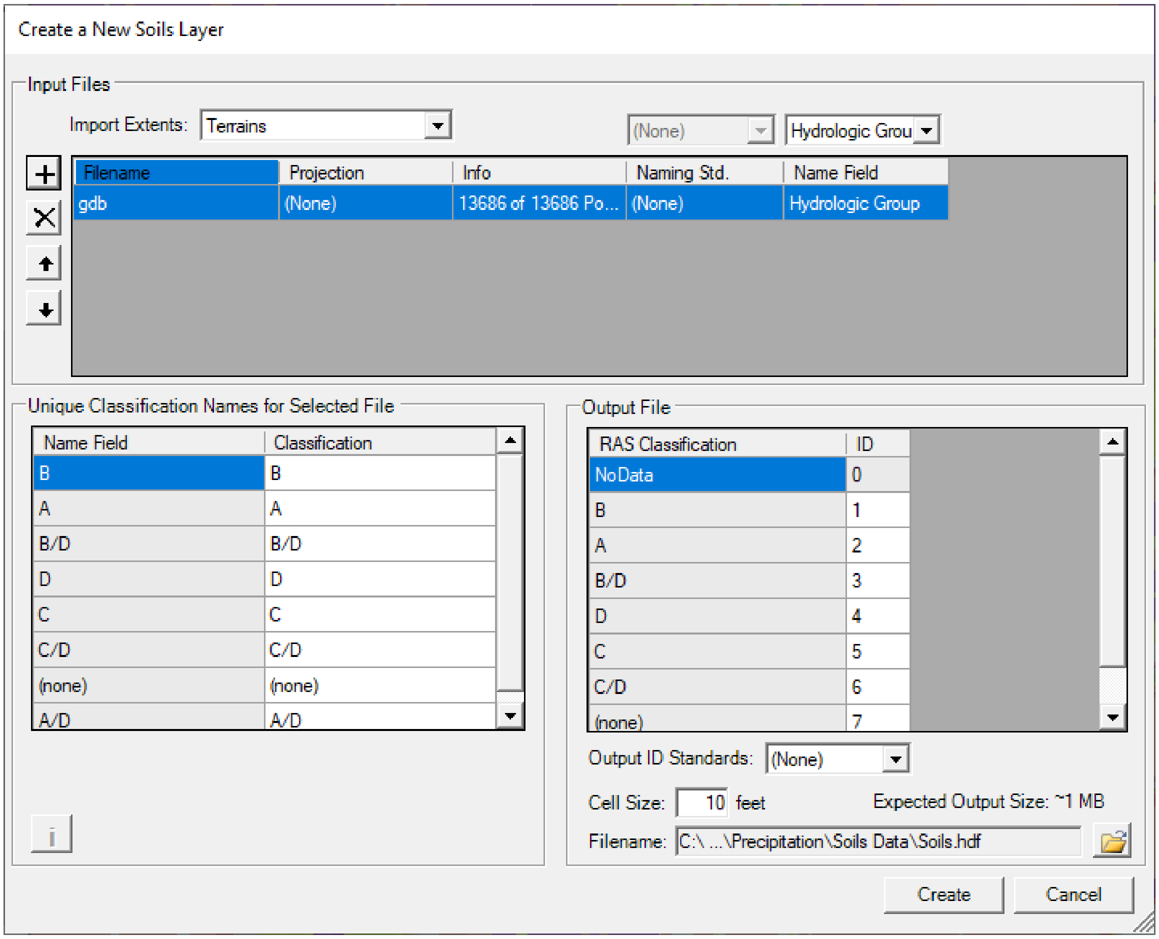


Figure 2-19. Example of Importing Soils Data from a GSSURGO Database file.

Creating an Infiltration Layer

RAS Mapper allows for the creation of an Infiltration Layer. An Infiltration Layer defines the Infiltration Method (Deficit Constant, SCS Curve Number, or Green and Ampt) that will be used for surface losses from a precipitation event. There are two different ways to generate an Infiltration Layer in RAS Mapper to represent infiltration parameters: (1) using and existing RAS Land Cover layer or RAS Soils layer or the intersection of a Land Cover layer and Soils layer; or (2) to use a single classification shapefile layer to define the infiltration parameters. The approach used to create an infiltration layer for parameterization will depend on the selected infiltration method and data available.

Using a Land Cover and Soils Layer

This will create a complex table which cross reference the Land Cover and Soils classifications from which you can parameterize the infiltration data. To create a complex Infiltration layer, select **Map Layers | Create New RAS Layer | Infiltration Layer from Land Cover / Soils Layers** menu item. A dialog will be provided to allow the user to select the Infiltration Method, Land Cover Layer, and Soils Layer and provide the new Infiltration Layer name and directory. Note, that you do not need to specify both a Land Cover layer and a Soils Layer, this is a user option. However, for the SCS Curve Number Method, that is the correct approach. See Figure 2-20 below.

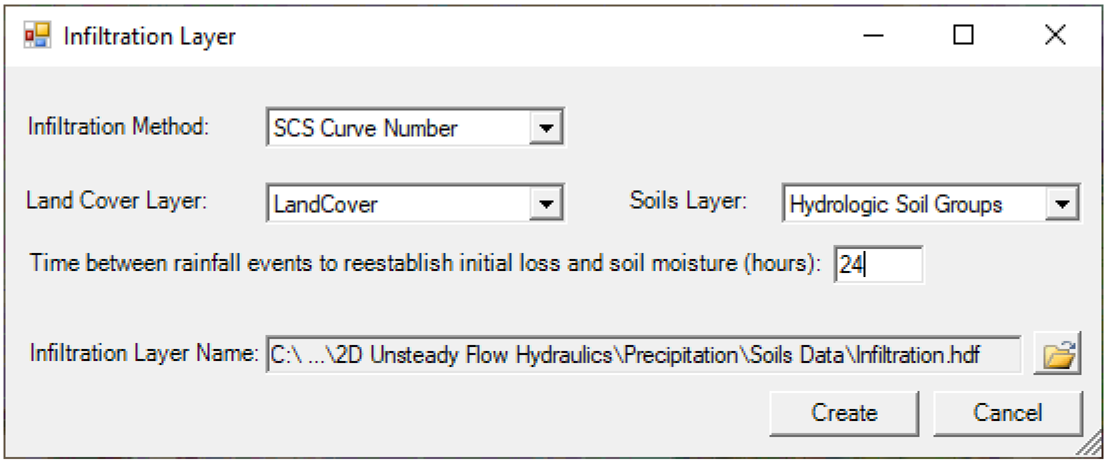


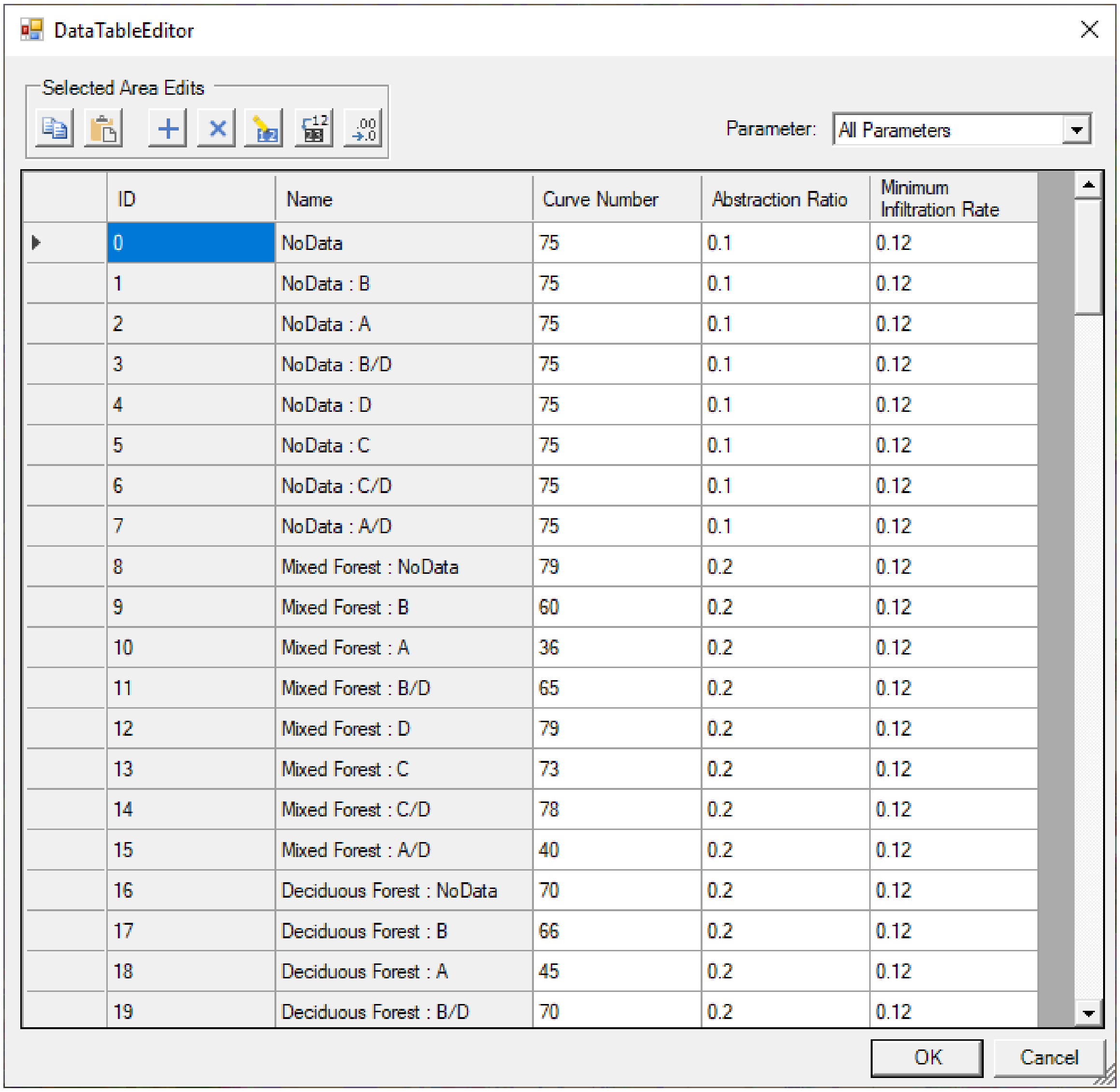
Figure 2-20. Example of Creating an Infiltration Layer from Land Cover and Soils Data.

Press the **Create** button to create a new raster layer that is the intersection of the land cover and soils layers. As shown in Figure 2-21 below, the intersection of the layers can become quite complex, creating a table with rows for each overlapping classification from the base layers. Depending on the infiltration method selection, additional columns for the required parameters will be provided. It is up to the user to fill out this table with an estimate of each of the parameters, based on the Land Cover and the Soils data being used. In the example shown in Figure 2-21, the user is required to enter an SCS curve Number and an Abstraction Ratio (This defines the initial losses). The Minimum Infiltration Rate field is optional. Use of the Minimum Infiltration Rate will not allow the infiltration rate to be lower than the user entered value, even when the soil becomes saturated. This is not a standard feature of the SCS Curve number method, but an option added for HEC-RAS.

NOTE: If the User has specified Percent Impervious Data within the Land Cover layer that will be attached to the same Geometry data being used in conjunction with an Infiltration Layer, then the Curve Number values in this Infiltration Layer table should only be representative of the pervious land cover portion of all land cover types. Otherwise the user will be double accounting the Impervious area in the model run. User entered Impervious Area percentages are treated separately and assumed to have no losses (i.e. CN = 100). This approach is the preferred

method, as it is more physically based. Runoff will occur immediately when precipitation covers any impervious surfaces.

If the User has not entered in any Percent Impervious data within the Land Cover data Layer, then the CN values in this table should reflect both pervious and impervious land surfaces within each Land Cover Layer.



ID	Name	Curve Number	Abstraction Ratio	Minimum Infiltration Rate
0	NoData	75	0.1	0.12
1	NoData : B	75	0.1	0.12
2	NoData : A	75	0.1	0.12
3	NoData : B/D	75	0.1	0.12
4	NoData : D	75	0.1	0.12
5	NoData : C	75	0.1	0.12
6	NoData : C/D	75	0.1	0.12
7	NoData : A/D	75	0.1	0.12
8	Mixed Forest : NoData	79	0.2	0.12
9	Mixed Forest : B	60	0.2	0.12
10	Mixed Forest : A	36	0.2	0.12
11	Mixed Forest : B/D	65	0.2	0.12
12	Mixed Forest : D	79	0.2	0.12
13	Mixed Forest : C	73	0.2	0.12
14	Mixed Forest : C/D	78	0.2	0.12
15	Mixed Forest : A/D	40	0.2	0.12
16	Deciduous Forest : NoData	70	0.2	0.12
17	Deciduous Forest : B	66	0.2	0.12
18	Deciduous Forest : A	45	0.2	0.12
19	Deciduous Forest : B/D	70	0.2	0.12

Figure 2-21. Example Infiltration Layer using the SCS Curve Number Method.

Using a Classification Shapefile Layer

To use a single classification layer, select **Map Layers | Create New RAS Layer | Infiltration Layer from Shapefile | Method** menu item. The Create New Infiltration Layer dialog will come up allowing you to choose the shapefile of interest. Once the shapefile has been selected, select the unique classification name (like "Hydrologic Soils Group") to import the data, as shown in the figure below.

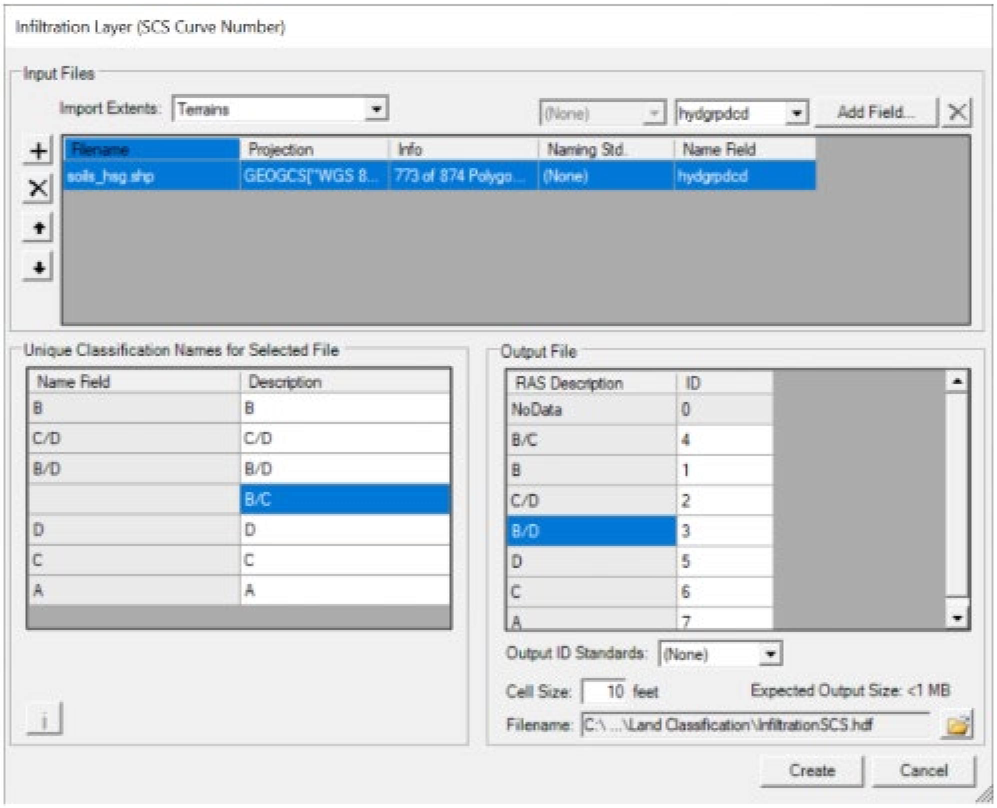


Figure 2-22. Example of Importing Infiltration Data from a Shapefile.

Once the data has been imported, you will need to enter infiltration parameters by right-clicking on the Infiltration Layer and selecting Edit Infiltration Data. A table will be provided with the infiltration parameter based on the Infiltration Method (Deficit Constant, SCS Curve Number, or Green and Ampt) selected.

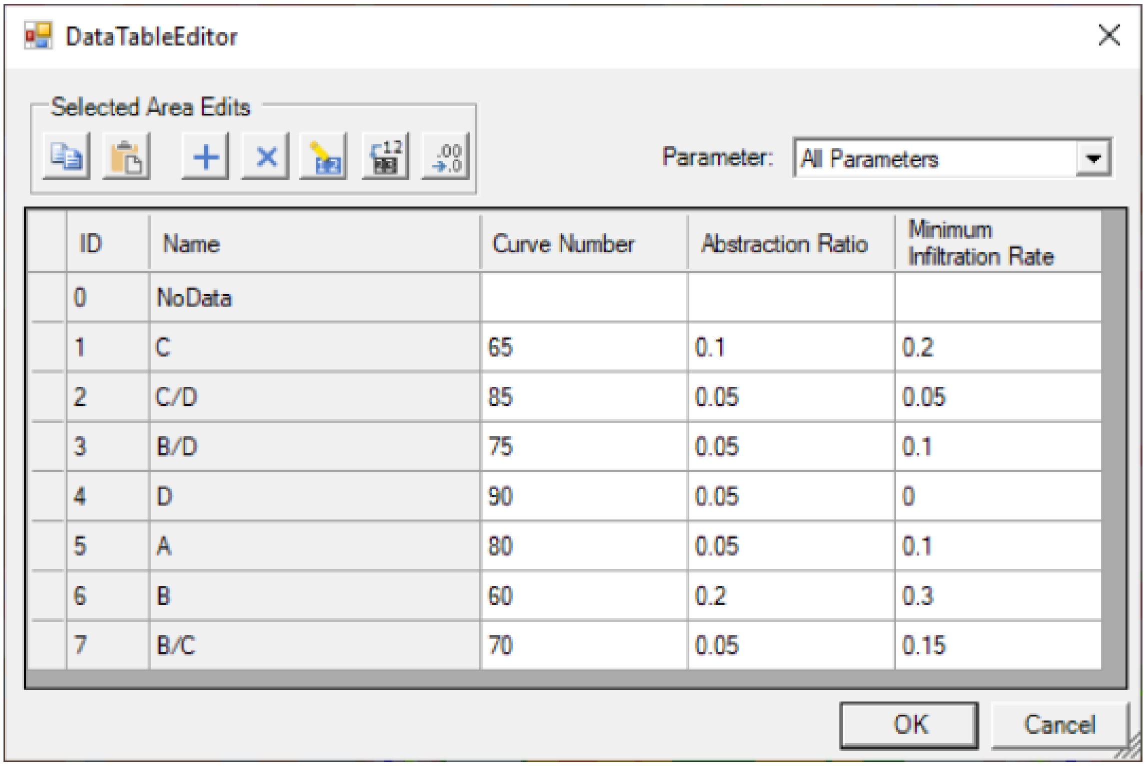


Figure 2-23. Example of Imported SCS Curve Number Infiltration data from a Shapefile.

Infiltration Methods

SCS Curve Number

The Curve Number (CN) method is an empirical surface runoff method developed by the US Department of Agriculture (USDA) Natural Resources Conservation Service (NRCS) while it was formerly called the Soil Conservation Service (SCS) (SCS 1985). The SCS CN method estimates precipitation excess as a function of the cumulative precipitation, soil cover, land use, and antecedent soil moisture. The input requirements for the SCS CN method within HEC-RAS are:

1. Curve Number CN (scalar value for each cell)
2. Initial abstraction ratio $r = I_a / S$ [-] (scalar value for each cell)
3. Length of recovery period (only when utilizing the recovery method) (optional value for each 2D area, hrs)
4. Minimum infiltration rate (optional value for each cel) [in/hr or cm/hr]

The curve number CN values range from approximately 30 for permeable soils with high infiltration rates to 100 for water bodies, impervious surfaces, and soils with near zero infiltration rates. Publications from the Soil Conservation Service (1971, 1986) provide further background and details on use of the CN model. The initial abstraction may be estimated as a function of the potential maximum retention as

$$I_a = rS \tag{2-1}$$

where r is the user-defined initial abstraction ratio, typically ranging between 0.05 and 0.2. The potential maximum soil retention S is computed from the runoff curve number CN as

$$S = \frac{1000}{CN} - 10 \tag{2-2}$$

where S is in inches.

One thing to keep in mind, the curve number method was not originally developed for simulating historic events. The same loss amount will be computed for a rainfall of 5 inches regardless of whether it occurred in 1 hour or 1 day. However, the method has been adapted to single events.

The curve number is related to soil type, soil infiltration capability, land use, and the depth of the water table. The NRCS has divided soils into four hydrologic soil groups (HSGs) according to the ability to infiltrate. The groups are defined as follows:

- **Group A**: Soils with high infiltration rates (low runoff potential) even when thoroughly wetted. These consist chiefly of deep, well-drained sands and gravels. These soils have a final infiltration rates greater than 0.30 in/hr (7.6 mm/hr).

- **Group B**: Soils with moderate infiltration rates when thoroughly wetted. These consist mostly of soils that are moderately deep to deep, moderately well drained to well drained with moderately fine to moderately coarse soil textures. These soils have final infiltration rates of 0.15–0.30 in/hr (3.8–7.6 mm/hr).

- **Group C**: Soils with slow infiltration rates when thoroughly wetted. These consist chiefly of soils with a layer that impedes downward movement of water or soils with moderately fine to fine textures. These soils have final infiltration rates of 0.05–0.15 in/hr (1.3–3.8 mm/hr).

- **Group D**: Soils with very slow infiltration rates (high runoff potential) when thoroughly wetted. These consist chiefly of clay soils with a high swelling potential, soils with a permanent high-water table, soils with a claypan or clay layer at or near the surface, and shallow soils over nearly impervious materials. These soils have final infiltration rates of less than 0.05 in/hr (1.3 mm/hr).

Selection of a hydrologic soil group should be done based on measured infiltration rates, soil survey, or judgment from a qualified soil scientist or geotechnical professional. The table below presents curve numbers for various hydrologic soil-cover complexes.

Table 2-2. Runoff Curve Numbers for Hydrologic Soil-Cover Complexes

Cover			Hydrologic Soil Group			
Land Use	**Treatment or Practice**	**Hydrologic Condition**	**A**	**B**	**C**	**D**
Fallows	Straight row		77	86	91	94
Row crops	Straight row	Poor	72	81	88	91
		Good	67	78	85	89
	Contoured	Poor	70	79	84	88
		Good	65	75	82	86
	Contoured and terraced	Poor	66	74	80	82
		Good	62	71	78	81
Small grain	Straight row	Poor	65	76	84	88
		Good	63	75	83	87
	Contoured	Poor	63	74	82	85
		Good	61	73	81	84
	Contoured and terraced	Poor	61	72	79	82
		Good	59	70	78	81
Close-seeded legumes or rotation meadow	Straight row	Poor	66	77	85	89
		Good	58	72	81	85
	Contoured	Poor	64	75	83	85
		Good	55	69	78	83
	Contoured and terraced	Poor	63	73	80	83
		Good	51	67	76	80
Pasture or range	Contoured	Poor	68	79	86	89
		Fair	49	69	79	84
		Good	39	61	74	80
Meadow		Good	30	58	71	78
Woods		Poor	45	66	77	83
		Fair	36	60	73	79
		Good	25	55	70	77
Farmsteads			59	74	82	86
Roads (dirt)			72	82	87	89
Road (hard surface)			74	84	90	92

The NRCS has also developed Runoff Curve Numbers of other land cover types, including Urban Areas. However, some of the CN values shown in their Urban Area tables include the percent impervious area already in the development of the Curve Number. Here is the NRCS table of CN values for Urban Areas:

Table 2-3. Runoff Curve Number for Urban Areas

Cover			**Hydrologic Soil Group**			
Land Use	**Cover Description**	**Hydrologic Condition**	**A**	**B**	**C**	**D**
Open space	(lawns, parks, golf courses, cemeteries, etc.)	Poor (<50% grass)	68	79	86	89
		Fair (50% to 75% grass)	49	69	79	84
		Good (>75% grass)	39	61	74	80
Impervious areas:	Paved.	parking lots, roofs, driveways, etc	98	98	98	98
	Streets and Roads	Paved; curbs and storm sewers	98	98	98	98
		Paved; open ditches	83	89	92	93
		Gravel	76	85	89	91
		Dirt	72	82	87	89
Western Desert Urban Areas	Natural Desert	Pervious areas only	63	77	85	88
	Artificial Desert	Impervious weed barrier	96	96	96	96
Urban Districts	Commercial/Buisiness	85 % Impervious	89	92	94	95
	Industrial	72 % Impervious	81	88	91	93
Residential Districts by Average Lot Size	1/8 acre (town house)	65 % Impervious	77	85	90	92
	1/4 acre	38 % Impervious	61	75	83	87
	1/3 acre	30 % Impervious	57	72	81	86
	1/2 acre	25 % Impervious	54	70	80	85
	1 acre	20 % Impervious	51	68	79	84
	2 acre	12 % Impervious	46	65	77	82
Developing Urban Areas	Newly Graded Areas Pervious areas only No vegetation		77	86	91	94

In Practice, it is much more accurate to develop Runoff Curve Numbers for the pervious area only and define the impervious area separately. In HEC-RAS, impervious area will be treated as 100% runoff with no infiltration. This is especially important for urban areas, where runoff will occur at the very beginning of storms due to impervious areas that are directly connected to the storm runoff system.

The recovery method for the SCS CN consists of setting the cumulative rainfall depth to zero (i.e. $P = 0$) after a user-specified time in which the infiltration is zero.

The SCS CN model outputs the following variables:

1. Cumulative excess [in or cm] (per cell)
2. Cumulative precipitation [in or cm] (per cell)
3. Infiltration rate [in/hr or cm/hr] (per cell)
4. Infiltration depth [in or cm] (per cell)
5. Dry time [hrs] (per cell)

Deficit and Constant

The soil moisture deficit is defined here as the depth of rainfall needed to bring the soil up to field capacity. When the soil is not saturated all of the rainfall will infiltrate until the soil is saturated. This assumption can lead to unreasonably high infiltration rates. The input parameters for the Deficit-Constant method are:

1. Maximum deficit [in or mm] (scalar value for each cell)
2. Initial deficit [in or mm] (scalar value for each cell)
3. Potential evapotranspiration [in/hr or mm/hr] (scalar time-series for each cell)
4. Potential percolation rate [in/hr or mm/hr] (scalar value for each cell)

The following table list the SCS soil groups and typical potential percolation rates:

Table 2-4. SCS Soil Groups and filtration rates (SCS, 1986; Skaggs and Khaleel 1982).

SCS Soil Group	Description	Range of Loss Rates (in/hr)
A	Deep sand, deep loess, aggregated silts	0.3 - 0.45
B	Shallow loess, sandy lam	0.15 - 0.30
C	Clay loams, shallow sandy loam, soils low in organic content, and soils usually high in clay	0.05 - 0.15
D	Soils that swell significantly when wet, heavy plastic clays, and certain saline soils	0.00 – 0.05

The Deficit and Constant model outputs the following variables:

1. Cumulative excess [in or mm] (per cell)
2. Cumulative precipitation [in or mm] (per cell)
3. Infiltration rate [in/hr or mm/hr] (per cell)
4. Infiltration depth [in or mm] (per cell)
5. Soil moisture deficit [in or mm] (per cell)
6. Percolation rate [in/hr or mm/hr] (per cell)
7. Percolation depth [in/hr or mm/hr] (per cell)
8. Evapotranspiration rate [in/hr or mm/hr] (per cell)

Green-Ampt

Green and Ampt (1911) presented a physics-based approach to computing soil infiltration. They use the following simplification of infiltration.

For the first implementation of Green-Ampt in RAS a simple approach will be used for recovery. The approach assumes that the upper layer of the soil remains saturated even during a hiatus period and solves a simple water balance equation to compute the cumulative infiltration depth. Although simplified, the approach has several important characteristics. Specifically, it is volume conservative and represents soil infiltration evapotranspiration and unsaturated gravity driven flow.

The input data required for the GA method are:

1. Wetting front suction [in or mm] (scalar value for each cell)
2. Saturated hydraulic conductivity [in/hr or mm/hr] (scalar value for each cell)
3. Initial soil water content [-] (scalar value for each cell)
4. Saturated soil water content [-] (scalar value for each cell)
5. Soil Potential Evapotranspiration [in/hr or mm/hr] (time series for each cell)

The initial water content should be set to a value close to the wilting point water content if there is vegetation or the field capacity otherwise. The soil field capacity is the amount of water content remaining after the excess water has been drained away. The wilting point is the minimum amount of water content in the soil that the plant requires not to wilt. The physical definition of the wilting point is the water content at −1,500 kPa of suction pressure, or negative hydraulic head (Weil and Brady 2016). In general, the wilting point is less than the field capacity.

In order to utilize the Green-Ampt with Redistribution (GAR) the following additional input parameters are required:

1. Residual soil water content [in or cm] (scalar value for each cell)
2. Pore-size distribution index [in or cm] (scalar value for each cell)

Table 2-5. Green-Ampt Parameter Estimates and Ranges based on Soil Texture (from Gowdish and Muñoz-Carpena 2009; Rawls and Brakensiek 1982 and Rawls et al. 1982).

Soil Texture	Residual Water Content (-)	Wilting Point (-)	Field Capacity (-)	Total Porosity (-)	Pore-size Distribution Index (-)	Saturated Hydraulic Conductivity (mm/hr)	Wetting Front Suction (mm)
Sand	0.02	0.033	0.048	0.437	0.694	210 – 235.6	96.2 – 106
Loamy sand	0.035	0.055	0.084	0.437	0.553	59.8 – 61.1	119.6 – 142
Sandy loam	0.041	0.095	0.155	0.453	0.378	21.8 – 25.9	215.3 – 222
Loam	0.027	0.117	0.20	0.463	0.252	13.2	175.0 – 315
Silt loam	0.015	0.133	0.261	0.501	0.234	6.8	329.6 – 404
Sandy clay loam	0.068	0.148	0.187	0.398	0.319	3.0 – 4.3	449 – 538.3
Clay loam	0.075	0.197	0.245	0.464	0.242	2.0 – 2.3	408.9 – 446
Silty clay loam	0.040	0.208	0.30	0.471	0.177	1.5 – 2.0	538.3 – 581
Sandy clay	0.109	0.239	0.232	0.430	0.223	1.2	466.5 – 636
Silty clay	0.056	0.250	0.317	0.479	0.150	0.9 – 1.0	577.7 – 647
Clay	0.09.	0.272	0.296	0.475	0.165	0.6	622.5 – 714

CHAPTER 3

Development of a 2D or Combined 1D/2D Model

Development of the 2D Computational Mesh

The HEC-RAS 2D modeling capability uses a Finite-Volume solution scheme. This algorithm was developed to allow for the use of a structured or unstructured computational mesh. This means that the computational mesh can be a mixture of 3-sided, 4-sided, 5-sided, etc… computational cells (HEC-RAS has a maximum of 8 sides in a computational cell). However, the user will most likely select a nominal grid resolution to use (e.g. 200 x 200 ft cells), and the automated tools within HEC-RAS will build the computational mesh. After the initial mesh is built, the user can refine the grid with break lines, refinement regions, and the mesh editing tools. A 2D computational mesh is developed in HEC-RAS Mapper by doing the following:

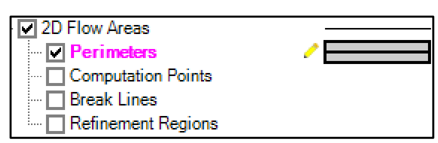
Drawing a Polygon Boundary for the 2D Area

The user must add a 2D flow area polygon to represent the boundary of the 2D area using the geometry editing tools in HEC-RAS Mapper. The best way to do this in HEC-RAS is to first bring in terrain data and aerial imagery into HEC-RAS Mapper. Additionally, the user may want to bring in a shapefile that represents a protected area, if they are working with a leveed system. The terrain and background images will assist the user in figuring out where to draw the 2D flow area boundaries.

Note: The boundary between a 1D river reach and a 2D flow area should be high ground that separates the two. For levees and roadways this is obviously the centerline of the levee and the roadway. However, when using a lateral structure to connect a 1D main river to a 2D floodplain (when there is no actual levee), try to find the high ground that separates the main river from the floodplain. Use this high ground as a guide for drawing the 2D boundary, as well as defining the Lateral Structure Station Elevation data.

To create the 2D flow area in HEC-RAS Mapper, do the following:

1. Right click on the Geometry Layer that you want to edit, and select **Edit Geometry**.



2. Expand the **2D Flow Areas** layer and select the **Perimeters** layer.

3. Draw the perimeter using the **Add New Feature**  tool.

Zoom in to the point at which you can see with great detail, where to draw the boundary of the 2D Flow Area. Begin by left-clicking to drop a point along the 2D flow area polygon boundary. Then continue to use the left mouse button to drop points in the 2D flow area boundary. As the user runs out of screen real-estate, they can **right-click** to re-center the screen, this will give you more area to continue drawing the 2D flow area boundary. Double-click the left mouse button to finish creating the polygon. Once the 2D area polygon is finished, the interface will ask the user for a Name to identify the 2D flow area. Shown in Figure 3-1 is an example 2D flow area polygon for an area that is protected by a levee. The name given to the 2D flow area in this example is: "**2D Interior Area**".

Note: A 2D flow area must be drawn within the limits of the terrain model area being used for the study.

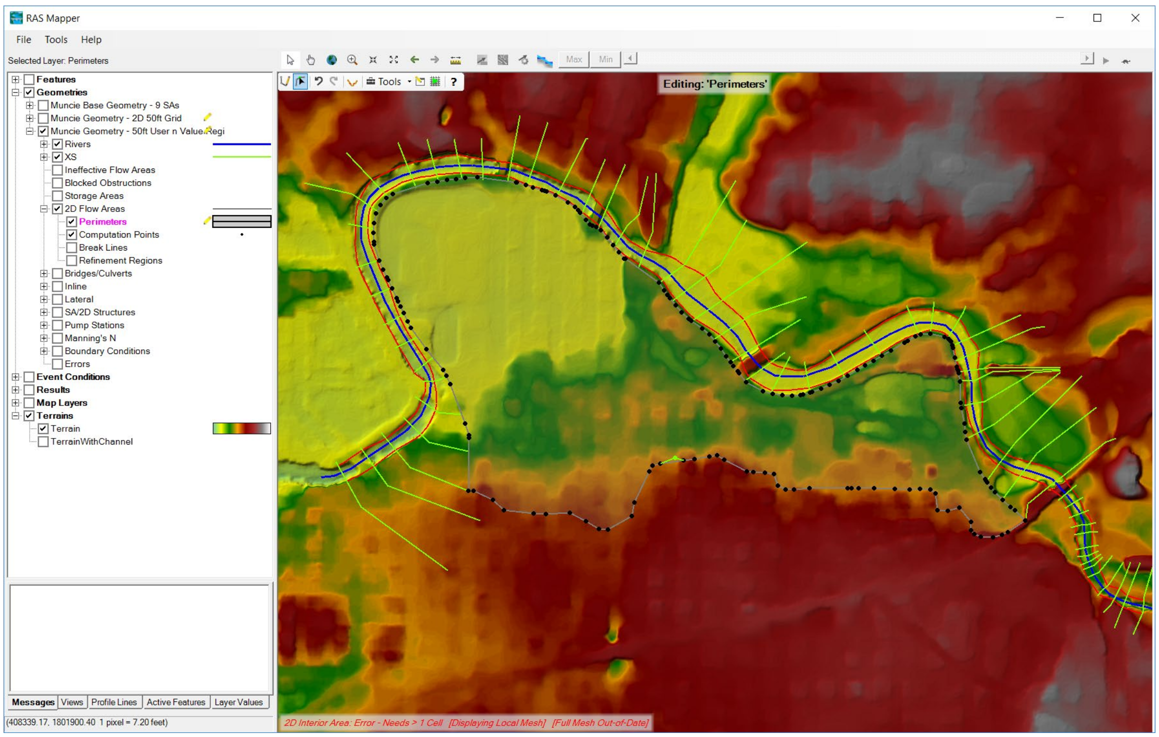


Figure 3-1. Example 2D flow area polygon.

Create the 2D Computational Mesh

The HEC-RAS terminology for describing the computational mesh for 2D modeling begins with the 2D flow area. The 2D flow area defines the boundary for which 2D computations will occur. A computational mesh (or computational grid) is created within the 2D flow area. Each cell within the computational mesh has the following three properties (Figure 3-2).

Cell Center: The computational center of the cell. This is where the water surface elevation is computed for the cell. The cell center does not necessarily correspond to the exact cell centroid.

Cell Faces: These are the cell boundary faces. Faces are generally straight lines, but they can also be multi-point lines, such as the outer boundary of the 2D flow area.

Cell Face Points: The cell Face Points (FP) are the ends of the cell faces. The Face Point (FP) numbers for the outer boundary of the 2D flow area are used to hook the 2D flow area to a 1D elements and boundary conditions.

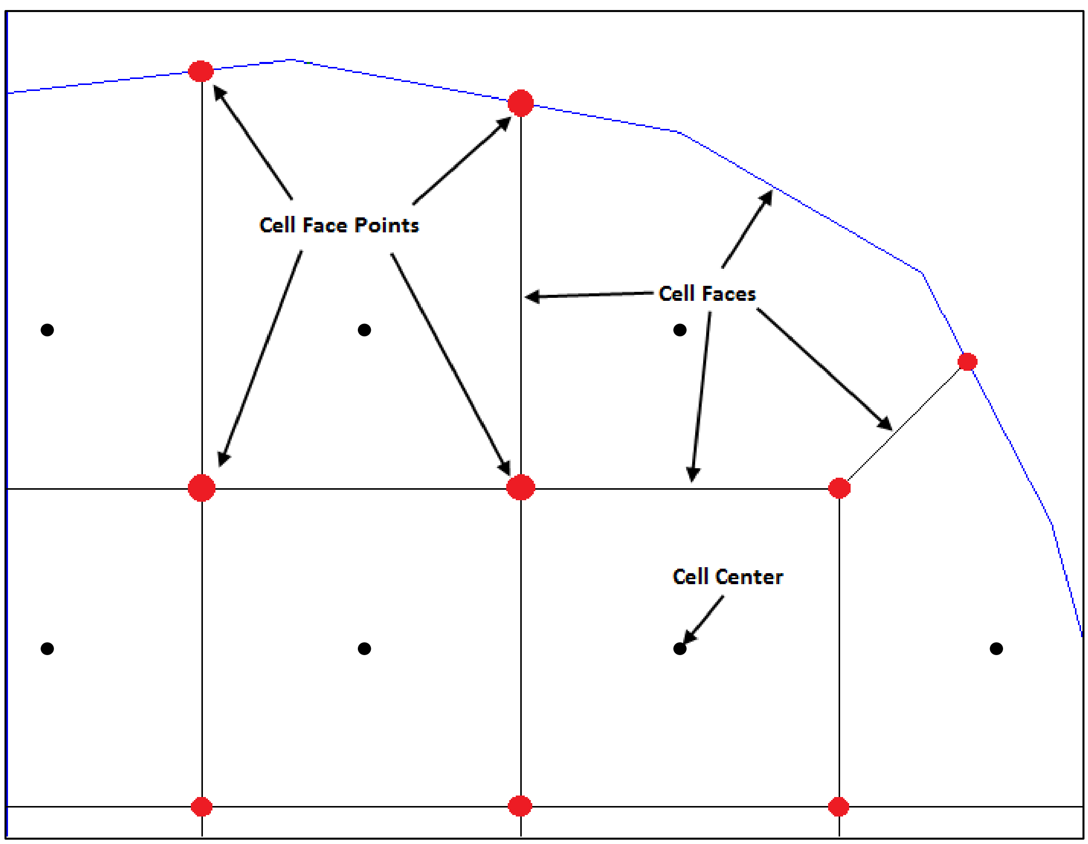


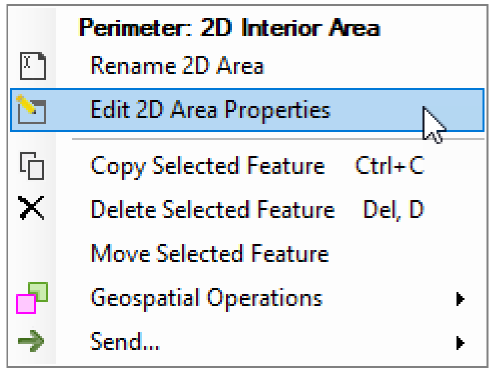
Figure 3-2. HEC-RAS 2D modeling computational mesh terminology.

After the 2D Flow Area polygon boundary is created, the next step is to begin creating the computational mesh. In general, the user should decide on a base cell size to use for the 2D flow area. Keep in mind that you will be able to refine the mesh with breaklines and refinement regions where needed. To create the base mesh, do the following:

1. Select the 2D Flow Area polygon to be edited.

2. Right click on the 2D Flow Area and select **Edit 2D Area Properties**.

3. Once that menu option is selected a window will appear as shown in Figure 3-3 below.



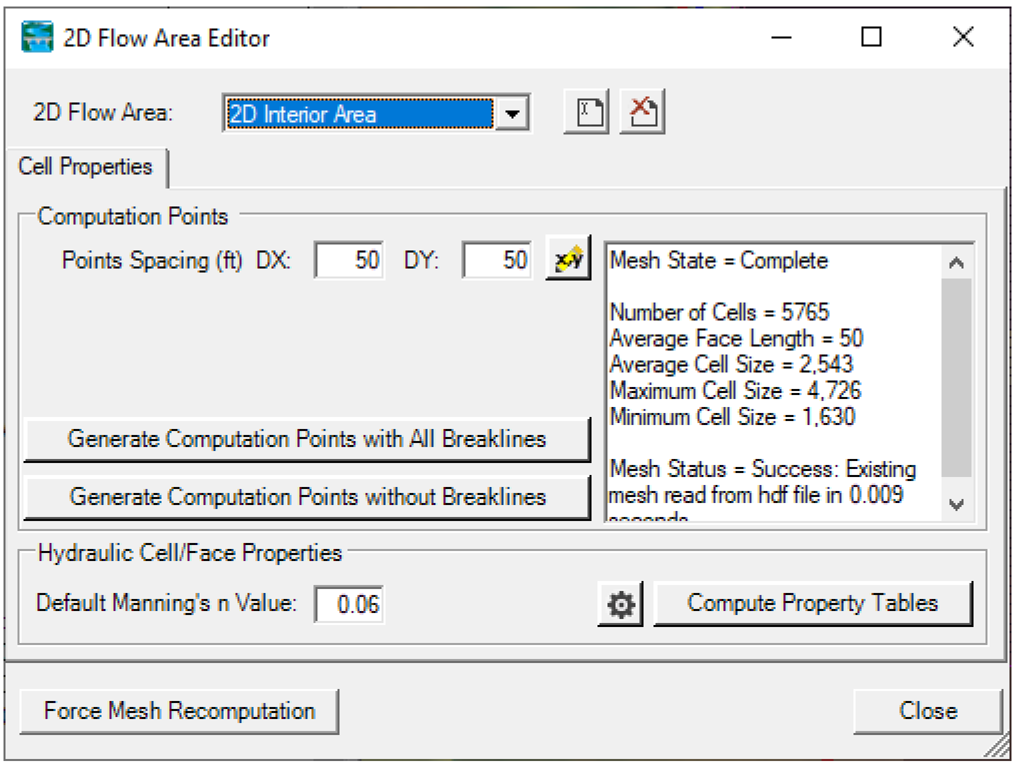


Figure 3-3. 2D Flow Area Editor.

4. In the **Computational Points** portion of the editor, enter the **Points Spacing DX and DY values.** Press the **Generate Computation Points with All Breaklines** button to generate the base mesh. Pressing this button will cause the software to compute a series of X and Y coordinates for the cell centers. The user can view these points by pressing the **View/Edit Computational Point's** button , which brings the points up in a table.

The user can cut and paste these into a spreadsheet or edit them directly if desired (It is not envisioned that anyone will edit the points in this table or Excel, but the option is available).

Warning: If there is an existing computational mesh and the "Generate Points in 2D Flow Area" option is used, all of the existing mesh points will be replaced with the newly generated points. Any hand editing that was done by the user will be lost.

5. If you want to have a default Manning's n value for the 2D Flow Area, it can be entered under the **Hydraulic Cell/Face Properties** area of the editor. The **Default Manning's n Value** will be used for the cell faces in the 2D flow area. Users have the option of adding a spatially varying Land Use classification versus Manning's n value table (and a corresponding Land Classification layer in RAS-Mapper), which can be used to override the base Manning's n values where polygons and roughness are defined. Even if a Land Use Classification versus Manning's n value table is defined, for any areas of the 2D flow area not covered by that layer, the base/default Manning's n value will be used for that portion of the 2D flow area.

Additionally, there is a gear button  that can be used to bring up an editor that allows the user to change the tolerances used when creating the hydraulic property tables for the 2D cells and faces. The following is a description of these tolerances:

Cell Elev-Vol Filter Tol: This tolerance is used to reduce the number of points in the 2D cell elevation volume curves that get developed in the 2D pre-processor. Fewer points in the curve will speed up the computations but reduce the accuracy of the elevation volume relationship. The default tolerance for filtering these points is 0.01 ft.

Cell Minimum Area Fraction: This field is used to enter a fraction that is multiplied by the cell area to establish a minimum area at the lowest elevation of the cell. The purpose of this field is to prevent the cell elevation volume curves from going down to an extremely small area at the bottom of the cell, and then creating a curve that is very abrupt at the lowest end. Establishing a reasonable minimum area for the cell helps out with the stability of solving that cell when it first starts to get water. The default is 0.01 (1 %).

Face Profile Filter Tol: This filter tolerance is used to reduce the number of points that get extracted from the detailed terrain for each face of a 2D cell. The default is 0.01 ft.

Face Area-Elev Filter Tol: This filter tolerance is used to reduce the number of points in the cell face hydraulic property tables. Fewer points in the curves will speed up the computations but reduce the accuracy of the face hydraulic property relationships. The default is 0.01 ft.

Face Conveyance Tol Ratio: This tolerance is used to figure out if more or less points are required at the lower end of the face property tables. It first computes conveyance at all of the elevations in the face property tables. It then computes the conveyance at an elevation half way between the points and compares this value to that obtained by using linear interpolation (based on the original points). If the computed value produces a conveyance that is within 2% (0.02) of the linear interpolation value, then no further points are needed between those two values. If linear interpolation would produce a value of conveyance that is more than 2% from the computed value at that elevation, then a new point is added to that table. This reduces the error in computing hydraulic properties, and therefore conveyance due to linear interpolation of the curves. A higher tolerance will result in fewer points in the hydraulic property tables of the cell faces, but less hydraulic accuracy for the flow movement across the faces. The default value is 0.02, which represents a 2% change.

Face Laminar Depth (ft): At very shallow depths on planar surface (ex. A parking lot or a road) the flow can be laminar instead of turbulent. If the flow is assumed to be turbulent all the way to a zero depth, then the program will underestimate the velocity of flows down a plane at very shallow depths. This field is used as the transition point from turbulent to laminar flows. Anything below this depth will assumed to be laminar flow and treaded as such.

6. You have the option to run the 2D Flow Area pre-processor right from this editor if you want to. This is accomplished by pressing the **Compute Property Tables** button. This step is not necessary, until the mesh is further refined, and you plan on performing a computation. The 2D pre-processor will automatically run when the user performs a Compute if the tables are out of date.

7. Press the **Close** button to close the editor and accept the mesh and property settings. An example of a base mesh, with not break lines or refinement regions, is shown in Figure 3-4.

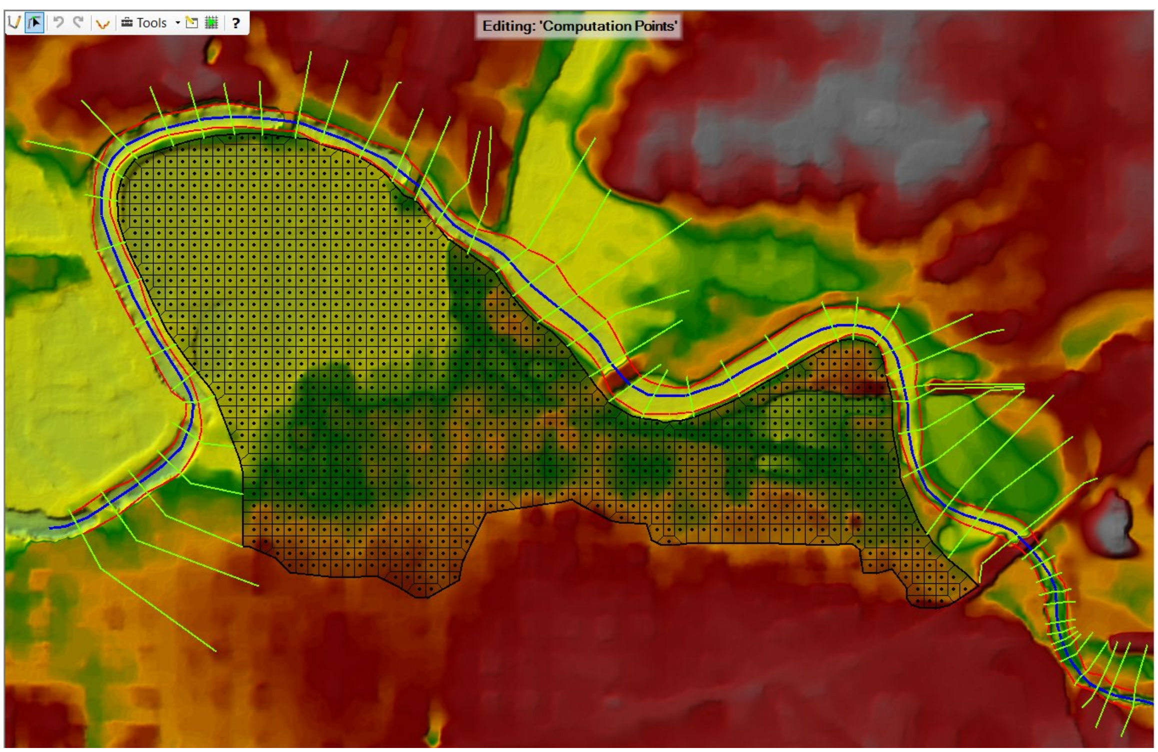


Figure 3-4. Example 2D Mesh with no Break Lines or Refinement Regions.

Editing/Modifying the Computational Mesh.

The computational mesh will control the movement of water through the **2D flow area**. Specifically, one water surface elevation is calculated for each grid cell center at each time step. The computational cell faces control the flow movement from cell to cell. Within HEC-RAS, the underlying terrain and the computational mesh are preprocessed in order to develop detailed elevation–volume relationships for each cell, and detailed hydraulic property curves for each cell face (elevation vs. wetted perimeter, area, and roughness). By creating hydraulic parameter tables from the underlying terrain, the net effect is that the details of the underlying terrain are still considered in the water storage and conveyance, regardless of the computational cell size. However, there are still limits to what cell size should be used, and important considerations for where smaller detailed cells are needed versus larger coarser cells.

In general, the cell size should be based on the slope of the water surface in a given area, as well as barriers to flow within the terrain. Where the water surface slope is flat and not changing rapidly, larger grid cell sizes are appropriate. Steeper slopes, and localized areas where the water surface elevation and slope change more rapidly will require smaller grid cells to capture those changes. Since flow movement is controlled by the computational cell faces, smaller cells may be required to define significant changes to geometry and rapid changes in flow dynamics.

HEC-RAS makes the computational mesh by following the Delaunay Triangulation technique and then constructing a Voronoi diagram (see Figure 3-5 below, taken from the Wikimedia Commons, a freely licensed media file repository):

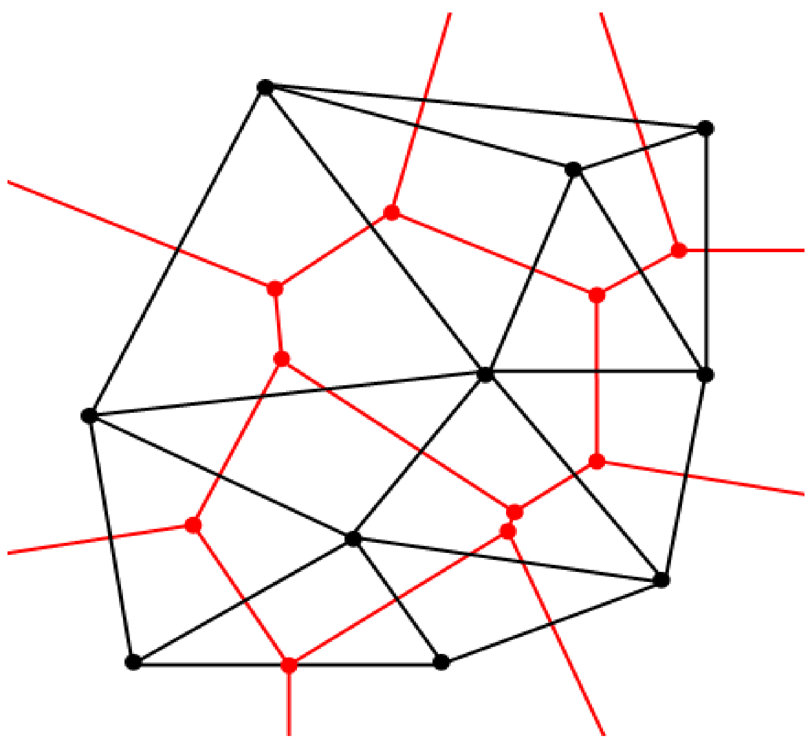


Figure 3-5. Delaunay - Voronoi diagram example.

The triangles (black) shown in Figure 3-5 are made by using the Delaunay triangulation technique (http://en.wikipedia.org/wiki/Delaunay_triangulation). The cells (red) are then made by bisecting all of the triangle edges (black edges), and then connecting the intersection of the red lines (Voronoi diagram). This is analogous to the Thiessen Polygon method for attributing basin area to a specific rain gage.

The computational mesh can be edited/modified with the following tools: break lines; refinement regions; moving points; adding points, and deleting points.

Break Lines

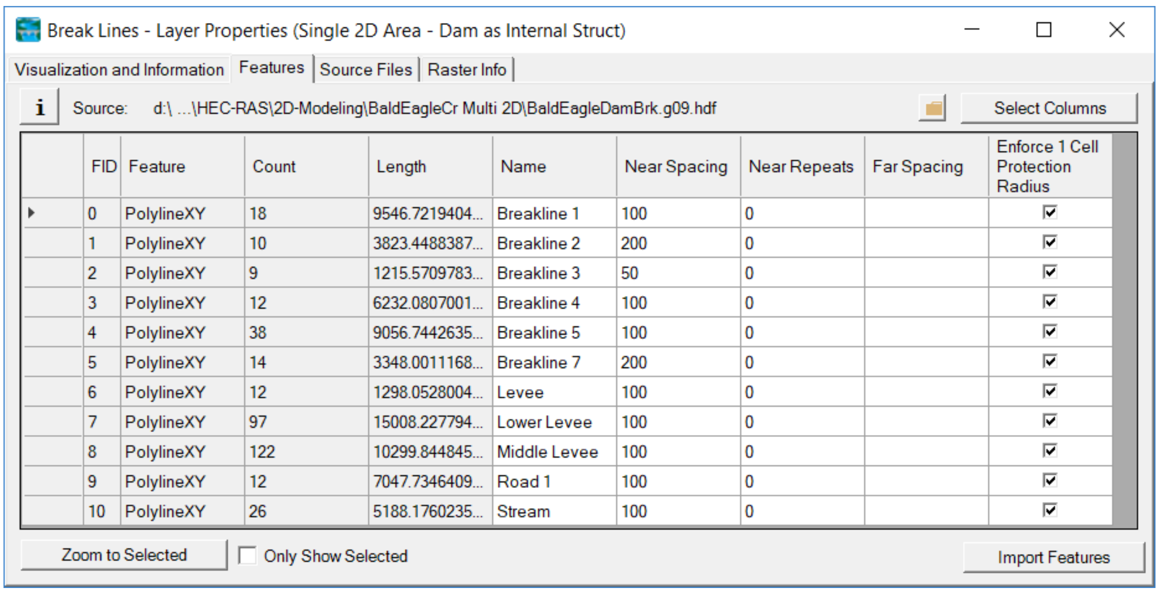
Either before or after the computational mesh is created the user may want to add break lines to force the mesh to align the computational cell faces along the break lines. In general, break lines should be added to any location that is a barrier to flow, or controls flow/direction. The user should align the faces of the 2D mesh for areas that are barriers to flow in order to accurately capture the high ground with cell faces. Examples where this should always be done are levees, roads, and natural berms in the terrain.

The user can add new break lines at any time. HEC-RAS allows the user to enter a new break line on top of an existing mesh and then regenerate the mesh around that break line, without changing the computational points of the mesh in other areas. Break lines can be drawn by hand; imported from Shapefiles; or detailed coordinates for an existing break line can be pasted into the break line coordinates table.

To add break lines by hand into a 2D flow area (while in the geometry editing mode), select the **Break Lines** layer (it should then be highlighted in magenta), then left click on the location within the 2D flow area that you want to start a break line. Left click to add additional points, and right click to re-center the view within the window. Double click to end a break line. Once a break line is drawn the software will ask you to enter a name for the break line. After the break line is entered, the mesh generation tools will automatically try to "snap" the cell faces to the break lines. The cells formed around break lines may not always have cell faces that are aligned perfectly with the break lines. The user can draw a new break line, then right click on the break line and select the option **Enforce Break line**. Once this option is selected, new cells will be generated around the break line with cell faces that are aligned along the break line. Any existing cell centers that were already in the mesh in the area of the break line are remove first (within a buffer zone around the break line, based on the cell size used around the break line). By default, the cell size used along the break line will be equal to the nominal cell size entered for the 2D flow area.

Additionally, the user can control the size/spacing of cells along the break line. When creating a mesh around a break line, it may be desirable or even necessary to use smaller cells than the nominal cell size used in other areas of the mesh. To control the cell spacing along a break line, right click on the **Break Lines** layer and select the option **Open Attribute Table (**Figure 3-6). A window will appear allowing the user to enter: a **Near Spacing**; **Near Repeats**; and a **Far Spacing**. There is also an option to "**Enforce a**

1 Cell Protection Radius" around the break line. This option protects the cells immediately around the break line from being deleted or moved due to other break lines that may be very close. The **Near Spacing** field is used to enter the cell size you want right along the break line. The **Near Repeats** field is an optional field that allows you to repeat the cell size define in the Near Spacing filed multiple times. For example, if the user entered a Near Spacing of 50 ft, then the cells on both sides of the break line would be 50 ft cells. If the user also entered a Near Repeats of 2, then two additional rows of 50 ft cells would be placed on both sides of the break line. The **Far Spacing** option allows the user to control how far the software will increase the cell size as it gradually transitions from the Near spacing to the user entered Far Spacing. If a Far Spacing is not entered, it is automatically set to the nominal cell size of the 2D flow area. The software will then increase the cell size around the break line, in order to provide a gradual cell size transition from the break line to the Far Spacing cell size being used for the mesh.



FID	Feature	Count	Length	Name	Near Spacing	Near Repeats	Far Spacing	Enforce 1 Cell Protection Radius
0	PolylineXY	18	9546.7219404...	Breakline 1	100	0		☑
1	PolylineXY	10	3823.4488387...	Breakline 2	200	0		☑
2	PolylineXY	9	1215.5709783...	Breakline 3	50	0		☑
3	PolylineXY	12	6232.0807001...	Breakline 4	100	0		☑
4	PolylineXY	38	9056.7442635...	Breakline 5	100	0		☑
5	PolylineXY	14	3348.0011168...	Breakline 7	200	0		☑
6	PolylineXY	12	1298.0528004...	Levee	100	0		☑
7	PolylineXY	97	15008.227794...	Lower Levee	100	0		☑
8	PolylineXY	122	10299.844845...	Middle Levee	100	0		☑
9	PolylineXY	12	7047.7346409...	Road 1	100	0		☑
10	PolylineXY	26	5188.1760235...	Stream	100	0		☑

Figure 3-6. Break Line Attributes Table.

To import Break lines from a shapefile, while editing the break lines, right click on the **Break Lines** layer and select **Open Attribute Table**. At the lower right hand corner of this editor is a button labeled **Import Features** (Figure 3-6). Select the **Import Features** button and a file browser will come up. Find the Shapefile that contains the lines that you want to use as break lines and select that file. Then press the **Open** button.

An example of using break lines within a 2D flow area for modeling levees and roads is shown in Figure 3-7.

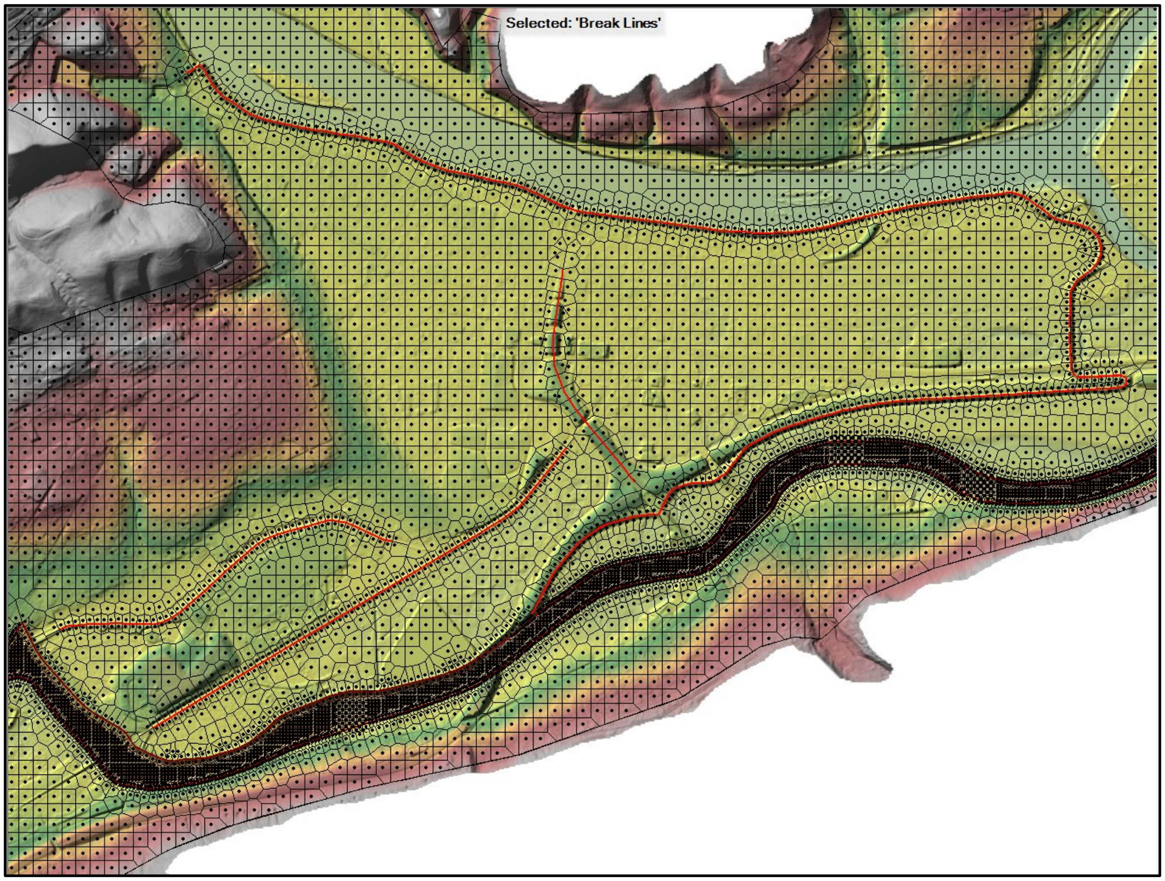
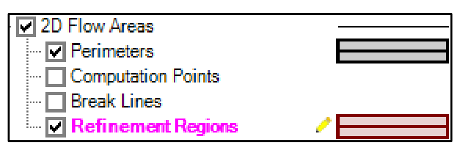


Figure 3-7. Example Break lines for Levees and roads

Mesh Refinement Regions

Refinement regions are a mesh editing tool that allows the user to refine or coarsen an area of the mesh. 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 break line). 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. Additionally, refinement regions can also be used to create a good mesh in the main channel regions of the model. To add a refinement region, do the following:



1. While editing the geometry in HEC-RAS Mapper, select the refinement region layer. It should turn to a magenta color if it is selected.

2. Select the **Add New Feature** editing tool, then left click to start drawing the refinement region polygon. Continue to left click to add additional vertices to the polygon. As you are drawing the polygon, users can right click to re-center the drawing window. Double click to end the refinement region polygon. A window will pop up asking the user to enter a name for the refinement region.

Additional properties for controlling how the refinement regions affect the 2D Flow Areas 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 break line, where the point spacing along the break line 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 **Error! Reference source not found.**, the properties for each refinement area include the Name, Cell Spacing X, Cell Spacing Y, Perimeter Spacing, Near Repeats, and Far Spacing. These properties (described below) control how the refinement region will be used to modify the mesh, when enforced (e.g., Figure 3-8).

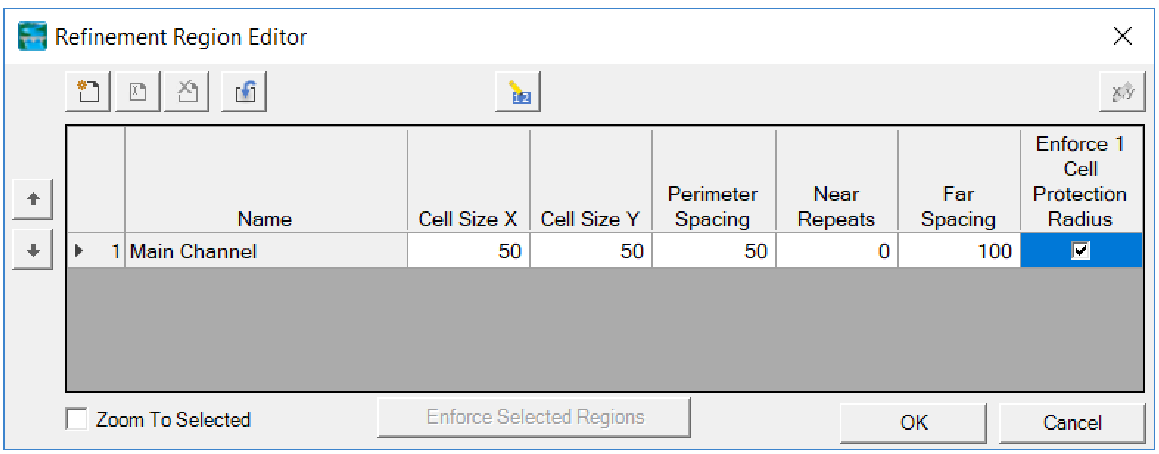


Figure 3-8. Refinement Region Editor.

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.

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.

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.

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 break lines cannot interfere with this already-enforced region.

A simple example of using two refinement regions to simplify and densify a mesh are shown in Figure 3-9.

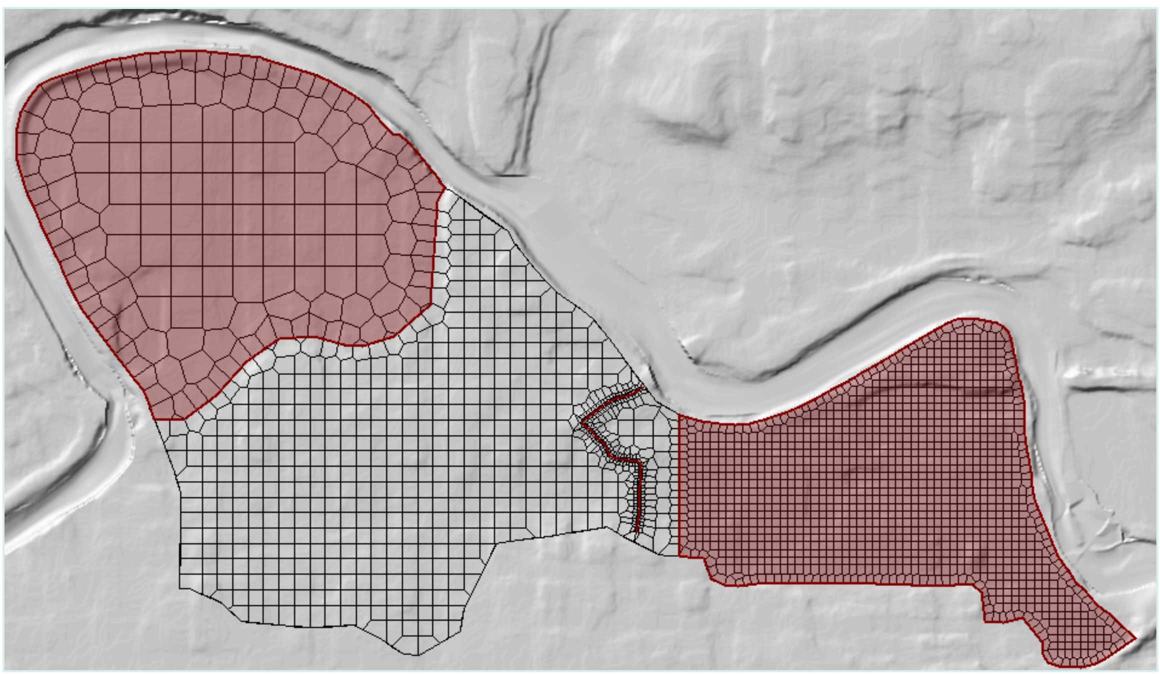


Figure 3-9. Example 2D Area with Refinement Regions – to simplify a portion of the Mesh and densify another portion of the Mesh.

Making a Channel Mesh

Another use for the refinement region is around the main channel of a stream. Shown in Figure 3-10 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. In addition to the refinement region, a break line was placed right down the center of the refinement region, following the path of the flow. The break line is used to align the cells in the middle of the channel with the direction of the flow. For the breakline in the example below, the number of near repeats was set to 4. For the refinement region, the near repeats was set to zero. Additionally, the option to "Enforce a 1 cell protection radius" was turned on for both the refinement region and the break line. Also, the break line was enforced after the refinement region. This combination of a refinement region and a break line down the center of the channel makes for a very nice channel mesh (Figure 3-10).

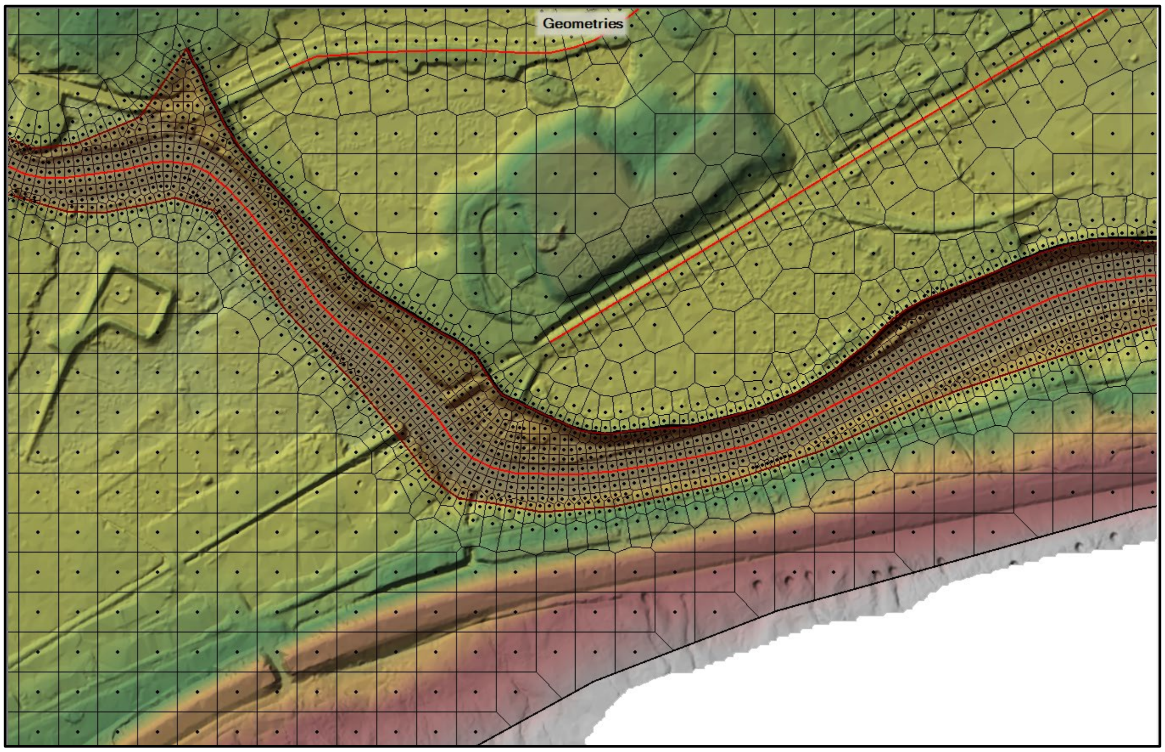


Figure 3-10. Example Refinement Region and Break Line Combination to form a Channel Mesh.

Hand Based Mesh Editing Tools

The hand editing mesh manipulation tools are available under the **Edit** menu of the HEC-RAS Geometric Data editor and are also available in HEC-RAS Mapper. From the Geometric editor, if the user selects **Edit** then **Move Points/Object**, the user can select and move any cell center or points in the bounding polygon. If a cell center is moved, all of the neighboring cells will automatically change due to this movement. If the user selects **Edit** then **Add Points**, then wherever the user left clicks within the 2D flow area, a new cell center is added, and the neighboring cells are changed (once the mesh is updated). The software creates a local mesh (Just the area visible on the screen, plus a buffer zone), such that while you are editing, just the local mesh will get updated. The entire mesh only updates once the user has turned off the editing feature, which saves computational time in creating the new mesh. If the user selects **Edit** then **Remove Points**, then any click near a cell center will remove that cell's point, and all the neighboring cells will become larger to account for the removed cell.

From HEC-RAS Mapper, the user can also hand edit the 2D Flow Area boundary and computation points. Start editing the Geometry in HEC-RAS Mapper, then select the **Perimeter** to edit the boundary, or select **Computational Points** to edit the cell computational points. To add new cell points, use the **Add New Feature** editing tool. To move or delete points use the **Edit Feature** tool.

The user may want to add points and move points in areas where more detail is needed. The user may also want to remove points in areas where less detail is needed. Because cells and cell faces are preprocessed into detailed hydraulic property tables, they represent the full details of the underlying terrain. In general, the user should be able to use larger grid cell sizes than what would be possible with a model that does not preprocess the cells and the cell faces using the underlying terrain. Many 2D models simply use a single flat elevation for the entire cell, and a single flat elevation for each cell face. These types of 2D models generally require very small computational cell sizes in order to model the details of the terrain.

Potential Mesh Generation Problems

The automated mesh generation tool in HEC-RAS works well, however, nothing is perfect. On occasion a bad cell will be created due to the combination of the user defined polygon boundary and the selected nominal cell size, or when the user is adding/modifying points inside of the polygon. After the mesh is made, the software automatically evaluates the mesh to find problem cells. If a problem cell is found, that cell's center is highlighted in a red color, and a red message will show up on the lower left corner of the geometric data window. Here is a list of some problems that are possible, and how to fix them with the mesh editing tools described above:

- **Mesh Boundary Issues:** When the user draws a 2D flow area boundary that is very sharp and concave, depending on the cell size selected, the mesh generation algorithm may not be able to form a correct mesh at this location (Figure 3-11). To fix this problem, the user can either add more cell centers around the concave portion of the boundary, or they can smooth out the boundary by adding more points to the boundary line, or both.

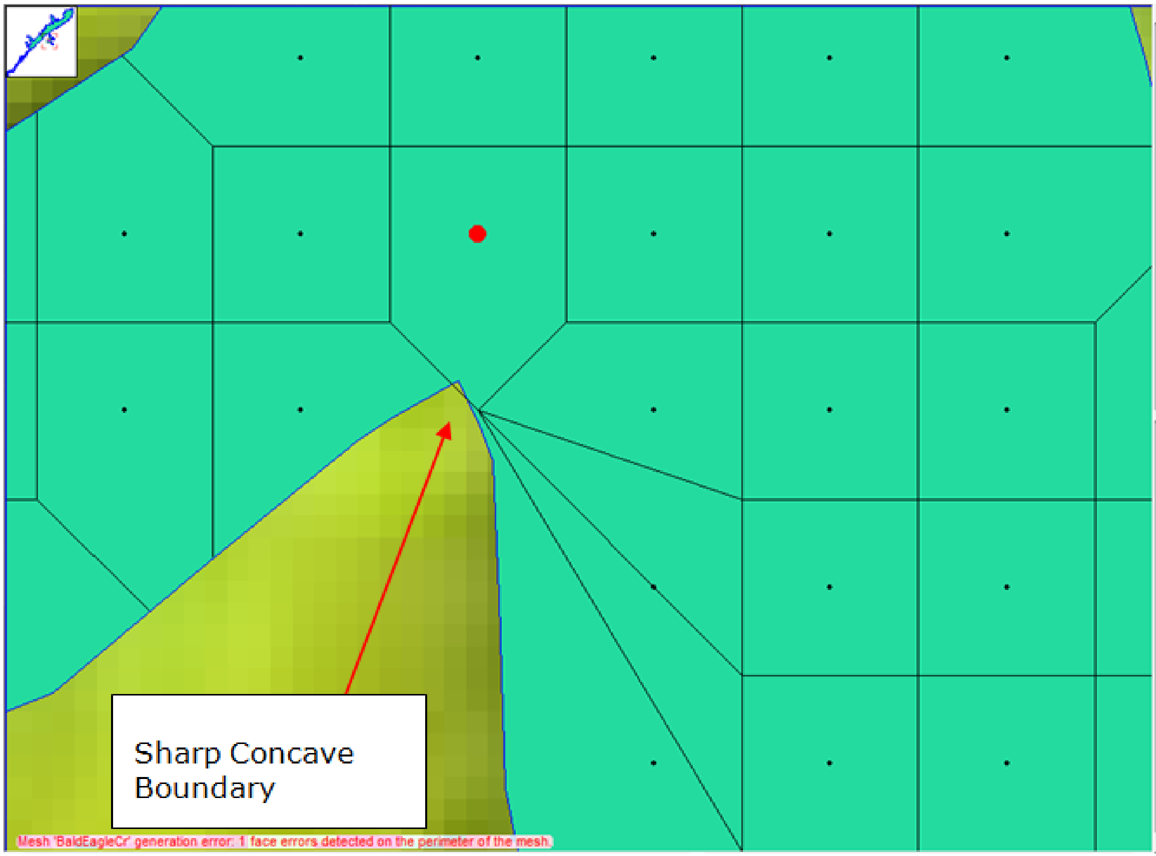


Figure 3-11. Example of a Sharp Concave Boundary Causing Mesh Generation Problem.

Shown in Figure 3-12 is the fixed mesh. The mesh was fixed by smoothing out the sharp concave boundary and adding some additional cell centers around the sharp portion of the boundary.

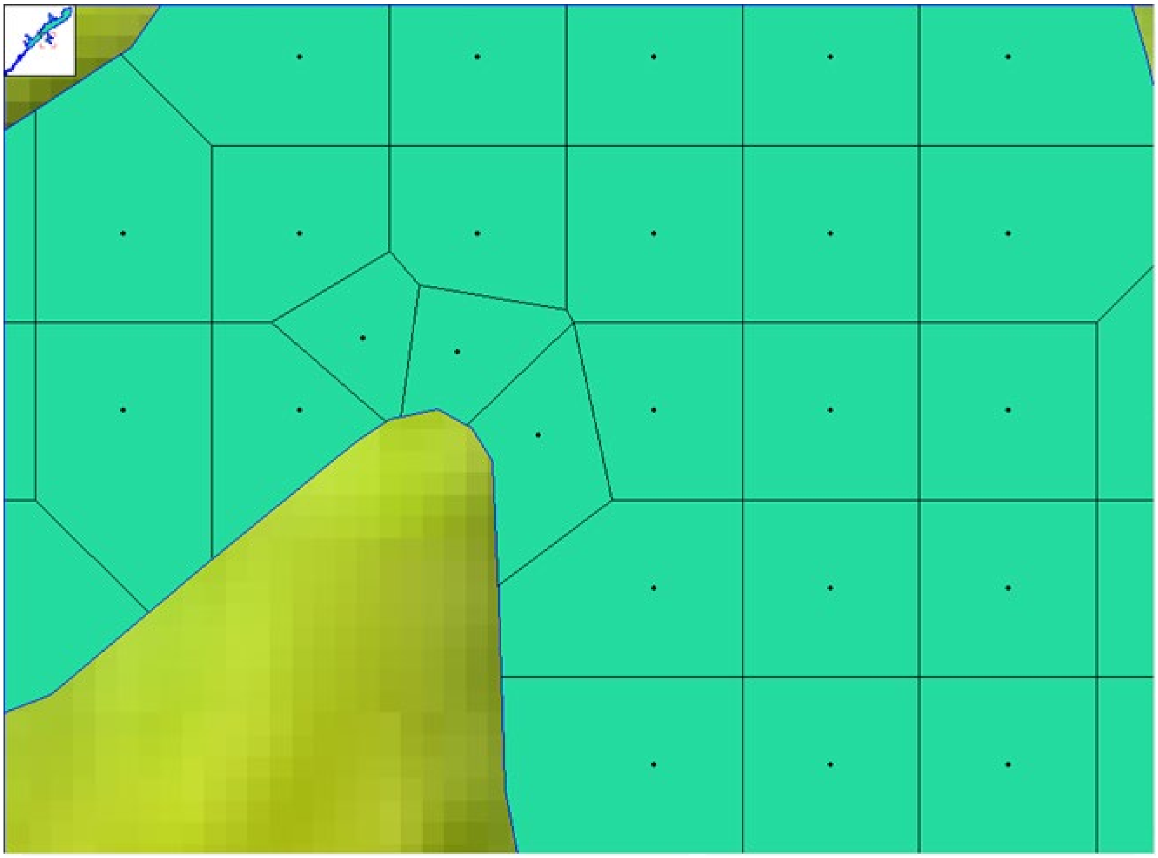


Figure 3-12. Corrected Mesh with Smoothed Boundary and Additional Cell Centers.

- **Too Many Faces (sides) on a Cell:** Each cell is limited to having 8 faces (sides). The HEC-RAS mesh development routines check for cells with more than 8 sides. If a cell exists with more than 8 sides it will be highlighted in red and a message will appear in the lower left portion of the geometric data window. An example of a cell with more than 8 sides is shown in Figure 3-13. If you have a cell with more than 8 sides you will need to edit that cell and/or the cells that bound it. Use the tools found in the Geometric editor under the "**Edit**" Menu. Available tools are (1). **Add Points**: to add points to the cell boundary polygon, or additional cells; (2). **Remove Points**: to delete points in the boundary polygon or delete cells; and (3). **Move Points/Objects**: to move the boundary points or the cell centers.

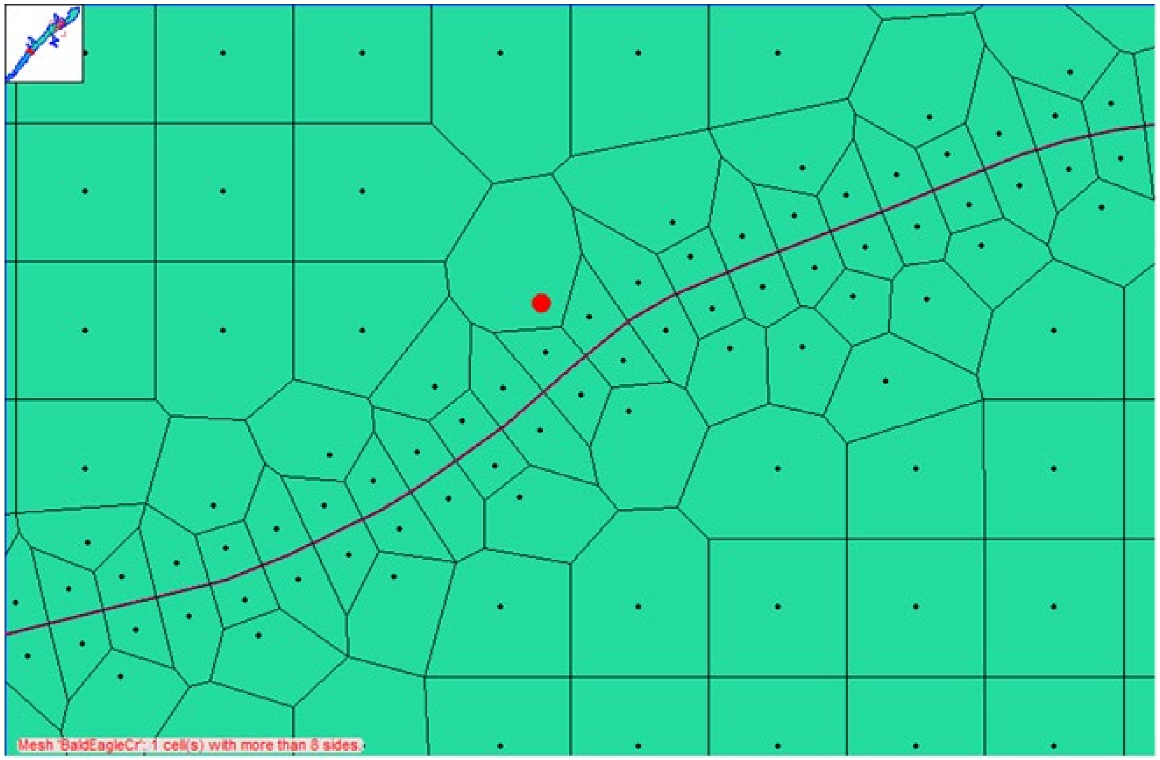


Figure 3-13. Example Mesh with a Cell that has more than 8 sides.

The mesh problem shown in Figure 3-13 (cell with more than 8 sides) was fixed by adding additional cell centers in the area of this cell, which made the cell size smaller and reduced the number of sides of the cells. The fixed mesh is shown in Figure 3-14.

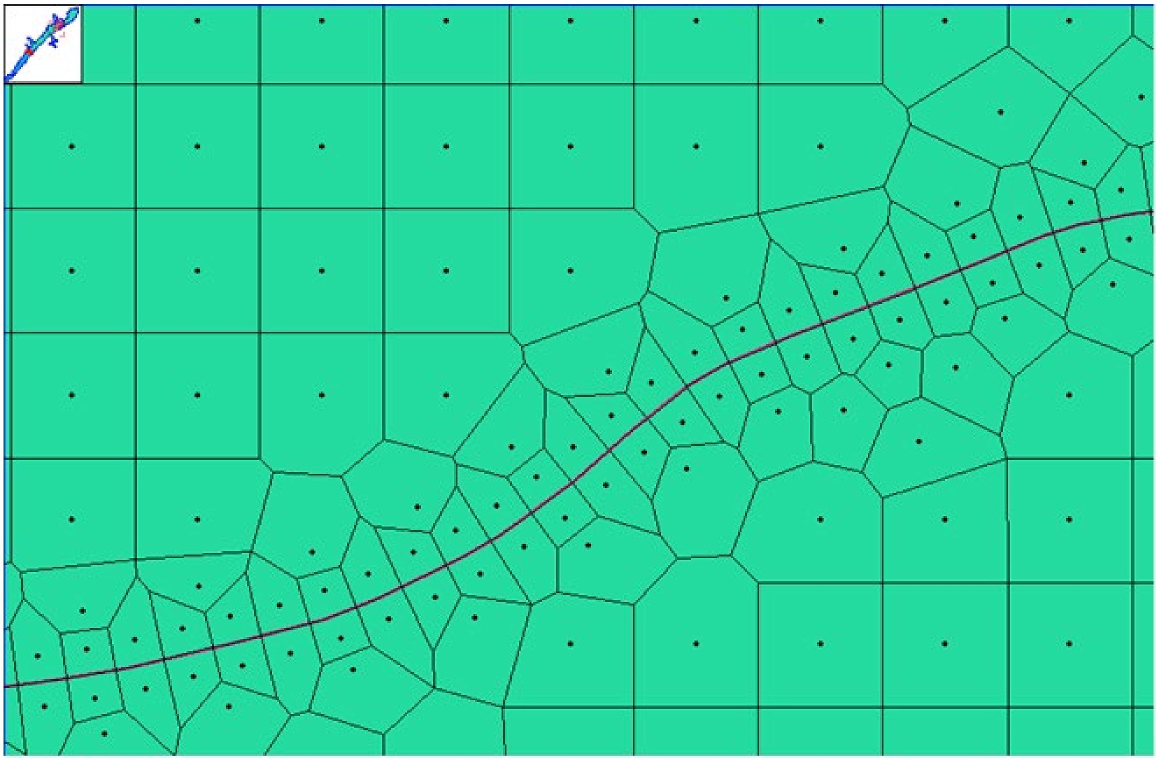


Figure 3-14. Final mesh after adding additional points to fix mesh problem.

- **Duplicate Cell Centers:** If a user accidentally puts a point right on top of, or very close to an existing cell center, this will cause a mesh generation problem. The mesh generation tools search for this issue and will identify any cell with duplicate cell centers. The solution to this problem is to delete one of the cell centers.

- **Cell Centers Outside of the 2D flow area Polygon.** If a user accidentally drops a point outside of the 2D flow area polygon, or they move the polygon boundary to the point in which cell centers are outside the polygon, this will generate a mesh error. The mesh routines will identify any points that are outside the current boundary. To fix the mesh, simply delete the points that are outside of the 2D flow area boundary.

- **Cells with Collinear Faces (Break Lines too close together).** Computational cells used within the HEC-RAS 2D code cannot have two faces that are collinear (i.e. they cannot form a straight line). Where two cells meet (at a face point), the outside angle formed by the two faces must be greater than 180 degrees. This is called "Strict Convex" in mathematical terms. Meaning all cells are require to be strictly convex, and therefore no two faces within the same cell can be collinear (Form a straight line). If cells end up like this, the software will run, but the computation across cell faces that are like this will not be correct.

This problem is generally caused by placing two or more break lines parallel to each other, and close together, such that the creation of cells along one break line can create problems with cells along the other break line. Cells are created along break lines one at a time. Additionally, the user can specify a cell min and max size to form the cells around a break line. The min cell size gets used right along the break line, then it transitions out the max cell size by doubling the cell size as it goes outward from the break line. If the user does not put in a max cell size, the software assumes that they want to transition out to the nominal cell size that was entered in the 2D Flow Area editor.

If the break lines are close together, this transitioning out can "Bleed" over into the cell space along a neighboring break line. This can have the effect of forming cells that no longer follow the first break line, but even worse, cells could be formed that are not correct/consistent with the HEC-RAS 2D solution scheme. Below (Figure 3-15) is an example in which break lines were used for the channel bank lines and the channel centerline. In this case, all break lines were being transition out to the maximum cell size, which caused this bleed over effect. In this example the channel centerline break line was enforce first, then the channel bank lines. The cells around the channel bank lines were formed correctly, but the cells along the channel centerline were not, due to the fact that the cells formed from the channel bank line break lines override the previously formed cells along the channel centerline (Simply because the order in which they were enforced).

To fix this issue, you can either enter a max cell size that prevents that break lines cells from bleeding over into the neighboring break line cells, or you can enforce the break lines by hand, thus controlling the order in which cells get formed. This specific example was fixed by enforcing the Channel centerline by hand as the last break line to be enforced. See the fixed mesh in Figure 3-16.

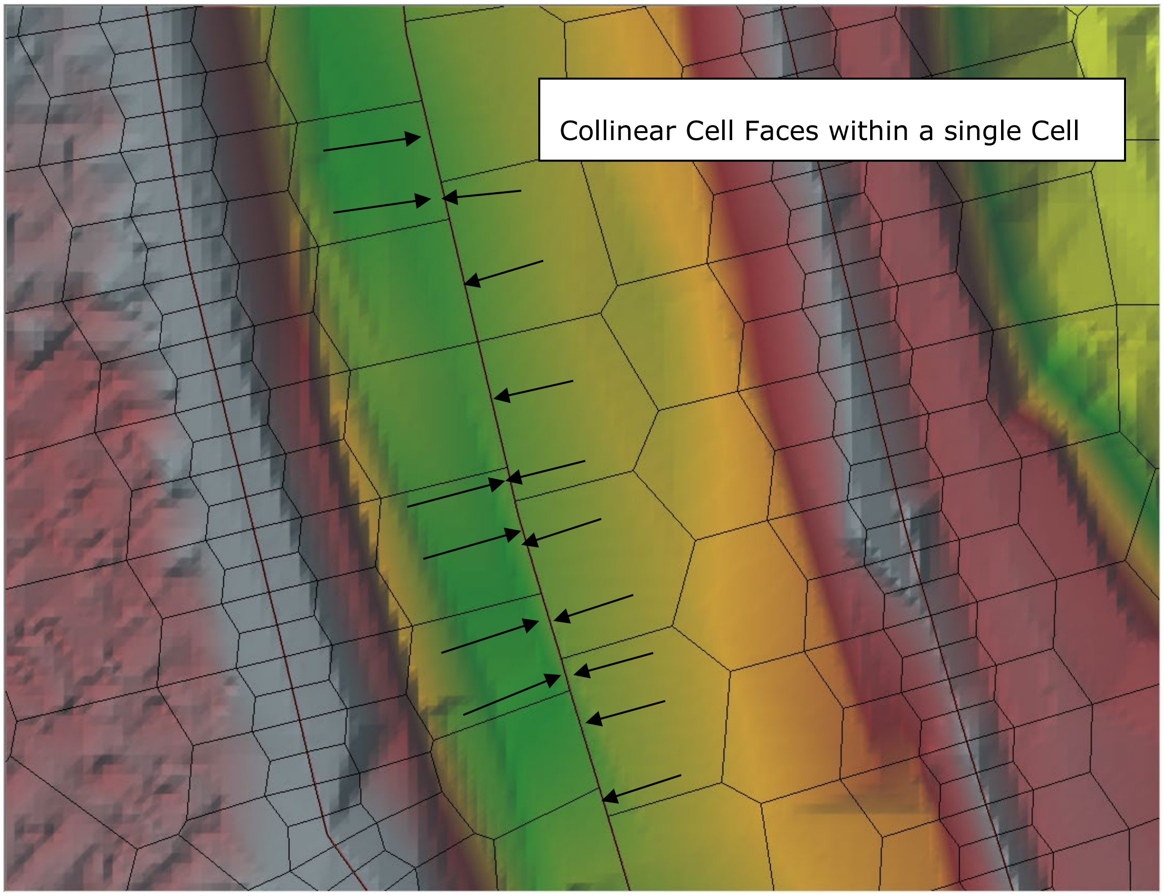


Figure 3-15. Example of bad cells with two or more faces that are collinear.

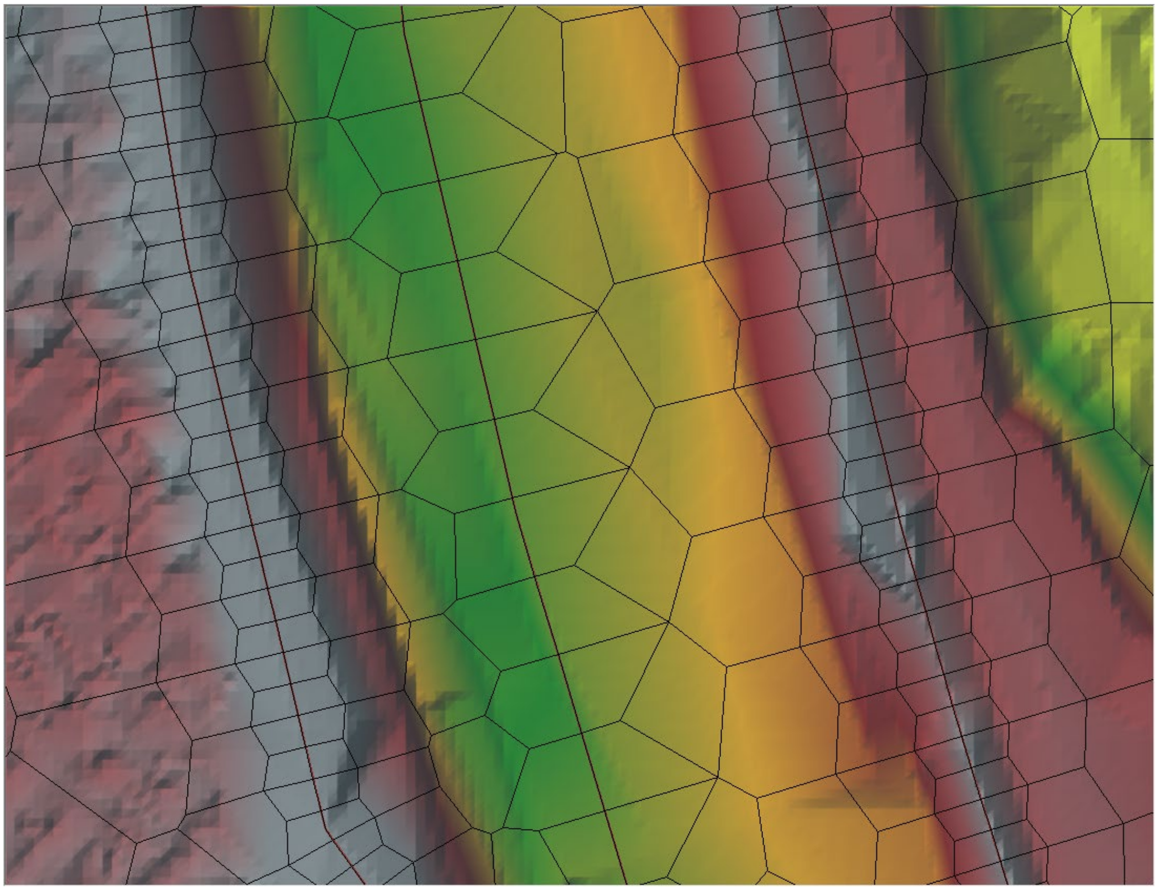


Figure 3-16. Example of properly formed cells along the centerline break line.

Creating Hydraulic Property Tables for the 2D Cells and Cell Faces

As previously mentioned, the 2D Computation Mesh is preprocessed into an elevation – volume curve for each cell, and a series of hydraulic property curves for each cell face (elevation vs. wetted perimeter, area, and roughness). This pre-processing is accomplished in RAS Mapper. The hydraulic property tables are derived from the details of the underlying terrain used for the model, as well as any user defined Manning's n by land cover relationships set in the geometry file. A terrain model is required to use 2D modeling within HEC-RAS. The terrain data is also required in order to do any mapping of the computed results, for both the 1D and the 2D areas of the model. Please review Chapter 2 of this manual for instructions on creating a Terrain model for use in 2D modeling and results visualization.

Once a terrain model is created, and optionally a Manning's n by Land Cover table, then the following steps are required to create the hydraulic property tables for the 2D cells and cell faces, which are used in the 2D hydraulic computations:

Associating a Terrain and Manning's Layer with a Geometry

After a new terrain layer and Manning's n layer are added, the user **must** associate the terrain and Manning's layers with any or all of the geometry files within the HEC-RAS project. This is accomplished in RAS Mapper by right-clicking on the **Geometries** layer at the top of the layer list (on the left hand side of the RAS Mapper window), then selecting the **Manage Geometry Associations** option from the popup menu. When this is done a window will appear, as shown in Figure 3-17, in which the user can select a terrain layer for each geometry file.

Note: RAS Mapper will attempt to associate a terrain model with the RAS Geometry when the Terrain Layer is first created. However, the user should verify the correct Terrain has been associated. The user will also be required to associate the terrain with a geometry, if there are multiple terrains.

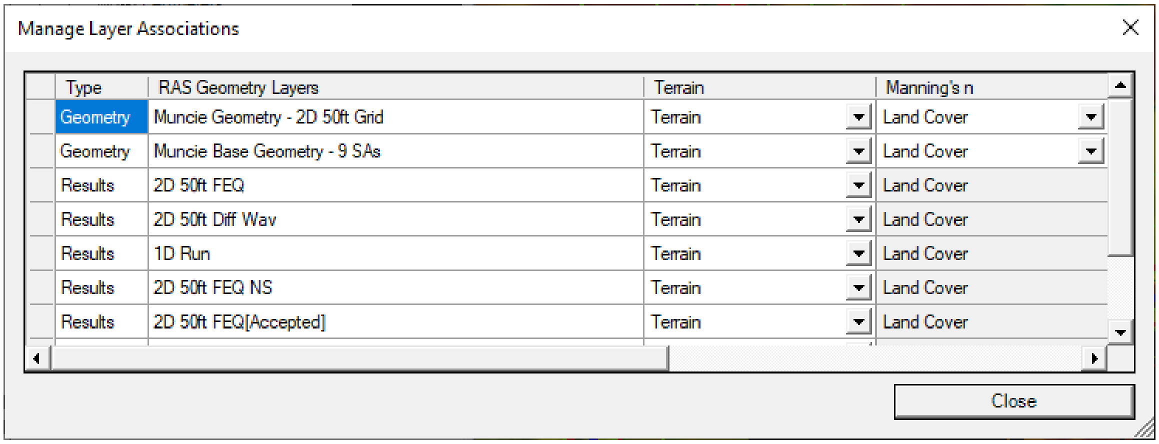


Figure 3-17. Terrain Association Editor.

After associating all of the geometry files with the terrain and Manning's n layer(s), select the **Close** button. This will ensure that these associations are saved.

2D Geometric Preprocessor

Overview of Cell and Face Properties

Each cell, and cell face, of the computational mesh is preprocessed in order to develop detailed hydraulic property tables based on the underlying terrain used in the modeling process (Casulli, 2008). The 2D mesh pre-processor computes a detailed elevation-volume relationship for each cell. Each cell face of a computational cell is pre-processed into detailed hydraulic property tables (elevation versus wetted perimeter, area, roughness, etc…). This allows the user to use larger computational cells, while keeping the details of the underlying terrain. The net effect is that larger cells results in fewer computations, which means much faster run times. Additionally, HEC-RAS will produce more detailed results for a given cell size than other models that use a single bed elevation for each cell and cell face.

An example of how HEC-RAS pre-processes cells and faces into detailed property tables is shown in Figures 3-18 through 3-21. Shown in Figure 3-18 are the details of the underlying terrain within a single computational cell.

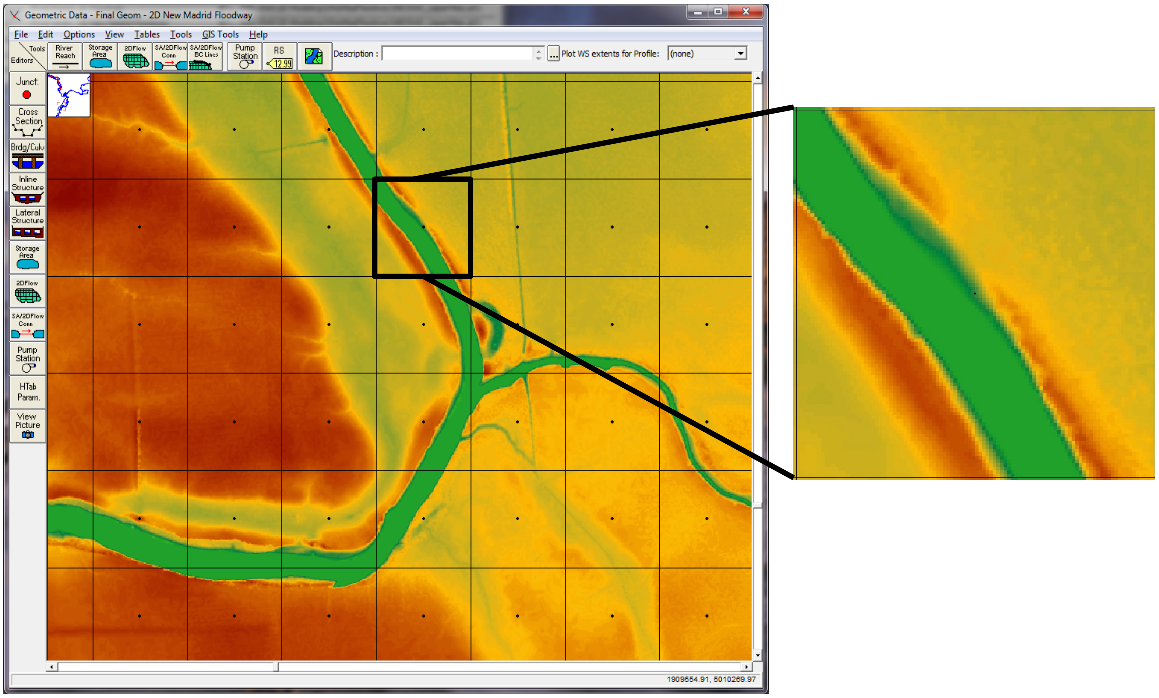


Figure 3-18. Details of Underlying Cell Terrain Data.

When the 2D Geometric Pre-processor runs a detailed elevation-volume relationship is developed for each cell. See an example of this in Figure 3-19 below.

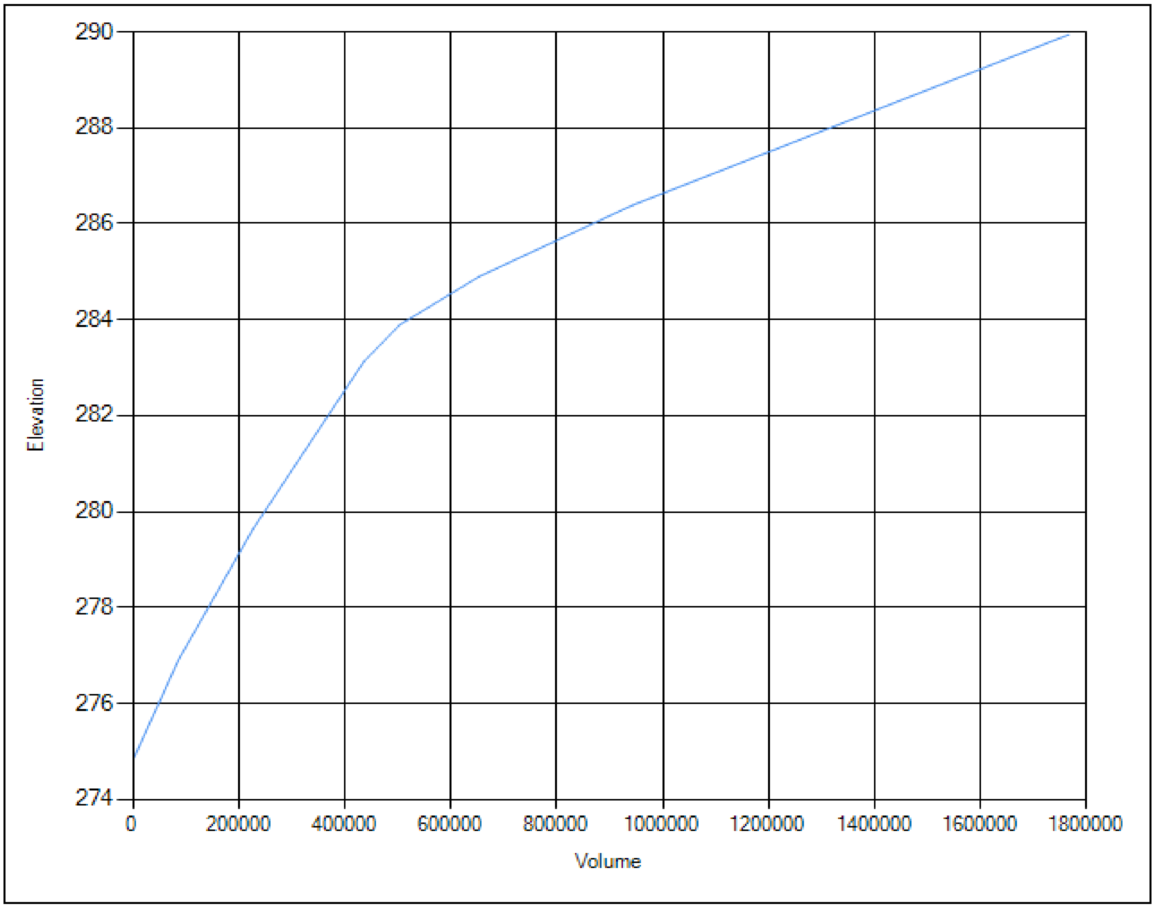


Figure 3-19. Elevation – Volume relationship for a 2D cell.

In addition to the processing of the cells, the faces of the cells are preprocessed into tables of elevation versus area, wetted perimeter, and roughness. See Figure 3-20 below:

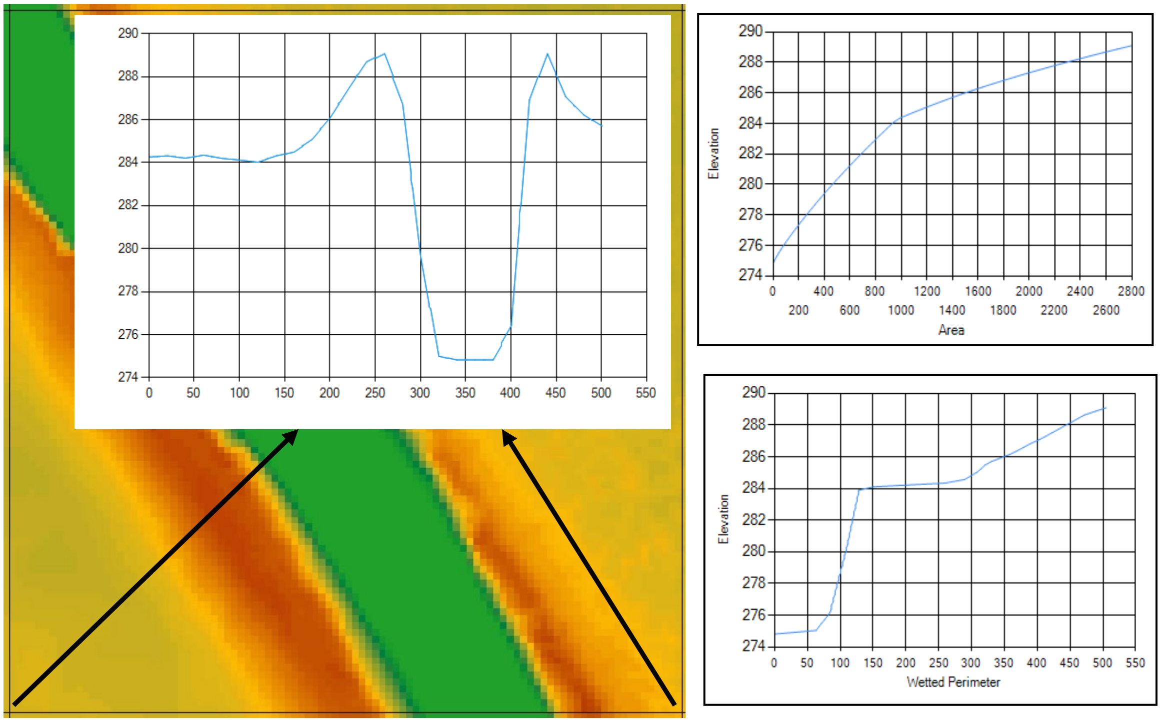


Figure 3-20. Example of how Cell Faces are processed into detailed hydraulic tables.

As shown in Figure 3-20, each cell face is like a detailed cross section. So the flow of water into, through, and out of a cell is controlled by the details of these face properties, and the cell elevation-volume relationship. The benefit of this is much greater hydraulic details at the cell level over other models that use a single elevation for each cell and face. With HEC-RAS, users can have much larger cells, but still retain great hydraulic detail within a cell. Additionally, HEC-RAS cells can be partially wet (i.e. water does not have to cover the entire cell, and can move through a portion of the cell). An example of this is shown in Figure 3-21.

Figure 3-21. Example of detailed channel moving through larger cells in HEC-RAS

Shown in Figure 3-21 is an example of how the computational cells in HEC-RAS contain enough hydraulic detail such that flow can move through a channel, even though the channel is smaller than the cell size. In the above example, the 2D cells are 500 x 500 ft (the underlying terrain is 2 x 2 ft grids). Water will move through the channel portion of the cells, because the details of the channel cross sections are contained within the cell faces. Additionally, the details of the elevation-volume relationship in the channel are contained within the cell hydraulic properties table. In this type of example, flow can move through a channel in a 1D-type of mode, while flow in the overbank areas will be 2D from cell to cell. If the user wants more detail within the channel, such as two-dimensional flow velocities and varying water surface elevations, then a cell size in that would allow 5 to 7 (or more) cells across the channel is needed. The smaller cell size will allow the model to capture the two dimensional flow effects within the channel itself. However, if the user only needs to capture the two-dimensional flow effects on the floodplain, then the approach shown in Figure 3-21 is a viable option. This type of model is not a detailed channel model, but it does allow water to move through the channels in a 1D fashion, with a reasonable amount of conveyance computed for the channel, based on the terrain at the cell faces.

The 2D flow capabilities in HEC-RAS can be used in many ways. The user can develop a mesh with very small cell sizes that can be used to model both channels and floodplains in great detail. Or the user can use larger cell sizes, which will give you less detail in the channel, but still 2D flow hydraulics in the floodplain. The level of detail the user chooses depends on what is being modeled, and the purpose of the study. HEC-RAS provides the user with the maximum amount of flexibility in modeling the details of a channel and the floodplain in 2D. Preprocessing the cells and cell faces into detailed hydraulic property tables is an advantage over 2D models that use a single elevation for each cell (flat cells), and a single line for each face (flat or linear sloping faces).

Running the 2D Geometric Preprocessor

After associating the geometry files with the terrain layer, the user can run the 2D flow area geometric pre-processor from within RAS Mapper. This step does not have to be done in RAS Mapper. If the user does not run the 2D Geometric preprocessor in RAS Mapper, it will automatically be done as a separate process during the unsteady flow computations.

In the Geometry group there will be a sub layer called **2D flow area**. Checking the box for this layer will turn on the mesh for all of the 2D flow areas contained within that geometry file. In this example, there is only one 2D flow area. Right click on the sub layer called **2D flow area**, then select the option labeled **Compute 2D Flow Areas Hydraulic Tables** (see Figure 3-22). This is the option to pre-process the 2D flow area computational cells and faces into detailed tables based on the underlying terrain data. If the user does not do this step here, the HEC-RAS user interface will detect that the pre-processing step has not been done, and it will do it during the unsteady flow computational process (right before it performs the existing 1D pre-processing of the cross sections and hydraulic structures). Also, if the user later changes anything about the 2D area (adds, moves, deletes cells, changes Manning's n-values, etc…), then the 2D pre-processor step will automatically be rerun during the unsteady flow computational process.

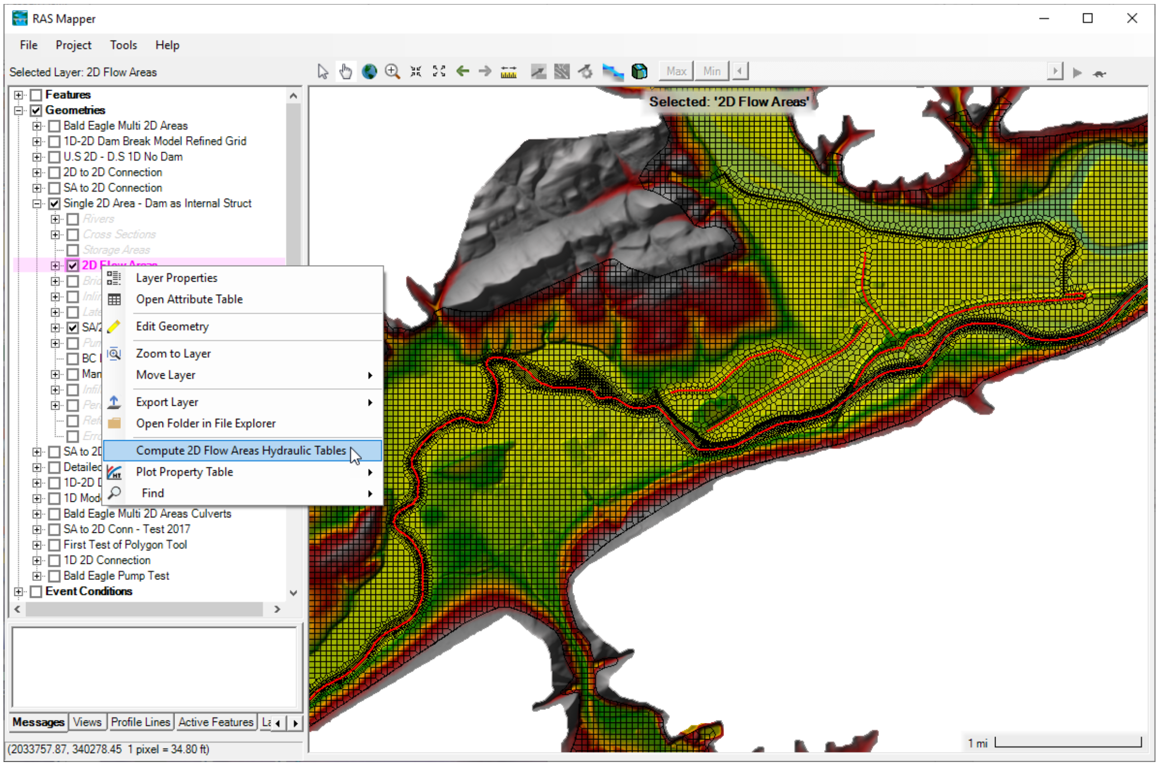


Figure 3-22. Computing 2D flow area Hydraulic Tables from RAS Mapper.

Connecting 2D flow areas to 1D Hydraulic Elements

The 2D flow area elements can be connected to 1D elements in several ways: directly to the downstream end or the upstream end of a river reach; laterally to 1D river reaches using a Lateral Structure(s); and/or directly to another 2D area or storage area using the SA/2D Area Connection. The process for connecting a 2D flow area to other hydraulic elements can be accomplished in HEC-RAS Mapper or in the HEC-RAS Geometric Data editor. This example shows how to develop and connect a Lateral Structure from a 1D reach to a 2D Flow Area in the HEC-RAS Geometry Editor. To learn how to do this in HEC-RAS Mapper, please review the HEC-RAS Mapper User's manual.

Connecting a 2D flow area to a 1D River Reach with a Lateral Structure.

2D flow areas can be used to model areas behind levees or overbank flow by connecting a 1D river reach to the 2D area using a Lateral Structure (see Figure 3-23).

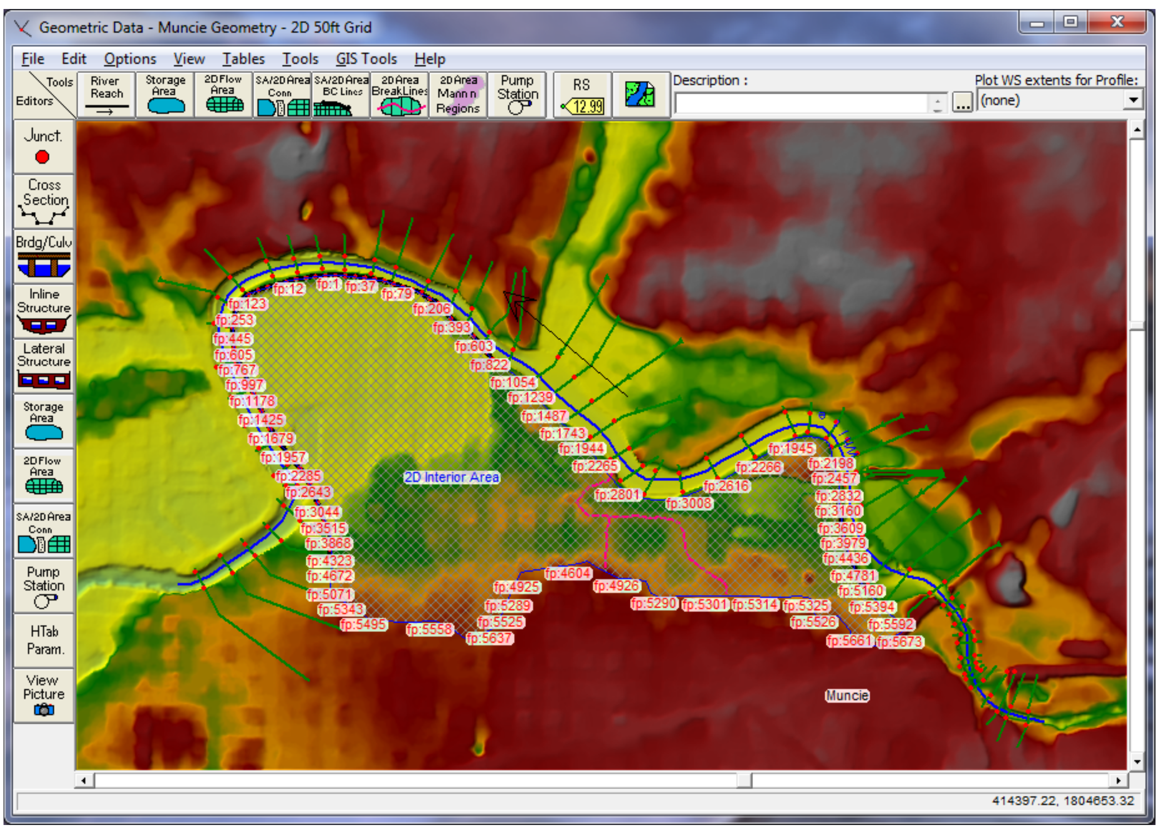


Figure 3-23. HEC-RAS Geometric Editor with the 2D flow area boundary Face Points displayed.

For this example, zoom into the upstream end of the river, which is on the right hand side of the schematic. At this location add a Lateral Structure that represents the levee in that region. When a Lateral Structure is added to the 1D River Reach, the user can select to link it to another 1D River Reach, a Storage Area, or a 2D flow area. If the user selects to link the Lateral Structure to a 2D flow area, then the stationing of the Lateral Structure will be linked to the 2D Area's face points automatically (this is analogous to the Lateral Structure automatically determining the location and intersection of the 1D cross sections). The connected levee is shown in Figure 3-24.

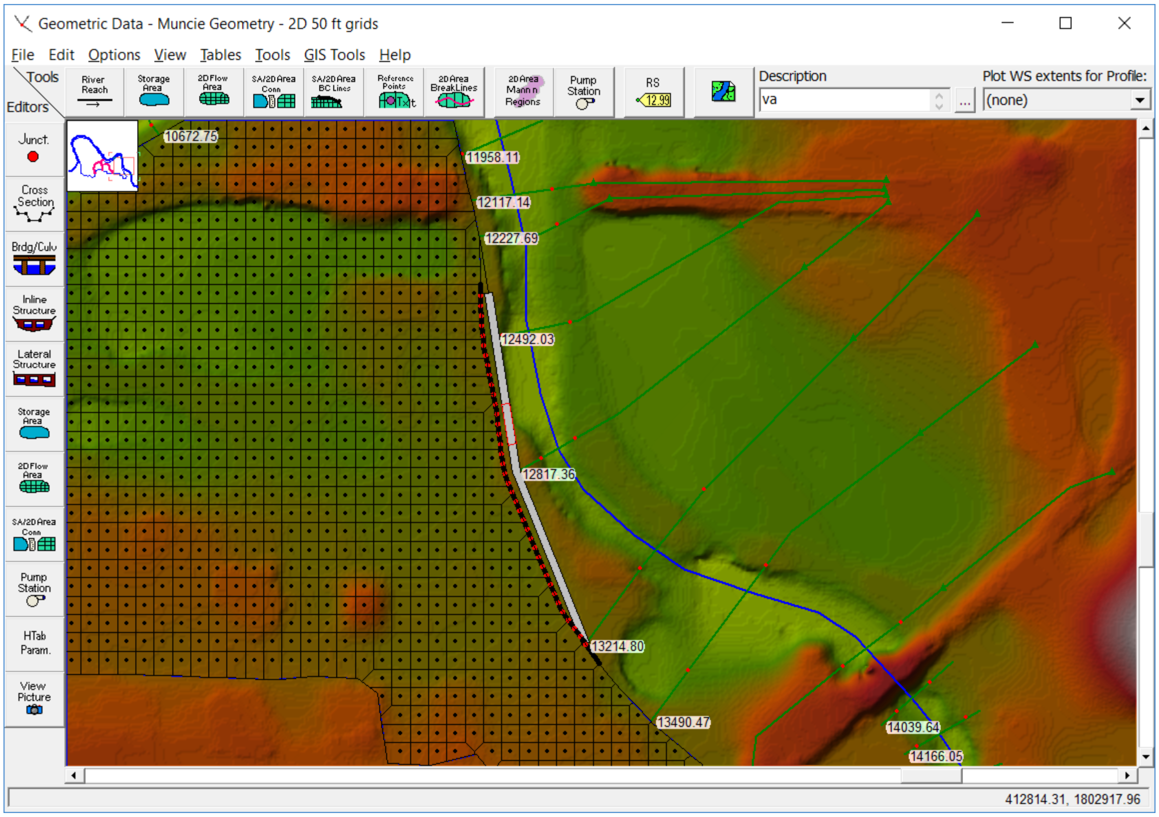


Figure 3-24. Example of a Lateral Structure (levee) hooked up to a 2D flow area.

In this example, this Lateral Structure (levee) will be used to model flow going over the levee, as well as a levee breach that will be added later.

HEC-RAS has the option to have georeferenced lateral structures. Under the menu item labeled **GIS Tools**, there is now a table option called **Lateral Structure Centerlines Table**. User can use the **Measure Tool** to draw a line that would represent the lateral structure geospatial X and Y coordinates, then paste those coordinates into the Lateral Structure Centerline Table (This is optional). If a user inserts geospatial coordinates for a lateral structure, not only will it be drawn geospatially correct, but HEC-RAS will figure out how elements (1D cross sections and 2D Face Points) are connected to the lateral structure based on its spatial location. User's can also layout Lateral structures in HEC-RAS Mapper by editing the geometry layer, then the Lateral structure layer.

Note: if you put in a Geospatial centerline for a lateral structure, the length of the lateral structure weir/embankment stationing must be within 0.5% of the length of the centerline put in (i.e. they need to be consistent with each other in terms of length).

Users can use the Geometry Editor measuring tool option to draw the geospatial line that will represent the Lateral Structure, or they can import the geospatial information from an ESRI shapefile. Shown in Figure 3-25 is the **Lateral Structure Centerlines Table**.

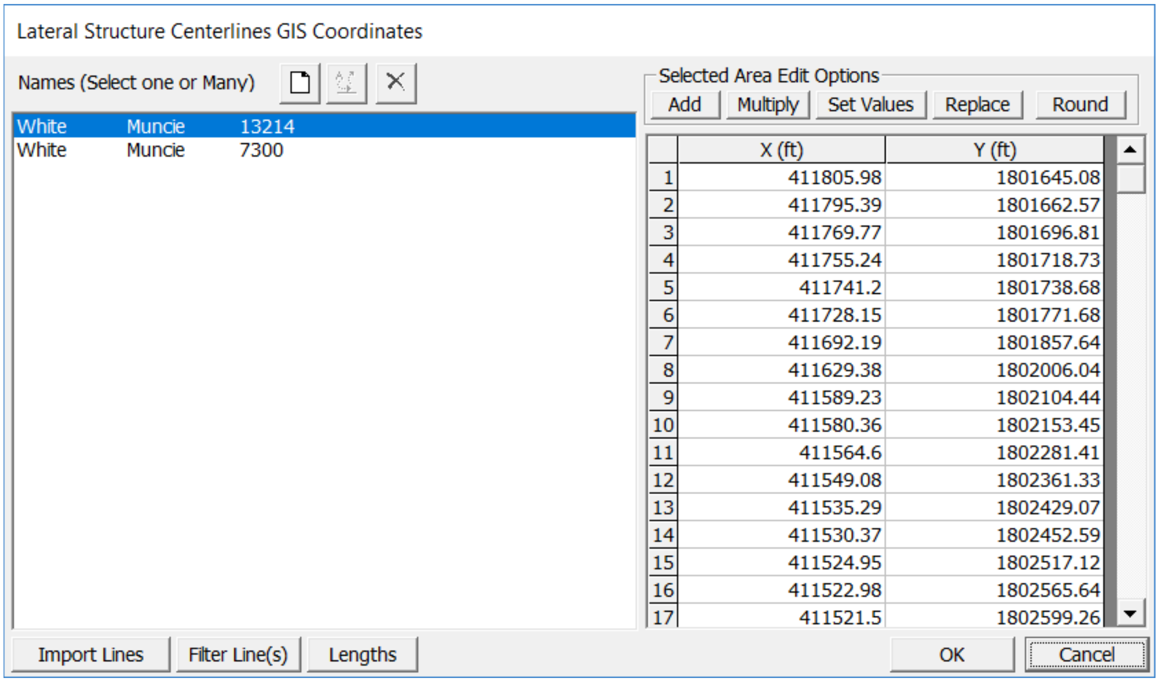


Figure 3-25. Example Lateral Structure Centerlines Table with Geospatial coordinate data for a levee.

To draw a geospatial line from within the Geometric Data editor, use the measuring tool option. This is accomplished by holding down the **Cntrl key**, then using the mouse pointer and the left mouse button to draw the line. Click the left mouse button to start the line, then move the mouse and continue to left click to add additional points in the line. As you are drawing the line, you can also right click the mouse to re-center the drawing within the Geometric Data editor. This is very helpful when you are zoomed in and need to continue the line to an area off of the screen. To finish the line, left click on the last point then release the Cntrl key. Once you release the Cntrl key, a window will appear as shown in Figure 3-26.

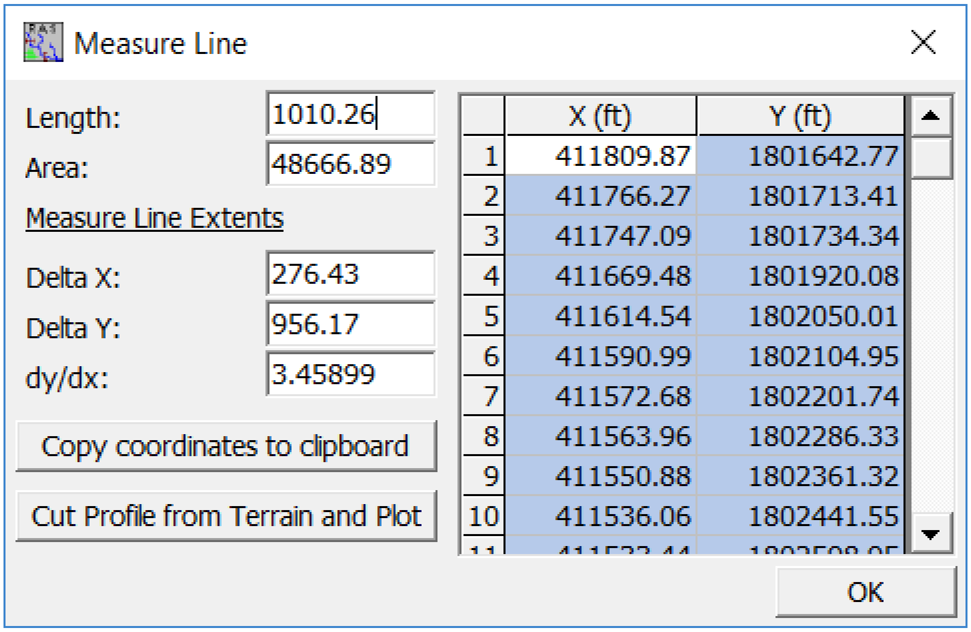


Figure 3-26. Example Measuring Tool line data window.

As shown in Figure 3-26, the measuring tool will show you the geospatial X and Y coordinates of the line in a table. To send these coordinates to the Windows Clipboard (so you can then paste them into the Lateral Structure Centerlines Table), simply press the button labeled: **Copy GIS to Clipboard**. These coordinates can be pasted into the Lateral Structure Centerline Table to georeferenced a lateral structure. The measuring tool window also shows you the length of the line; the area of a polygon if the first and last point were connected; Delta X; Delta Y; and dy/dx. Additionally you can plot the terrain data underneath that line by pressing the button labeled: **Cut Profile from Terrain and Plot** (This only works if you have a terrain data set in RAS Mapper and you have it associated with the currently opened Geometry file). This line can be used as a first cut for the user entered Weir Station and Elevation data for the Lateral Structure Weir profile. This is especially useful if the Lateral Structure is being used to represent the high ground barrier between the main channel (1D river reach) and overbank area (2D Flow Area).

The process of connecting a Lateral Structure to a 2D flow area is described below:

1. Add the Lateral Structure as would normally be done in HEC-RAS (i.e. create the Lateral Structure; define the upstream River Station of the structure; enter the station/elevation points that represent the weir/embankment of the top of the structure; and add geospatial data for the lateral structure as described above).

2. For the **Tailwater Connection** option on the **Lateral Structure** editor, select the **Type** as **Storage Area/2D Flow Area**. Then from the **SA/2D FA** field, select the name of the 2D flow area to be connected to the lateral structure by pressing the

Set SA/2DFA button and selecting the 2D flow area name. In this example the name of the 2D flow area is "2D Interior Area" (see Figure 3-27).

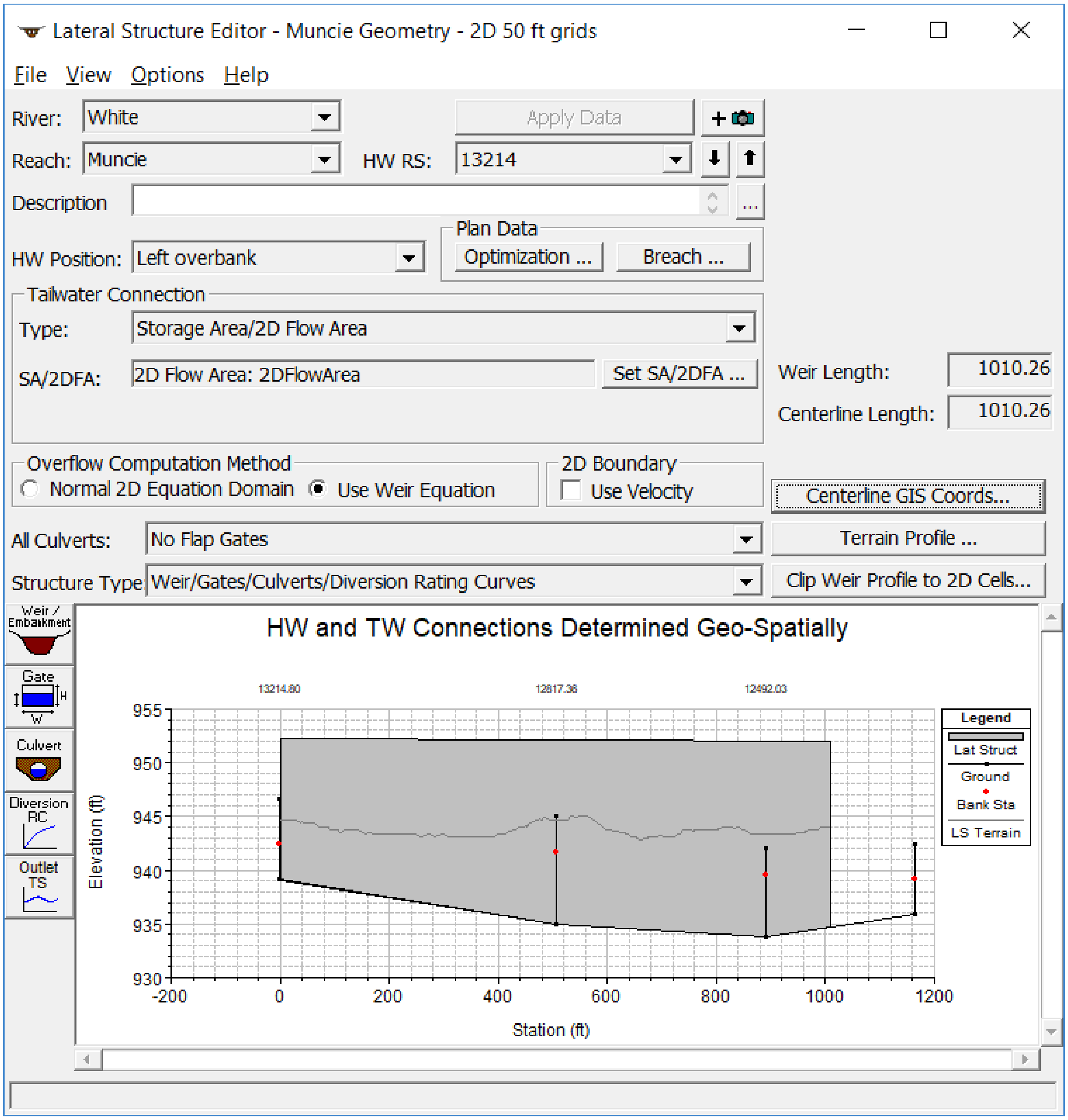


Figure 3-27. Lateral Structure Editor with tailwater connection to a 2D flow area.

3. Next, select the **Weir/Embankment** button on the left side of the graphic window. This will bring up the editor that will allow the user to define the top profile of the embankment, as well as determine how the lateral structure is connected to the 1D river cross sections (the headwater side of the structure), and

to the 2D area face points (the tailwater side of the structure), as shown in Figure 3-28.

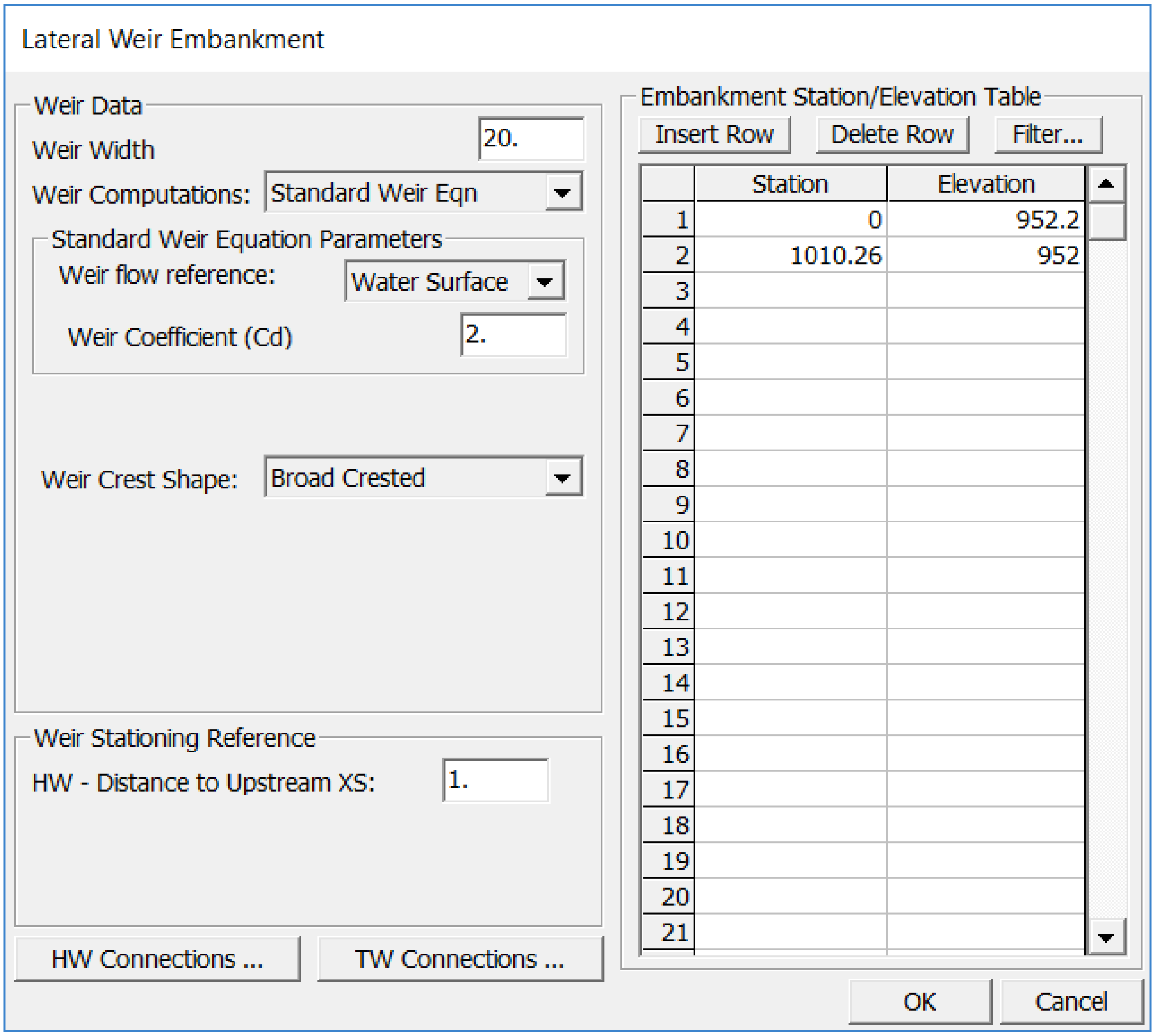


Figure 3-28. Lateral Structure Editor with structure Station/Elevation data, and 2D Face Point stationing.

As shown in Figure 3-28, the user goes about the normal process of entering a Lateral Structure in HEC-RAS by entering the: weir width, weir coefficient, HW (Headwater) Distance to Upstream XS, and the Weir Station and Elevation points. This will define the top of the lateral structure (levee) profile.

For the HW Connection to the 1D cross sections, the user can use the default, which is to have HEC-RAS compute the intersection of the 1D cross sections with the Lateral Structure based on the cross section overbank reach lengths (or based on the lateral structures geospatial data, if the user enters geospatial coordinates for the lateral structure) and the Lateral Structure weir profile stationing (see

Chapter 6 of the HEC-RAS User's Manual, "Entering and Editing Lateral Structure Data" section, for more detailed discussion). Or the user can choose the option called **User Defined Weir Stationing** to enter their own intersection locations between the 1D cross sections and the Lateral Structure Weir Stationing. To view and/or edit the Headwater connection data, press the button labeled **HW Connections** from the Lateral Weir Embankment editor. When you do a window will appear as shown in Figure 3-29. In this example, the 1D river cross sections are being lined up with the Lateral Structure Weir Station automatically by HEC-RAS. This example also shows that geospatial data was entered for the Lateral Structure Centerline, so the option for the user to enter their own connections from the 1D cross sections to the Lateral Structure is not available.

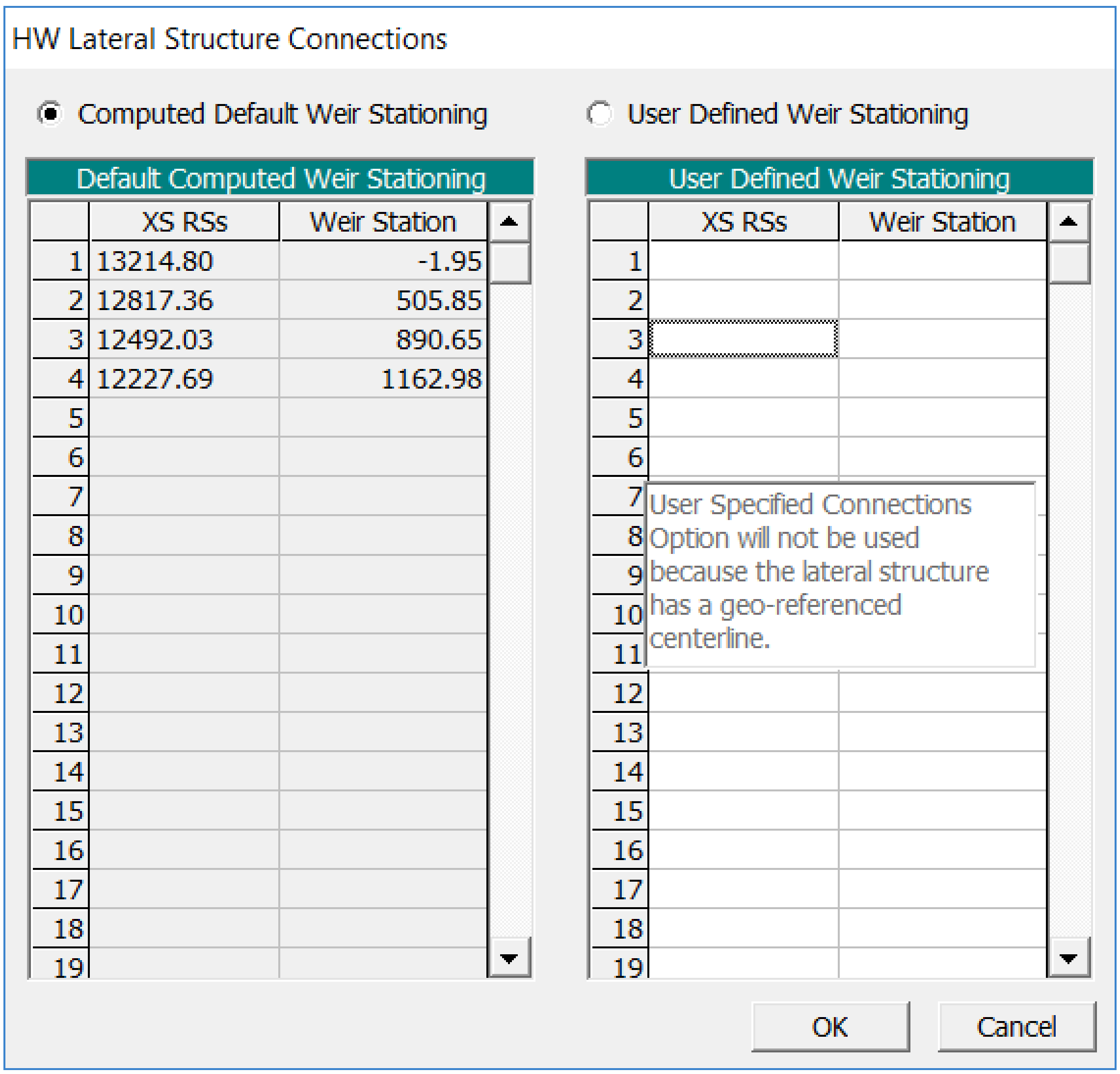


Figure 3-29. Head Water Connection Table for a Lateral Structure.

For the Tailwater Connection to the 2D Flow Area, the user can select either the default (HEC-RAS will compute the connection between the Lateral Structure Weir stationing and the 2D Flow Area Face Points), or they can define their own connection between the Lateral Structure Weir station and the 2D Flow Area Face Points. To view and/or edit the Tailwater connection data, press the button labeled **TW Connections** from the Lateral Weir Embankment editor. When you do a window will appear as shown in Figure 3-30. In this example, the Lateral Structure Weir Station is automatically connected to the 2D Flow Area Face Points by HEC-RAS. This example also shows that geospatial data was entered for the Lateral Structure Centerline, so the option for the user to enter their own connections from the 2D Flow Area Face Points to the Lateral Structure Weir stationing is not available.

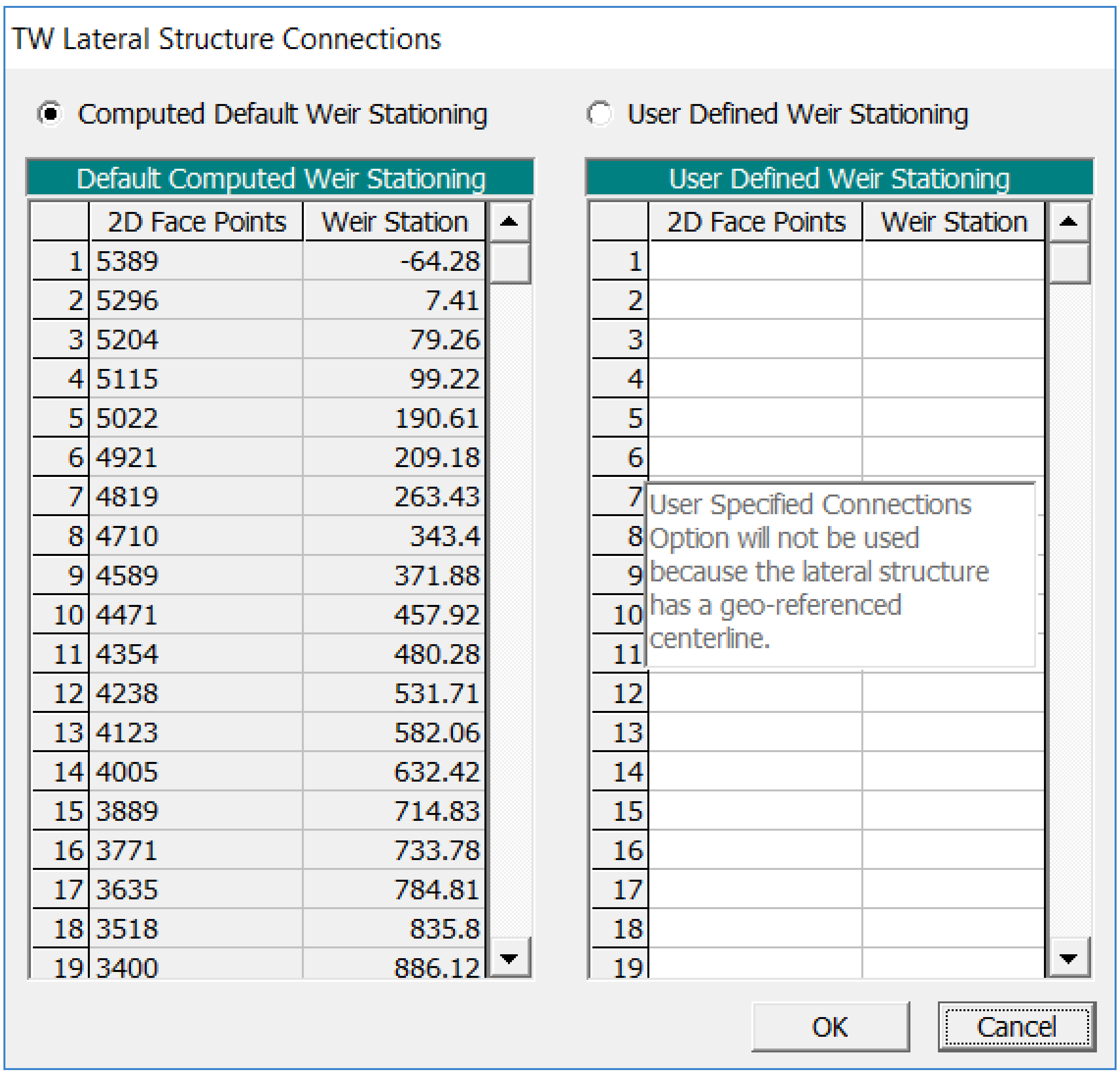


Figure 3-30. Tailwater Connection Table for a Lateral Structure.

4. The last step is to make sure that the 2D flow area Face Points are correctly linked to the stationing of the Lateral Structure. The Tailwater linking is done automatically by HEC-RAS, but the user can override what it does (as described above). By default, the software will come up with the Tailwater Connection table set to **Computed Default Weir Stationing.** In this mode, HEC-RAS will automatically determine the connections between the lateral structure and the 2D flow area. This means HEC-RAS will find the 2D flow area face points that start at the upstream end of the structure and go along the structure to the downstream end. Generally, a lateral structure will not start exactly at a 2D flow area face point. So, HEC-RAS will pick the face point just upstream of the lateral structure to start the connection. This point will normally be given a negative weir stationing, meaning that it is actually upstream of the lateral structure by that distance. So the zero weir stationing is actually in between two face points. The second face point in the table will be the next point downstream and it will have a positive weir stationing. This stationing will represent how far the upstream end of the lateral weir is from that Face Point, along the length/stationing of the lateral weir.

If Geospatial coordinates are not entered for the Lateral Structure, the user has the option to enter the face points and Lateral Structure weir stationing by hand. This is done by selecting **User Defined Weir Stationing** from the Tailwater Connection window. Once the user has selected **User Defined Weir Stationing** they can enter/change/modify the table as they see fit. However, the user must remember not to skip any face points as previously discussed. To connect an HEC-RAS Lateral Structure to a 2D flow area by hand, the user enters face point numbers, from upstream to downstream, that will be linked to the Lateral Structures weir profile stationing. If the Lateral Structure does not begin at a face point, start with the face point that is just upstream of the beginning of the Lateral Structure. Also, continue entering Face Points until you have just gone past the end of the lateral structure. The Face Point numbers must be in the order that they are labeled on the 2D flow area boundary, starting upstream and going downstream**.**

Note: The user cannot skip over (exclude) any of the face point numbers. If any face points along the boundary are skipped the model will not run, and it will give an error message saying the connection to the 2D flow area is incorrect.

Note: If the user makes any changes (such as adding, moving, or deleting cell centers) that cause the cell mesh to be regenerated, the face point numbers and locations may change causing the user entered face point intersections to no longer be valid.

Sometimes, the graphical length of the weir is shown longer or shorter than the true length. This occurs when the lateral structure is on the outside or inside of a bend, and the user has not entered any geospatial information for the Lateral Structure Centerline Table. When this occurs, the HEC-RAS automated Face Point connections will be adjusted such that the Lateral Structure weir stationing lines up with the correct Face

Points. This is done by figuring out the total length along the 2D flow area, from Face Point to Face Point, then proportioning those lengths based on the total Length of the Lateral Structure divided by the total length along the 2D flow area Face Points. If the user chooses to enter the Tailwater connection using **User Defined Weir Stationing**, then the user has to determine the intersections on their own. One way to do this is to measure the lengths along the face points, and then use Excel to reduce the lengths to equal the true structure length, by multiplying the computed face point lengths by the ratio of the true structure length divided by the graphical length of the structure.

Once the user has entered all of the data for the Lateral structure, including the links to the 2D flow area, press the **OK** button to close the Lateral Weir Embankment editor, then close the Lateral Structure editor (unless the user needs/want to add gates, culverts, rating curves, etc… to further define the details of the lateral structure). The HEC-RAS Geometric editor will now show a thick black line along the 2D Area Face Points, to show you where the Lateral Structure is connected to the 2D flow area (see Figure 3-35). If this black line does not follow all of the appropriate Face Points from the 2D flow area, then there is a mistake in the 2D flow area connection table. So the thick black line can be used as a guide to help identify if the Lateral Structure is connected correctly to the 2D flow area. The **Black line** represents the 2D Faces that are connected to the lateral structure. The Black line will always start and stop at the beginning or end of a 2D Face (Face Point). On top of the black line is a **Red dashed line**, the Red line represents where HEC-RAS has linked the lateral structure to the 2D Flow Area boundary. The Red line can start and stop in the middle of a 2D Face. The Red Line shows the user the exact location of how the 2D area is connected to the Lateral Structure.

Shown in Figure 3-31 is how the Lateral structure will look when geospatial data has been entered for the centerline of the structure.

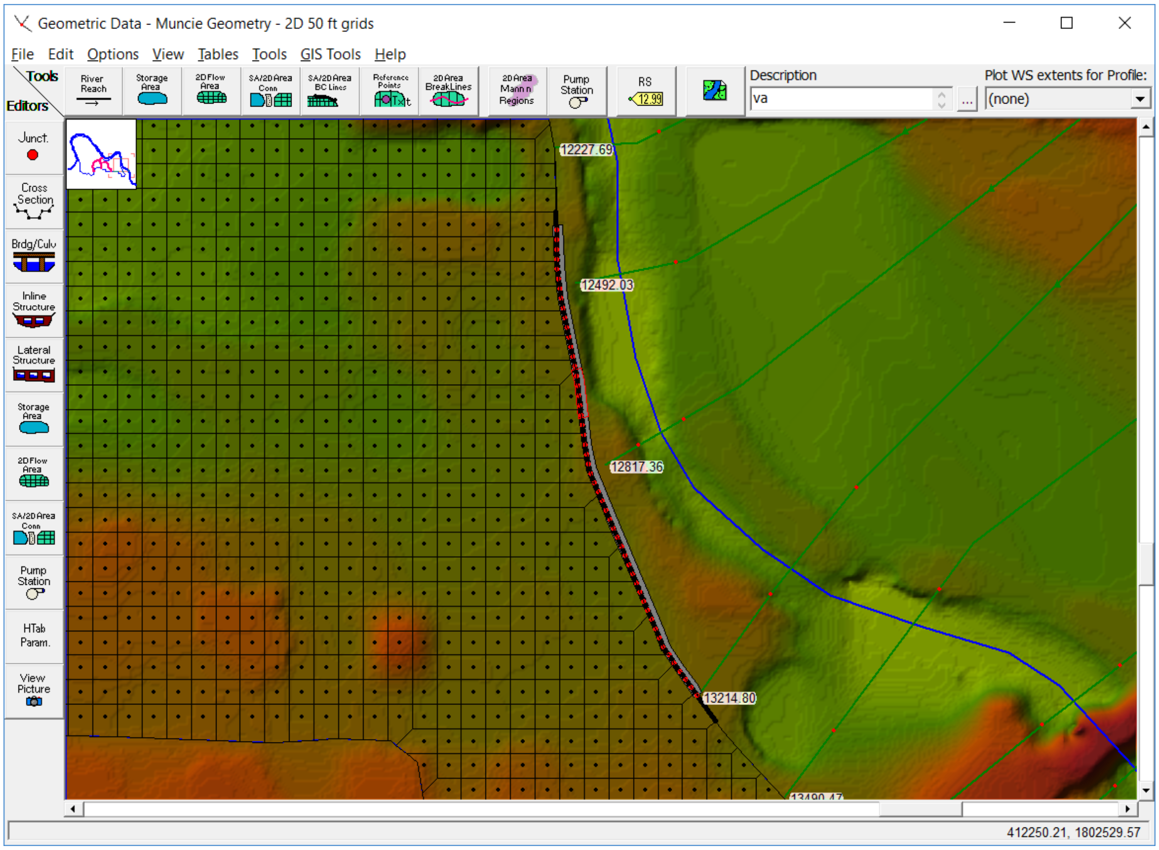


Figure 3-31. HEC-RAS Geometric Editor showing a thick black line for the connection of a Lateral Structure to the 2D flow area Face Points.

For this example a breach location/data has also been added for the analysis of this upper levee. Shown in Figure 3-32 is the Levee (Lateral Structure) Breach Data editor with the data for this levee.

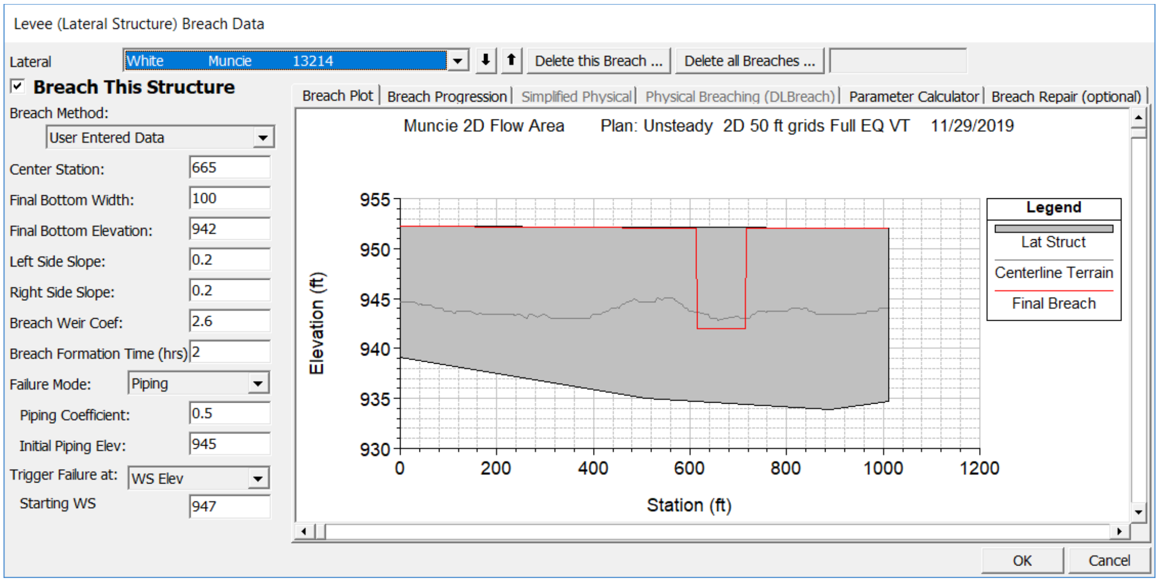


Figure 3-32. Levee (Lateral Structure) Breach Data editor, with breach data for the upstream levee.

For this specific example, a second additional lateral structure at the lower end of the 2D flow area will be added. This Lateral Structure will be used to model flow that ponds on the inside of the protected area, then flows back over the top of the Levee (Lateral Structure) into the 1D river system. See Figure 3-33 for the extents of this downstream Levee.

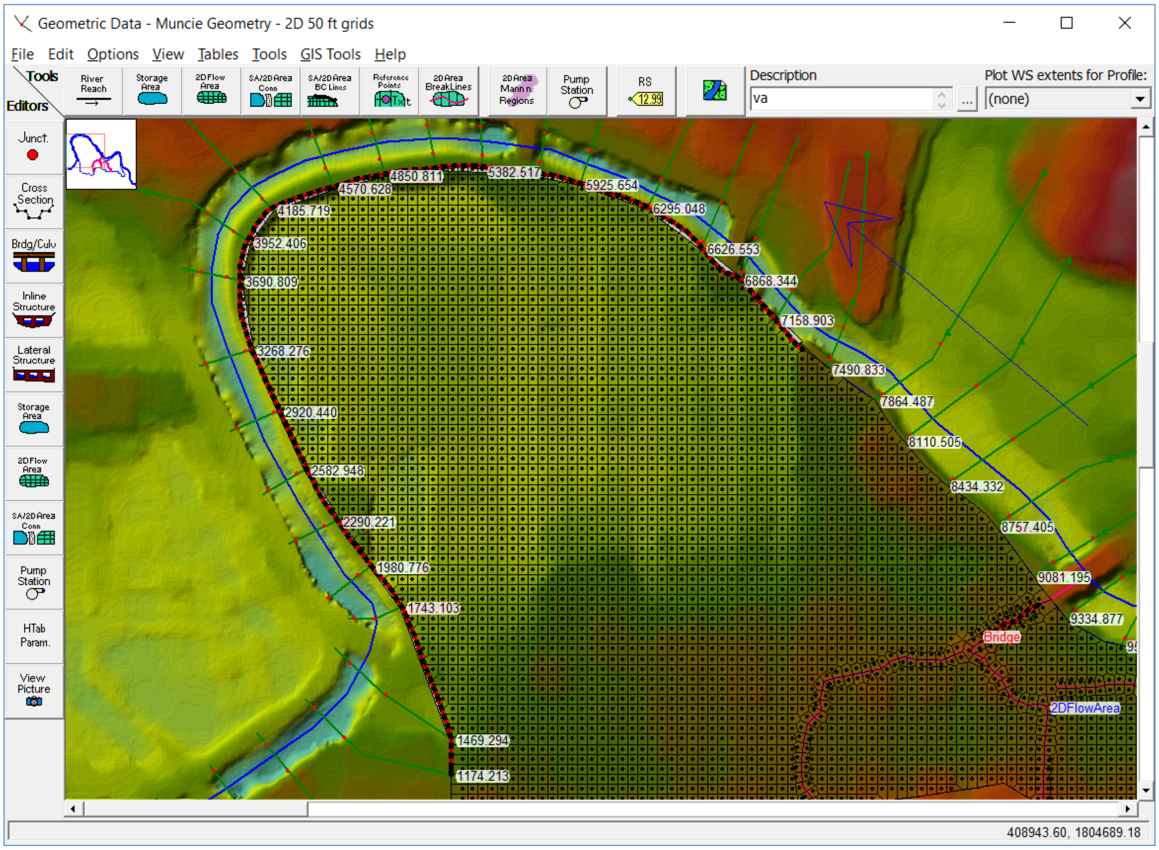


Figure 3-33. HEC-RAS Geometric Editor with downstream Levee (Lateral Structure) highlighted in red.

The downstream Lateral Structure in this example starts at River Station 7300, and goes along the entire downstream boundary of the protected area, tying back into high ground at the downstream end. The Lateral Structure editor for this Levee is shown in Figure 3-34.

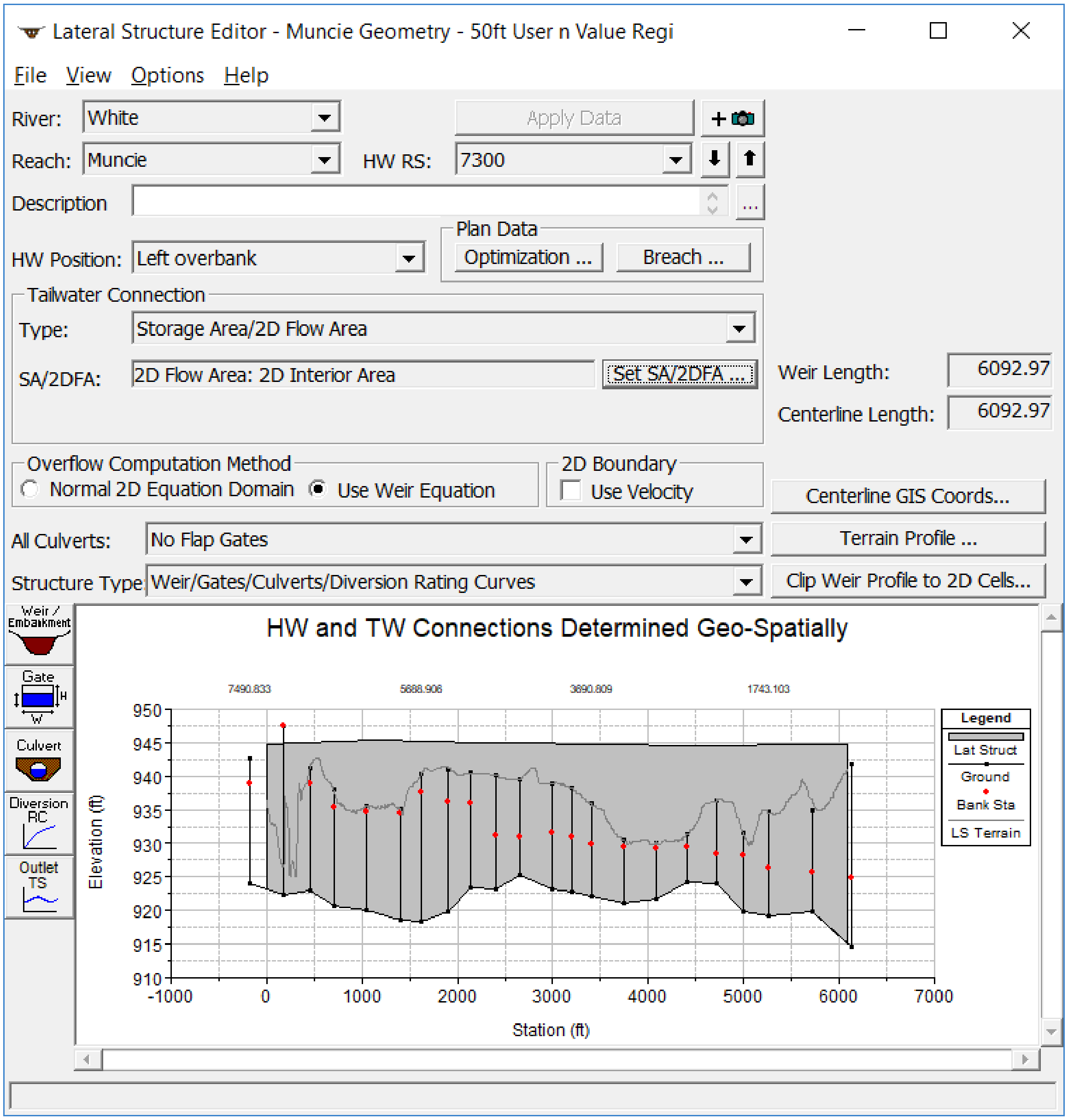


Figure 3-34. Downstream Levee (Lateral Structure) with a tailwater connection to the 2D flow area.

Shown in Figure 3-35, is the Weir Embankment editor, with the data for the Lateral Structure stationing and elevations, as well as access to the Lateral Structure linking to the 1D cross sections (HW Connections) and the 2D flow area Face Points (TW Connections).

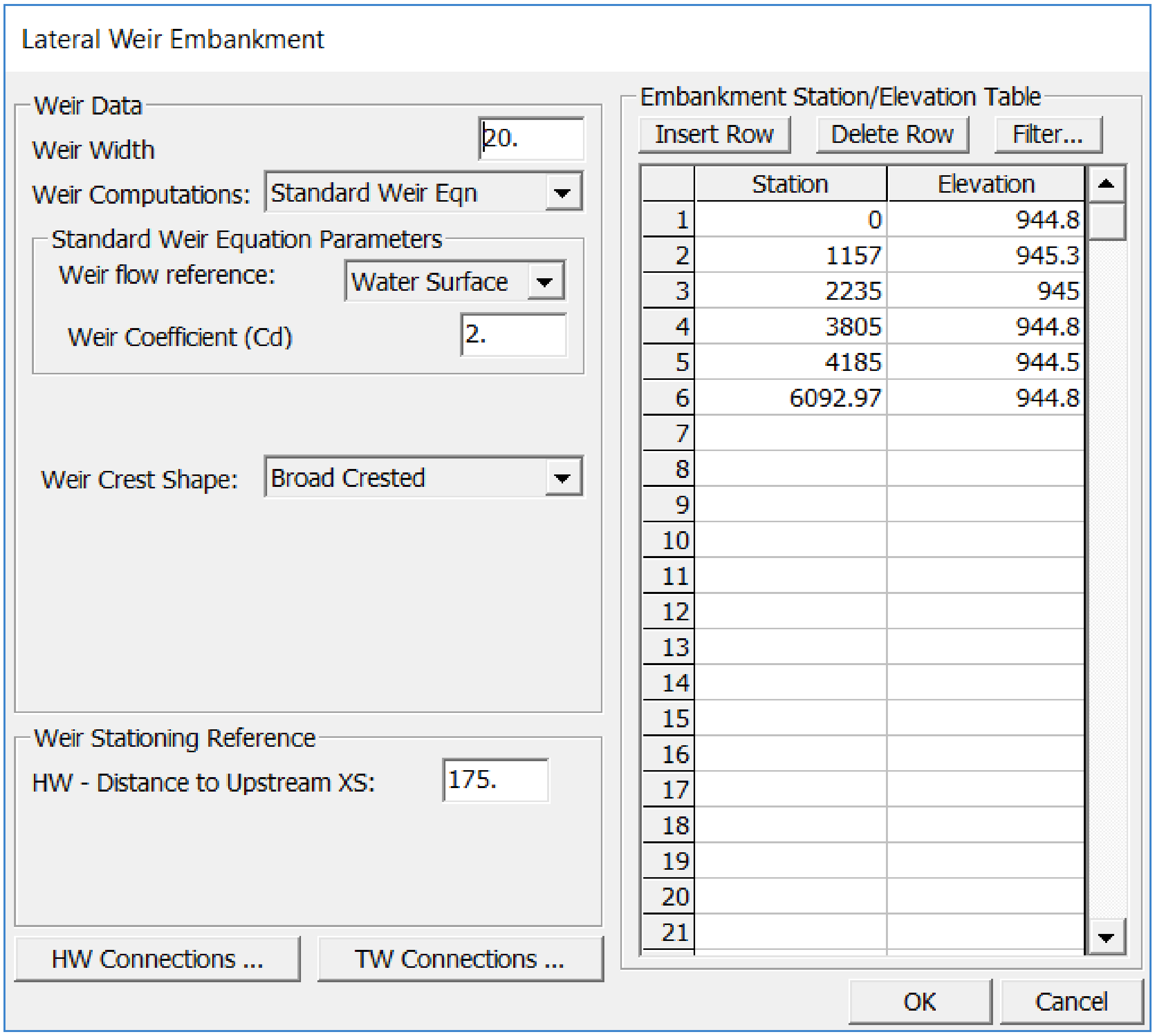


Figure 3-35. Lateral Weir Embankment editor with data for the downstream levee and linked to the 2D flow area Face Points.

Lateral Structure Weir Coefficients

In general, Lateral Structure weir coefficients should be lower than typical values used for inline weirs. Additionally, when a lateral structure (i.e. weir equation) is being used to transfer flow from the river (1D region) to the floodplain (2D flow area), then the weir coefficients that are used need to be very low, or too much flow will be transferred. Below is a table of rough guidelines for Lateral weir coefficients under different conditions:

Table 3-1. Lateral Weir Coefficients

What is being modeled with the Lateral Structure	Description	Range of Weir Coefficients
Levee/Roadway – 3ft or higher above natural ground	Broad crested weir shape, flow over levee/road acts like weir flow	**1.5** to **2.6** (2.0 default) SI Units: 0.83 to 1.43
Levee/Roadway – 1 to 3 ft elevated above ground	Broad crested weir shape, flow over levee/road acts like weir flow, but becomes submerged easily.	**1.0** to **2.0** SI Units: 0.55 to 1.1
Natural high ground barrier – 1 to 3 ft high	Does not really act like a weir, but water must flow over high ground to get into 2D flow area. Flow does not pass through critical depth	**0.5** to **1.0** SI Units: 0.28 to 0.55
Non elevated overbank terrain. Lat Structure not elevated above ground	Overland flow escaping the main river.	**0.2** to **0.5** SI Units: 0.11 to 0.28

Note: The biggest problem HEC-RAS users have when interfacing 1D river reaches with 2D flow areas is using a weir coefficient that is too high for the situation being modeled. If the lateral structure is really just an overland flow interface between the 1D river and the 2D floodplain, then a weir coefficient in the range of 0.2 to 1.0 should be used to get the right flow transfer and keep the model stable. However, weir coefficients should be calibrated to produce reasonable results whenever possible.

Note: A second issue is weir submergence. When a lateral structure gets highly submerged, HEC-RAS uses a weir submergence curve to compute the flow reduction over the weir. The curve is very steep (i.e. the flow reduction changes dramatically) between 95% and 100% submergence. This can cause oscillations and possible model stability issues. To reduce these oscillations, user can have HEC-RAS use a milder loping submergence curve by going to the 1D “Computational Options and Tolerances” and setting the field labeled “**Weir flow submergence decay exponent”** to 3.0.

Directly Connecting an Upstream River Reach to a Downstream 2D flow area

Users can connect a 1D River Reach directly to a 2D flow area. When this type of boundary condition is used, the last cross section of the 1D River Reach must be lined up with the upstream boundary of the 2D flow area (i.e., the last cross section of the 1D reach is directly linked to the boundary of the 2D area, so they need to be at the same exact location). See the example shown in Figure 3-36.

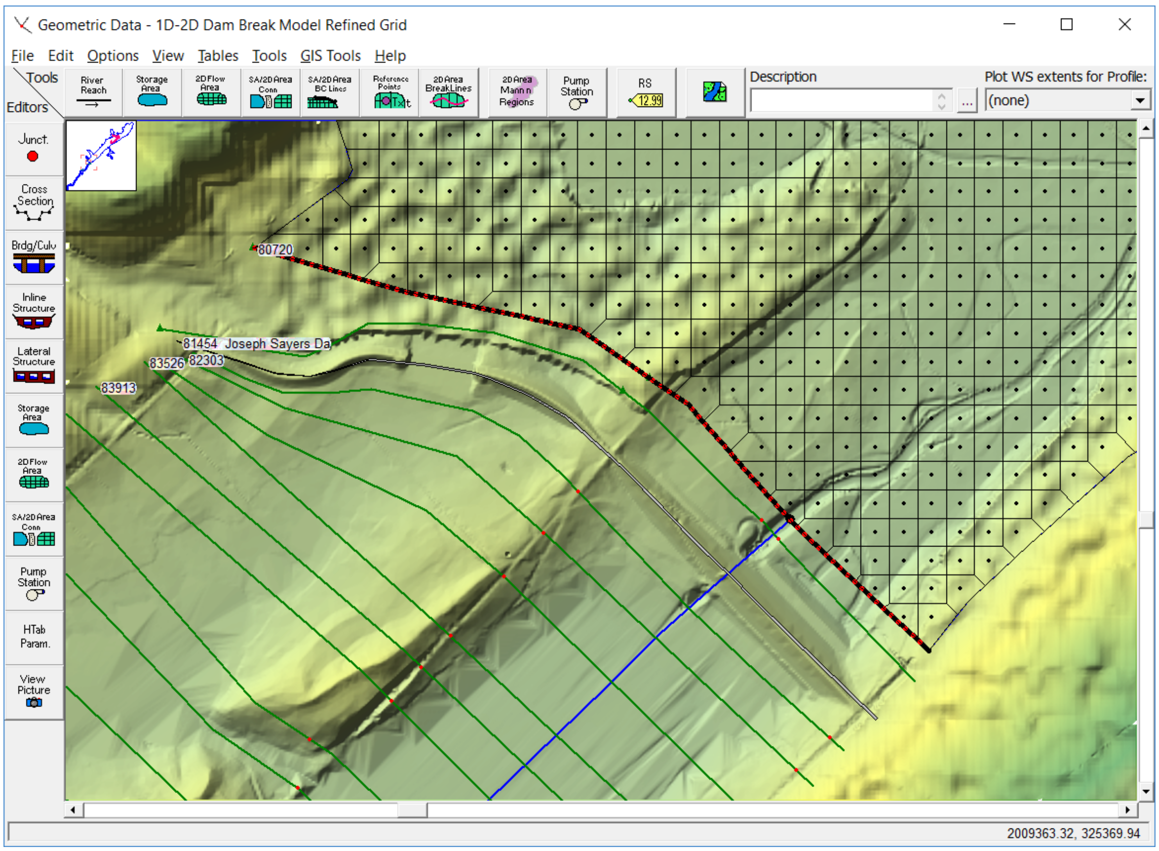


Figure 3-36. Example of an upstream 1D River Reach connected to a downstream 2D flow area.

For this type of boundary condition, the 1D River Reach passes flow each time step to the 2D flow area, while the stage in the cross section is based on the water surface elevation in the 2D cells that it is connected to. Flow is distributed to the 2D cells based on the conveyance distribution in the cross section, and the stationing of the cells linked to the cross section. The computed stage for the 1D cross section is based on computing a conveyance weighted stage from the connected boundary cells in the 2D flow area, and then forcing that stage on the 1D cross section each time step.

This type of boundary condition should only be placed in areas where the flow and stage are **highly one-dimensional in nature**. If the flow is not highly one-dimensional, you may run in to model stability issues with this type of boundary condition. User's can try turning on the option to allow the program to iterate back and forth between the 1D and the 2D computations during each time step, until the computed flow and stage at the boundary connection converges within a user specified tolerance. This may not solve the issue for areas that are highly two-dimensional (i.e. the water surface varies significantly). If the flow is highly one dimensional, 1D to 2D iterations are generally not necessary for this type of boundary condition.

To connect a 1D river reach to a 2D flow area, do the following:

- Draw the 2D area polygon such that the outer boundary at the upstream end is right on top of the last cross section of the 1D River Reach.
- Go to the **Edit** menu of the **Geometric Data editor**, and turn on the Option to **Move Points/Objects**.
- Move the last point of the stream centerline inside of the 2D flow area. The software will ask you if you want to connect the 1D River Reach to the 2D flow area. Select **Yes**.

This type of connection between a 1D cross section and a 2D area requires the following to be true:

- The location for this type of connection should be placed where the flow is highly one-dimensional (water surface is relatively horizontal and flow lines are perpendicular to the 1D cross section.
- The 1D cross section is exactly on top of the boundary of the 2D area that it is connected too.
- The terrain defining the 1D cross section must be exactly the same as the terrain along the boundary of the 2D flow area where it is connected to the 1D cross section.
- The Manning's roughness coefficients must be exactly the same spatially along the cross section and the 2D flow area boundary that it is connected too.

Once the 2D flow area and the 1D River Reach are connected, the software will draw a black line along the 2D flow area cells outer boundary to show the user how it is connected. The Black line represents the 2D cell Faces that the 1D cross section is connected to. A red line is drawn on top of the black line. The red line represents what HEC-RAS thinks is the exact location of where the 1D cross section starts and stops along the 2D Flow Area boundary. The red line is what HEC-RAS is using to figure out what portion of the 1D cross section corresponds to the 2D Flow Area Faces. That is all that needs to be done for the connection.

When a 1D River Reach is connected to a 2D area, the user will need to define the initial conditions for the 1D Reach and the 2D area. Initial conditions for the 2D Area can be: set to a single water elevation; set using a "Restart" file from a previous run; or the user can select to run a warm-up period at the beginning of the run, in which flow and stage boundaries connected to the 2D area will be applied slowly over time.

NOTE: when a 1D reach is connected to a 2D area, the 2D area must have water in it at the connection zone. If it does not the model will go unstable right away.

The 2D unsteady flow solver can handle “wetting” and “drying” of cells. However, the original finite difference 1D unsteady flow solver cannot handle “dry” cross sections. Therefore, a “wet” water surface at the 1D/2D boundary must be established at the beginning of the run and maintained during the simulation. If a restart file is not used, then HEC-RAS will compute the starting water surfaces in two distinct parts. The first part is the “initial condition” phase.

For the initial condition phase, water surfaces are determined for any 2D areas that start wet and the initial backwater is determined for all 1D reaches. Every 2D area that has a direct connection to a 1D reach must have enough water in it to provide a water surface at any and all of the 1D boundary connections. The user can specify a starting 2D water surface and/or use the 2D Initial Conditions Ramp Up option (This is highly recommended). To set the 2D Initial Conditions Ramp Up option, go to the **Unsteady Flow Analysis** window, then Select **Calculation Options and Tolerances** from the **Options** menu. Select the **2D Flow Options** Tab. Enter a time in hours for each of the 2D flow areas in the filed labeled **Initial Conditions Time (hrs)**. This is the time each 2D flow area will run on its own at the beginning in order to establish a good initial conditions within that 2D Flow Area.

During this phase, the program may cycle between 2D areas and 1D reaches in order to determine the flows and water surfaces at the boundaries. If the upstream end of a 1D reach is connected directly to a 2D area and the user has specified an initial flow for this reach, then the program will use that flow during the initial conditions. If an initial flow is not specified, then the program will attempt to determine this flow automatically. If the downstream end of a 1D reach is connected, then there is nothing for the user to specify. The program will try to determine the water surface in the 2D area first which will allow for a stage boundary for the 1D reach. If this is not possible, a critical depth boundary may be initially used.

The second part of determining the starting water surfaces is the optional warm-up period. This is the same warm-up period that 1D has always had except that it now also includes the 2D areas. As explained above, there must be a valid water surface at the 1D/2D boundaries before the warm-up period and/or the main simulation starts.

Directly Connecting an Upstream 2D flow area to a Downstream River Reach

Users can directly connect an upstream 2D flow area to a downstream 1D River Reach. When this type of boundary condition is used, the first cross section of the 1D River Reach must be lined up with the downstream boundary of the 2D flow area (i.e., the first cross section of the 1D reach is directly linked to the downstream boundary of the 2D area, so they need to be at the same exact location). See the example shown in Figure 3-37.

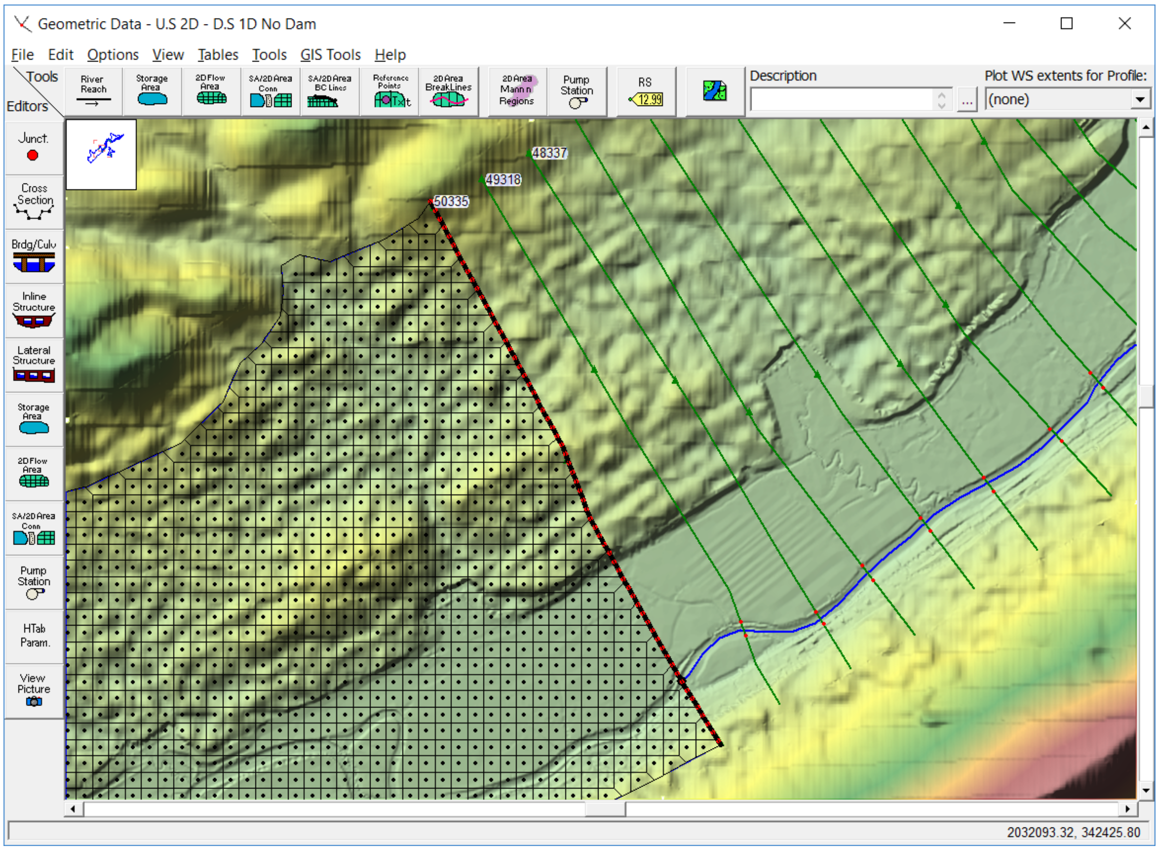


Figure 3-37. Example of an upstream 2D flow area connected to a downstream 1D River Reach.

For this type of boundary condition, the 2D flow area passes flow each time step to the 1D river reach, while the stage in the 2D flow area is based on the stage in the 1D cross section that it is connected too. Flow is passed to the 1D section by adding all of the flows leaving the 2D cells at the boundary for each time step. The stage for the 2D flow area downstream boundary is set to the computed stage of the 1D cross section each time step.

This type of boundary condition should only be placed in areas where the flow and stage are highly one-dimensional in nature. If the flow is not highly one-dimensional, the user will need to turn on the option to allow the program to iterate back and forth between the

1D and the 2D computations during each time step, until the computed flow and stage at the boundary connection does not change within a user specified tolerance. Even if the flow is highly one-dimensional, 1D to 2D iterations may be necessary for this type of boundary condition, depending on how quickly the flow and stage are changing, compared to the user selected computation interval.

To connect an upstream 2D flow area directly to a downstream 1D River Reach, do the following:

- Draw the 2D flow area polygon such that the outer boundary at the downstream end is right on top of the first cross section of the 1D river reach.
- Go to the **Edit** menu of the **Geometric Data editor**, and turn on the Option to **Move Points/Objects**.
- Move the first point of the stream centerline inside of the 2D flow area. The software will ask you if you want to connect the 1D River Reach to the 2D flow area. Select **Yes**.

This type of connection between a 2D area and a 1D cross section requires the following to be true:

- The location for this type of connection should be placed where the flow is highly one-dimensional (water surface is relatively horizontal and flow lines are perpendicular to the 1D cross section.
- The 1D cross section is exactly on top of the boundary of the 2D area that it is connected too.
- The terrain defining the 1D cross section must be exactly the same as the terrain along the boundary of the 2D flow area where it is connected to the 1D cross section.
- The Manning's roughness coefficients must be exactly the same spatially along the cross section and the 2D flow area boundary that it is connected too.

Once the 2D flow area and the 1D River Reach are connected, the software will draw a black line along the 2D flow area cells outer boundary to show the user how it is connected. The Black line represents the 2D cell Faces that the 1D cross section is connected to. A red line is drawn on top of the black line. The red line represents what HEC-RAS thinks is the exact location of where the 1D cross section starts and stops along the 2D Flow Area boundary. The red line is what HEC-RAS is using to figure out what portion of the 1D cross section corresponds to the 2D Flow Area Faces. That is all that needs to be done for the connection.

Note: When a 2D area is connected to a 1D River Reach, the user will need to define the initial conditions of the 1D Reach and 2D area. Initial conditions for an upstream 2D flow area cannot be dry. The initial conditions for an upstream 2D area can be: set to a single water elevation; set with a "Restart" file from a previous run; or the user can select to run a warm-up period at the beginning of the run, in

which flow and stage boundaries connected to the 2D area will be applied slowly over time.

For the initial condition phase, water surfaces are determined for any 2D areas that start wet and the initial backwater is determined for all 1D reaches. Every 2D area that has a direct connection to a 1D reach must have enough water in it to provide a water surface at any and all of the 1D boundary connections. The user can specify a starting 2D water surface and/or use the 2D Initial Conditions Ramp Up option (This is highly recommended). To set the 2D Initial Conditions Ramp Up option, go to the **Unsteady Flow Analysis** window, then Select **Calculation Options and Tolerances** from the **Options** menu. Select the **2D Flow Options** Tab. Enter a time in hours for each of the 2D flow areas in the filed labeled **Initial Conditions Time (hrs)**. This is the time each 2D flow area will run on its own at the beginning in order to establish a good initial conditions within that 2D Flow Area.

Connecting a 2D flow area to a Storage Area using a Hydraulic Structure

A 2D flow area can be directly connected to Storage Area by using a hydraulic structure called a **Storage Area/2D flow area Hydraulic Connector** (**SA/2D Area Conn).** See the example in Figure 3-38 below.

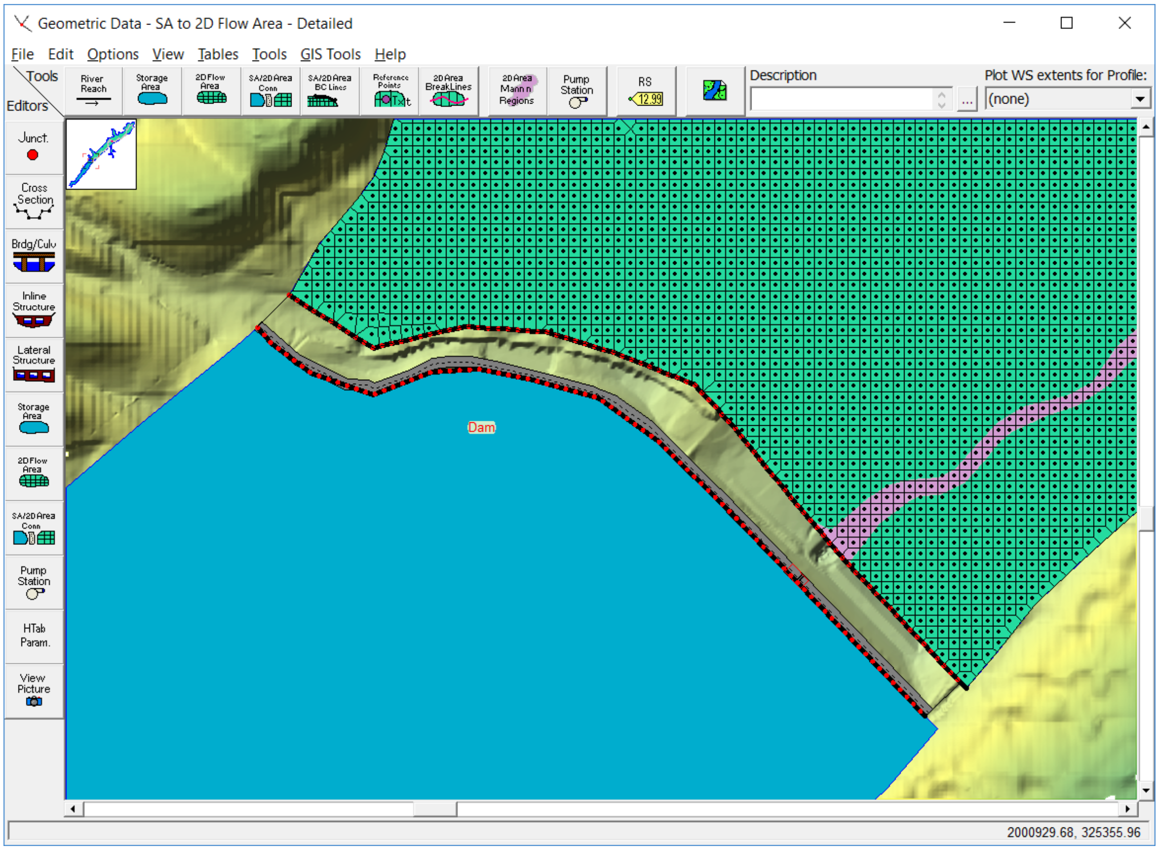


Figure 3-38. Example of a Storage Area connected to a 2D flow area.

In the example shown in Figure 3-38, the Storage Area is upstream of the 2D flow area, so the positive flow direction is from the storage area to the 2D flow area. Therefore, when defining the hydraulic structure that connects the two areas, the Storage Area will be considered the Headwater side, and the 2D flow area will be considered the Tailwater side. This can also be done the other way, in which the 2D flow area is on the upstream side (Headwater) and the Storage Area is on the downstream side (Tailwater). For the example shown in Figure 3-38, a Storage Area is being used to represent a reservoir pool. The hydraulic connection between the Storage Area and the 2D flow area is a dam (SA/2D Area Hydraulic Connection) in this example. The 2D flow area is being used to model the hydraulics of the flow downstream of the dam.

To hydraulically connect a Storage Area to a 2D flow area, do the following:

- Draw the storage area polygon right up to the edge of the hydraulic structure. This can be as close to the hydraulic structure as desired for mapping purposes.
- Draw the outer boundary of the 2D flow area right up to the other side of the hydraulic structure. This can also be very close to the hydraulic structure. However, keep in mind that the computed water surface elevations of the boundary cells of the 2D area will be used in the hydraulic calculations over/through the structure (don't put very small cells down the face of a steep embankment). Generally, the water surface computed for the 2D cells should represent what the user wants for the water surface in the hydraulic calculations of flow over and through the hydraulic structure. That is, don't put very small cells down the face of a steep embankment because the small boundary cells may end up with a transitional water surface that is between the "headwater" and the "tailwater" surfaces. If this happens, the accuracy of the hydraulic computations across the structure may be reduced.
- Select the drawing tool at the top of the Geometric Data editor labeled **SA/2D Area Conn**. Then draw a line directly down the center of the hydraulic structure that will be used to connect the two flow areas. Draw this line from left to right looking downstream. This is how HEC-RAS will detect what is upstream (headwater) and what is downstream (tailwater). The interface will ask for a label to define the hydraulic structure. See the red line shown in Figure 3-38.
- Next, select the Storage Area/ 2D flow area Hydraulic Connection (**SA/2D Area Conn**) editor on the left panel of the Geometric Data editor. This will bring up the editor shown in Figure 3-39.

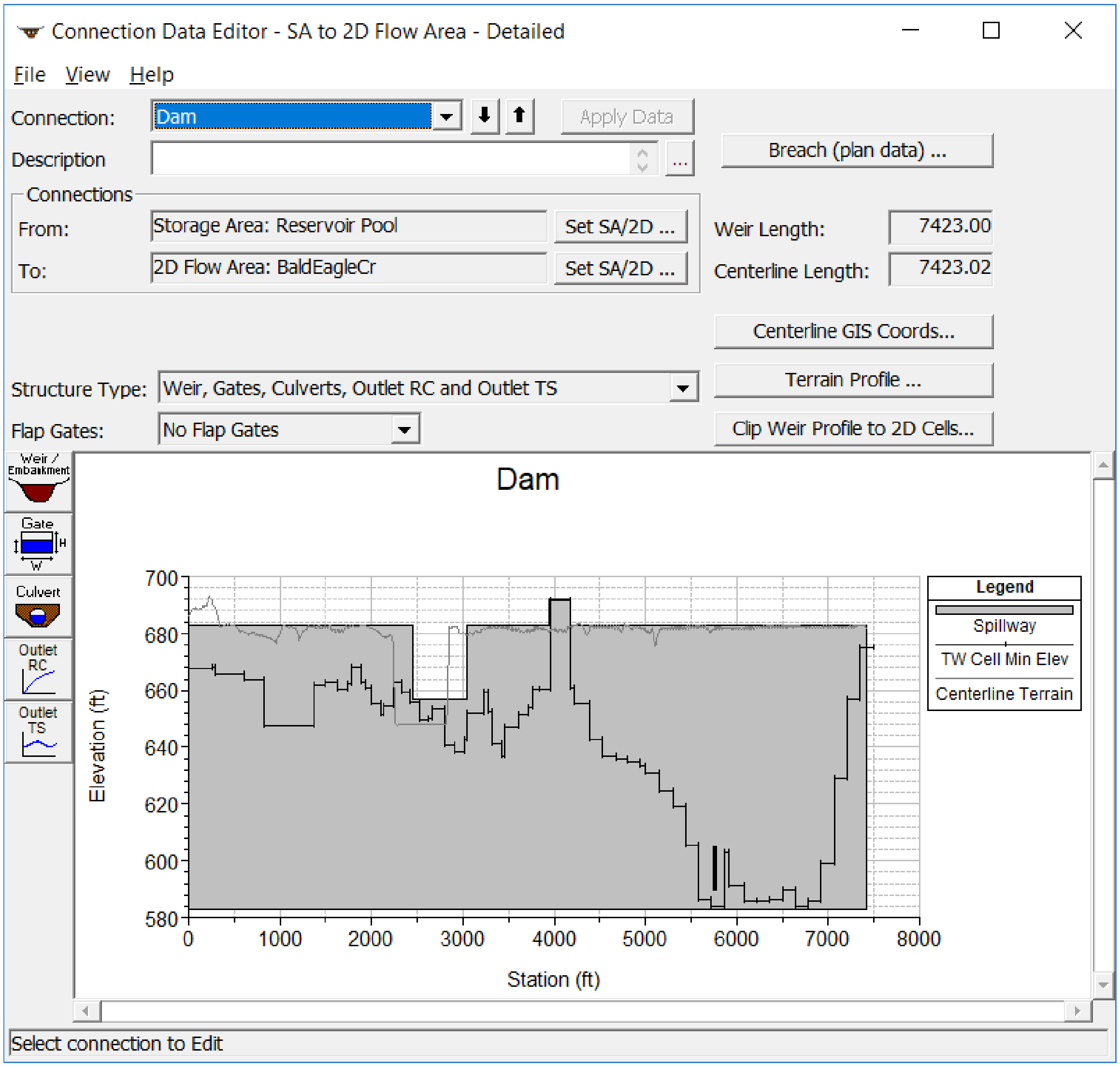


Figure 3-39. SA/2D Area Hydraulic Connection editor.

- On the **SA/2D Area Conn** editor set the **From** and **To** by selecting the buttons labeled **Set SA/2D Area**. For this example, the storage area labeled “Reservoir Pool” is the **From** element, and the 2D flow area labeled “BaldEagleCr” is the **To** element.
- Enter all the hydraulic structure information for the connection. This will consist of a Weir/Embankment profile, and any additional hydraulic outlets, such as culverts, gates, etc… In the example shown in Figure 3-39, there is an embankment with an emergency spillway defined, and there are also two low flow gates defined.

Geospatial coordinates can be entered can be entered for any of the outlet types avasilable in the SA/2D Area Connection editor (ie. Culvert, gates, rating curves, and time series outlet). This will allow the user to specify a location within the 2D area that you want that specific outlet to connect to, such as an interior cell instead of one of the boundary cells.

This is all that is needed for this type of hydraulic connection. HEC-RAS automatically computes the stationing along the centerline drawn for the hydraulic structure, and then lines it up with the outer boundary of the 2D flow area based on their spatial location. The connection to the Storage Area is very simple, since it can only have a single water surface elevation inside the storage area each time step.

Connecting a 2D flow area to another 2D flow area using a Hydraulic Structure

2D flow areas can be directly connected to other 2D flow areas by using a hydraulic structure called a Storage Area/ 2D flow area Hydraulic Connector (**SA/2D Area Conn).** See the example below in Figure 3-40.

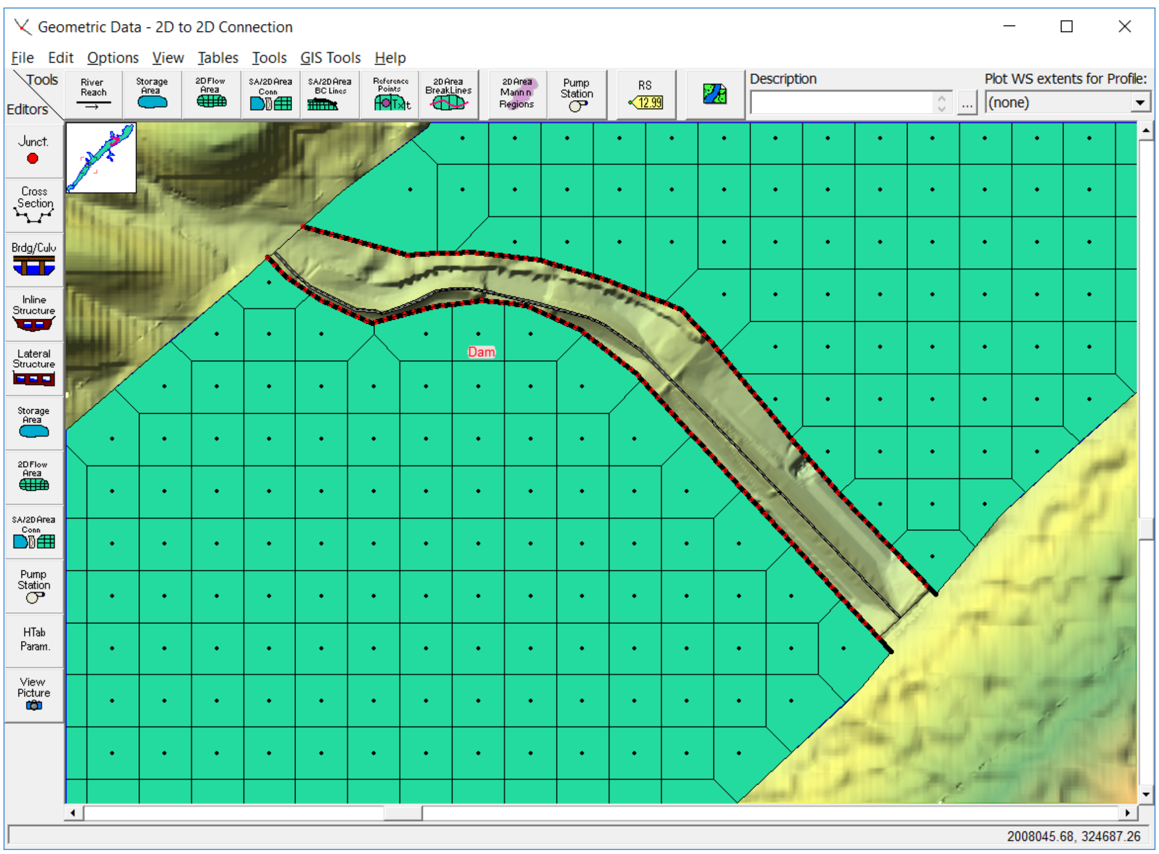


Figure 3-40. Example of connecting one 2D flow area to another 2D flow area with a Hydraulic Structure.

In the example shown in Figure 3-40, there is a 2D flow area upstream of another 2D flow area, so the positive flow direction is from the upstream 2D flow area to the downstream 2D flow area. When defining the hydraulic structure that connects the two areas, the upstream 2D flow area will be considered the headwater side, and the downstream 2D flow area will be considered the tailwater side. The hydraulic connection between the two 2D flow areas is a dam in this example. The downstream 2D flow area is being used to model the hydraulics of the flow downstream of the dam.

To hydraulically connect one 2D flow area to another 2D flow area, do the following:

- Draw the upstream 2D flow area polygon right up to the edge of the hydraulic structure. This should be relatively close to the hydraulic structure for mapping purposes.
- Draw the outer boundary of the downstream 2D flow area right up to the other side of the hydraulic structure. This can also be very close to the hydraulic structure, however, keep in mind that the computed water surface elevations of the boundary cells of the 2D area will be used in the hydraulic calculations over/through the structure (i.e., don't put very small cells down the face of a steep embankment). Generally, the 2D cells computed water surfaces should represent what the user wants to be used in the hydraulic calculations of flow over and through the hydraulic structure.
- Select the drawing tool at the top of the Geometric Data editor labeled **SA/2D Area Conn**. Then draw a line directly down the center of the hydraulic structure that will be used to connect the two flow areas. The interface will ask the user for a label to define the hydraulic structure. See the red line shown in Figure 3-40.
- Next, select the **Storage Area/ 2D flow area Hydraulic Connection** (**SA/2D Area Conn**) editor on the left panel of the **Geometric Data Editor**. This will bring up the editor shown in Figure 3-41.

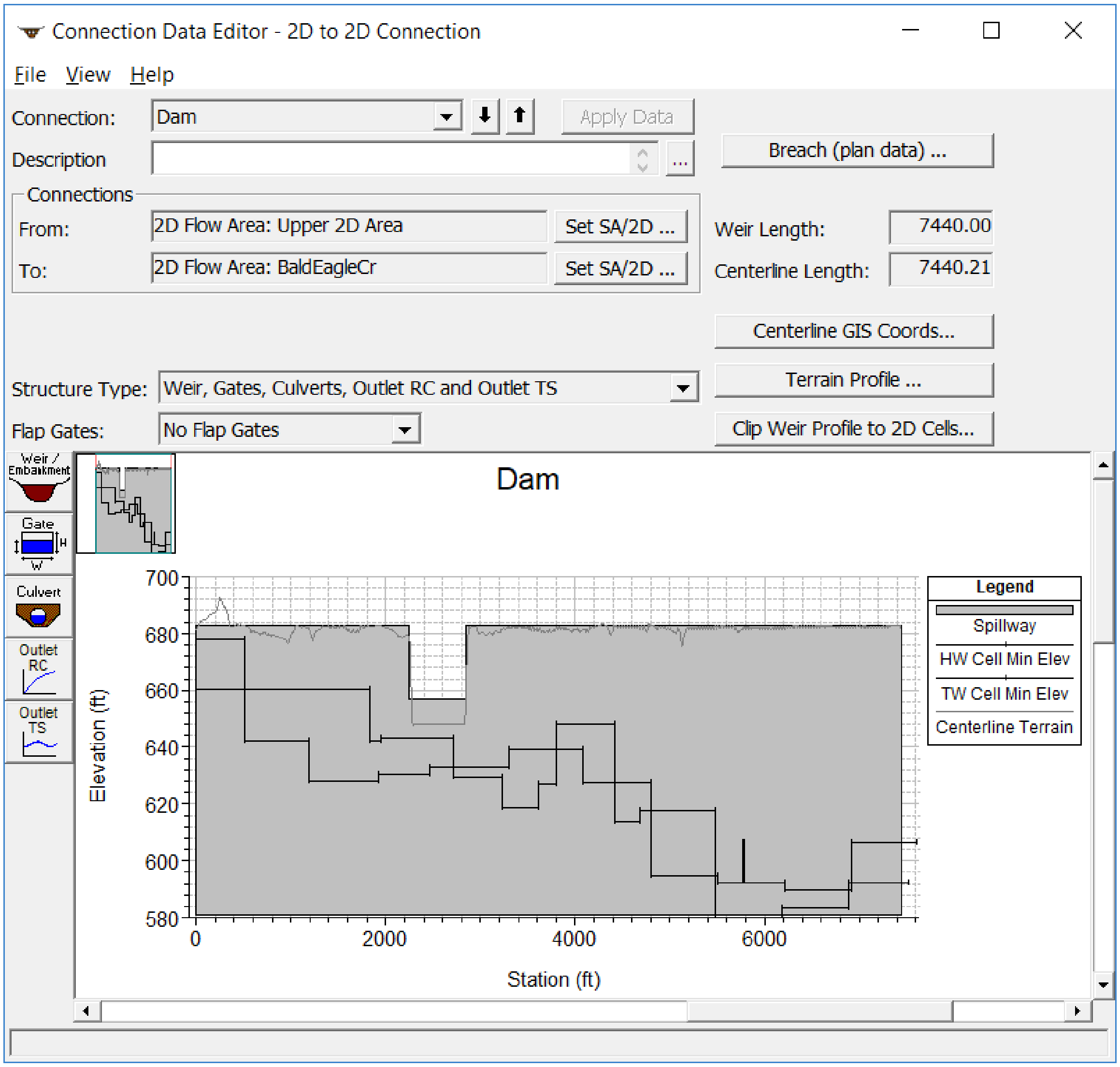


Figure 3-41. SA/2D Area Hydraulic Connection editor.

- On the **SA/2D Area Conn** editor set the **From** and **To** by selecting the buttons labeled **Set SA/2D Area**. In this example the upstream 2D flow area labeled "Upper 2D Area" is the **From** element, and the 2D flow area labeled "BaldEagleCr" is the **To** element.
- Enter all the hydraulic structure information for the connection. This will consist of a weir/embankment profile, and any additional hydraulic outlets, such as culverts, gates, etc… In the example shown in Figure 3-45, there is an embankment with an emergency spillway defined, and there are also two low flow gates defined.

This is all that is needed for this type of hydraulic connection. HEC-RAS automatically figures out the stationing along the centerline drawn for the hydraulic structure, and then lines it up with the outer boundary of the upstream and downstream 2D flow areas based on their spatial location.

Multiple 2D flow areas in a Single Geometry File

HEC-RAS has the ability to have any number (within the computer's memory limitations) of separate 2D flow areas within the same geometry file. Multiple 2D flow areas can be added in the same way as storage areas. Hydraulic connections can be made from 2D flow areas to 1D Elements, as well as between 2D flow areas. See the example in Figure 3-42.

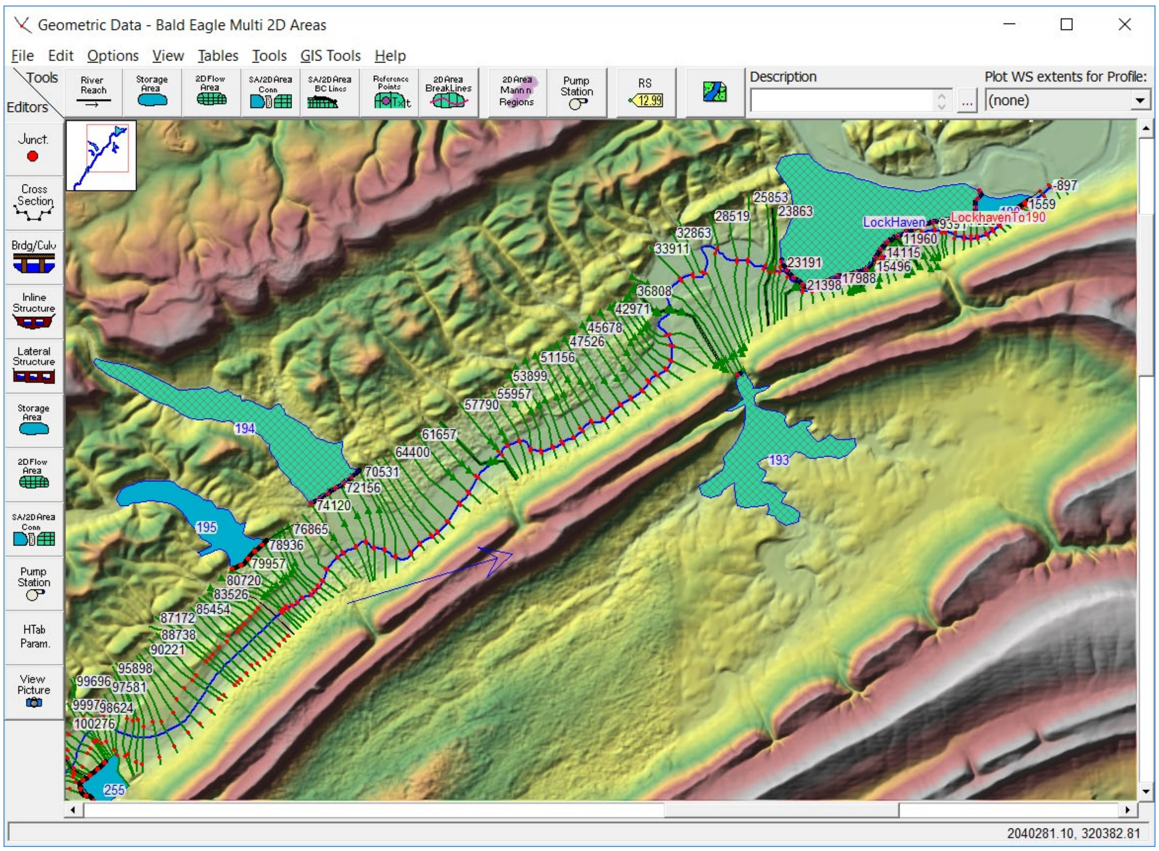


Figure 3-42. Multiple 2D flow areas in a single geometry file.

Hydraulic Structures Inside of 2D flow areas

HEC-RAS has the ability to add hydraulic structures inside of 2D flow areas. This is accomplished by using the **SA/2D Area Conn** option to make a hydraulic structure in the middle of a single 2D flow area. The hydraulic structure acts just like a breakline, in that users can align the cells along the structure, as well as control the cell size along the structure. See Figure 3-43.

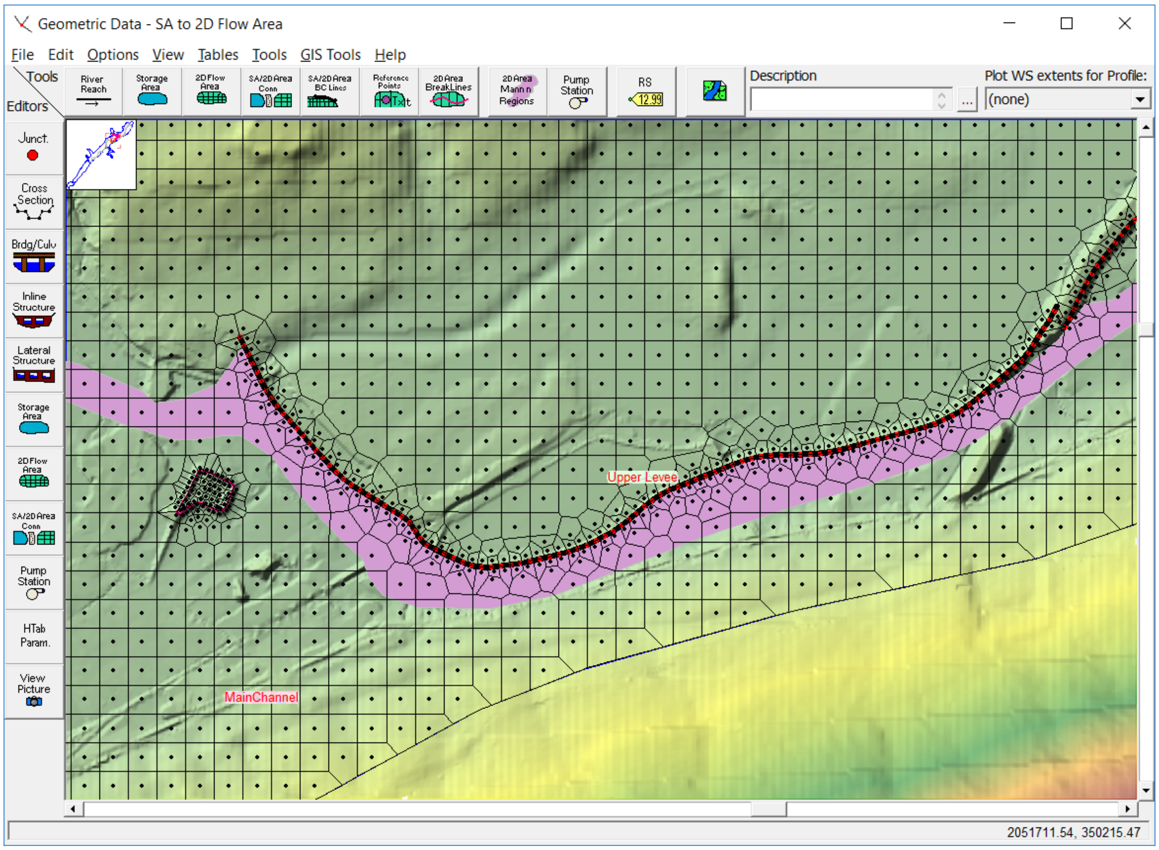


Figure 3-43. Example hydraulic structure inside of a 2D flow area.

The SA/2D Hydraulic Connection can be used to model all kinds of structures inside of a 2D Flow Area. Flow going over top of the structure can be modelled with either a weir equation, or the full 2D equations. Additionally there are options to add gates, culverts, rating curves, and time series of flows for independent outlets. Any combination of these outlet types can be used. 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).

To add a hydraulic structure inside of a 2D flow area, do the following:

- First, select the Drawing tool at the top of the Geometric Data editor labeled **SA/2D Area Conn**. Then draw a line directly down the center of the hydraulic structure (**Note: this line must be drawn from left to right, while looking from what is considered to be upstream to downstream. This is how the program figures out what is considered to be the headwater side and the tailwater side.**). This line will represent the hydraulic structure that will be used to connect the 2D flow areas cells on one side of it to the other side of it. The interface will ask for a label to define the name of the hydraulic structure. This structure centerline can also be drawn from HEC-RAS Mapper.
- Next, modify the 2D flow area mesh so that the faces of the cells go along the centerline of the top of the hydraulic structure. To do this, Left Click on the Hydraulic structure centerline and select the option called **Edit Internal Connection (Break Line) Cell Spacing**. A window will appear in which it will allow the user to enter a Minimum and Maximum cell spacing to be used for creating cells along the centerline of the hydraulic structure. By default it will use the nominal cell size for spacing cells along the hydraulic structure centerline, however, the user can change the cell spacing along the structure to get more detail along the hydraulic structure. This step is option, but generally a good idea to establish cells along the structure with an appropriate size. Next, left click on the hydraulic structure centerline and select the option called **Enforce Internal Connection as Break Line in 2D Flow Area**. When this option is selected, the software will use the structure centerline and the cell spacing information to create cells along the centerline of the structure that have faces exactly along the centerline. This is a necessary step in order to get the 2D mesh correctly developed for incorporating the hydraulic structure data (Station-elevation data; culverts, gates, breaches, etc…)

  For example, as shown in Figure 3-43, a levee is being modeled inside of a single 2D flow area as a hydraulic structure. The 2D flow area mesh was modified to have cells on both sides of the levee lined up on top of the levee. This requires adding small enough cell spacing along the hydraulic structure centerline to get the correct detail. However, you do not want the cells to be so small that you have cells going down the levee embankment, such that these cells would be very steep. Steep cells on the back side of a levee could cause the model to have stability issues when flow overtops the levee (i.e. the flow may appear to be going over a water fall). So make the cells large enough to encompass the levee embankment slope and a little of the area away from the toe of the levee.

- Next, select the Storage Area/ 2D flow area Hydraulic Connection (**SA/2D Area Conn**) editor on the left panel of the **Geometric Data Editor**. This will bring up the editor shown in Figure 3-44.

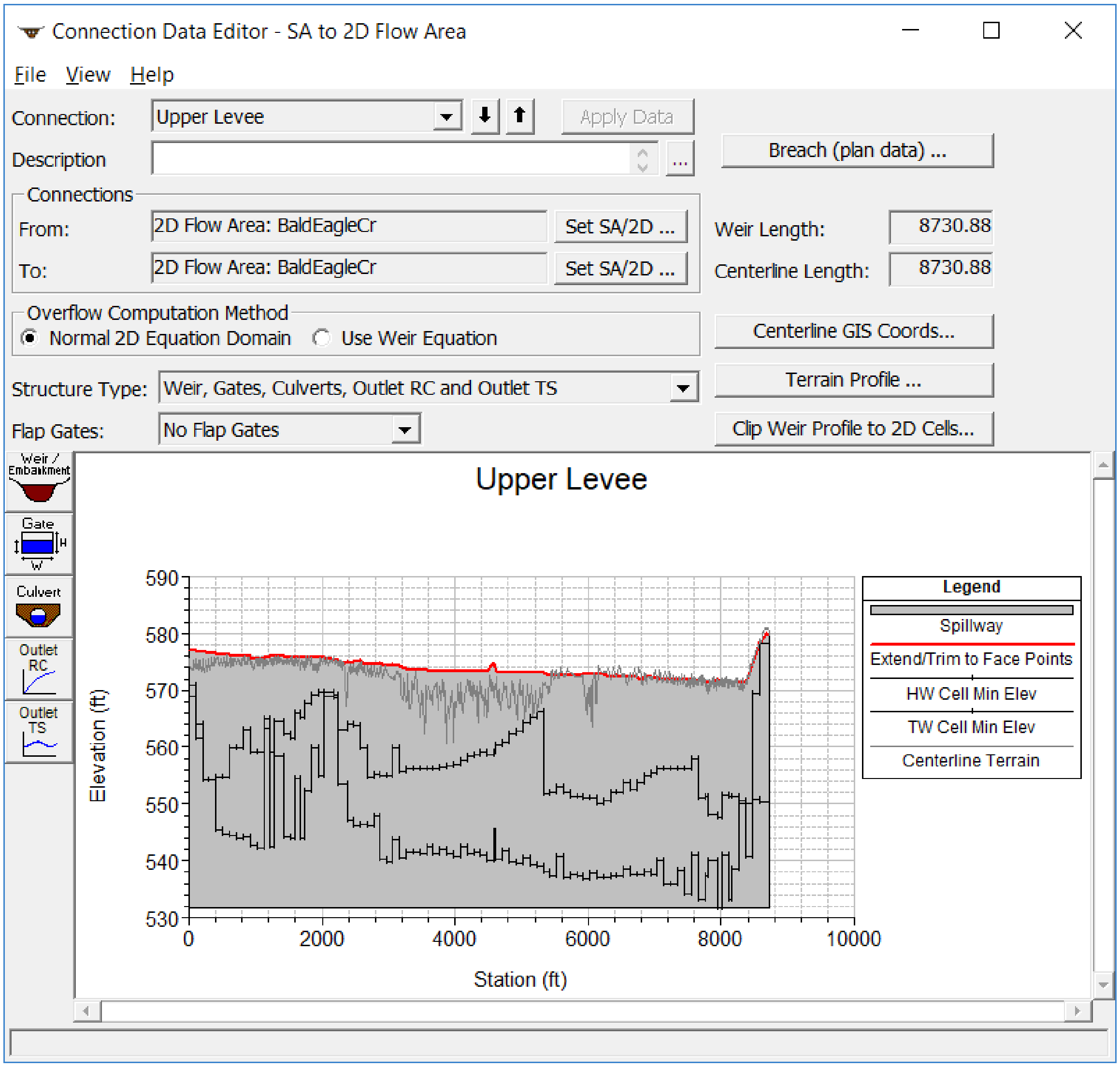


Figure 3-44. Example of using SA/2D Area Conn to put a hydraulic structure inside a 2D flow area.

The user can define station elevation data for the structure that is the same or higher than the natural ground using the **Weir/Embankment Editor**. Additionally, culverts, gated openings, rating curves, and time series outlets can be added into the hydraulic structure. The user entered weir line (station/elevation data), culverts, and gate openings, are not allowed to be lower than the minimum elevation of the cells they are connected to. However, a breach minimum elevation can go lower than the cells it is connected too (you will get a warning about this when you do it though). If a breach goes lower than the cells it is connected too, HEC-RAS will automatically lower the cell elevations (both upstream and downstream) dynamically as the breach is eroding down below the ground elevations. However, it will only lower the cells immediately next to the structure centerline on both sides.

The editor shows to black lines that represent the minimum elevations of the cells on each side of the internal hydraulic structure. These lines show the user both the elevation and stationing of how the hydraulic structure is connected to the 2D cells on each side of the structure. Users can click on any of these lines and get the cell number of the connection, as well as the stationing (stationing along the hydraulic structure) and elevation of the cell. Additionally the cell will be highlighted on the geometric data editor, so you can see where that cell lives spatially.

The user has the option for flow going over the top of the structure (Overflow Computation Method) to be computed by either the **Weir Equation** or the **Normal 2D Equation Domain**. If **Weir Equation** is chosen, all flow over the top of the hydraulic structure is computed with the weir equation. If **Normal 2D Equation Domain** is selected, the flow over the top of the structure is computed as normal 2D Flow between cells. In either case, the flow through the culverts and gates is computed separately and linked between the cells on each side of the culvert or gate. For a highly submerged structure, where the flow is not behaving like weir flow, the 2D equation will generally give better results, but the 2D equation is not as appropriate for traditional weir flow.

Warning: The "Normal 2D Equation Domain" option should NOT be used if the height of the structure is high, such that the water flowing over the structure will go into free fall (like a waterfall). The 2D equations cannot be solved in a stable solution through a waterfall. For this situation the user will need to use the "Weir Equation" option. We plan to investigate having the program automatically switch between the weir equation and the 2D equation based on the flow condition.

To enter the structure embankment data, select the **Weir/Embankment** button. The embankment editor will come up, and the user can enter station elevation data that is either the same as the ground profile, or they can enter elevations that are higher than the ground profile in order to represent a structure that is not accurately represented by the terrain data.

Geospatial Coordinates for Hydraulic Outlets Connected to 2D Flow Areas

Users can enter geospatial coordinates for any of the hydraulic outlets that are connected to a 2D flow area (Lateral Structures and SA/2D Hydraulic Connections) or for hydraulic structures inside of a 2D Flow Area. The example SA/2D Area Connection displayed in Figure 3-45 shows 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.

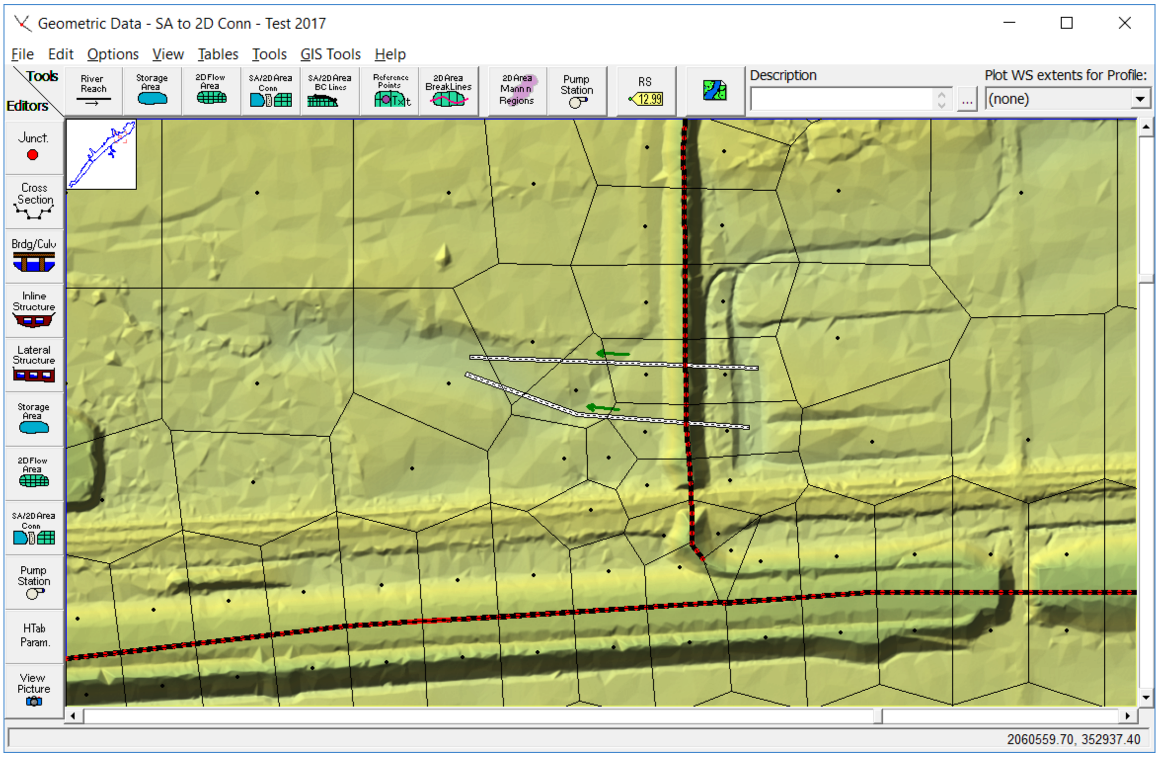


Figure 3-45. SA/2D Area Connection for Modeling a Levee (drawn left to right looking downstream) with Culverts (drawn upstream to downstream).

From the **Geometric Data** editor, click the **SA/2D Area Conn**  button (**Error! Reference source not found.**), and the updated SA/2D Area **Connection Data Editor** will open, as shown in Figure 3-46.

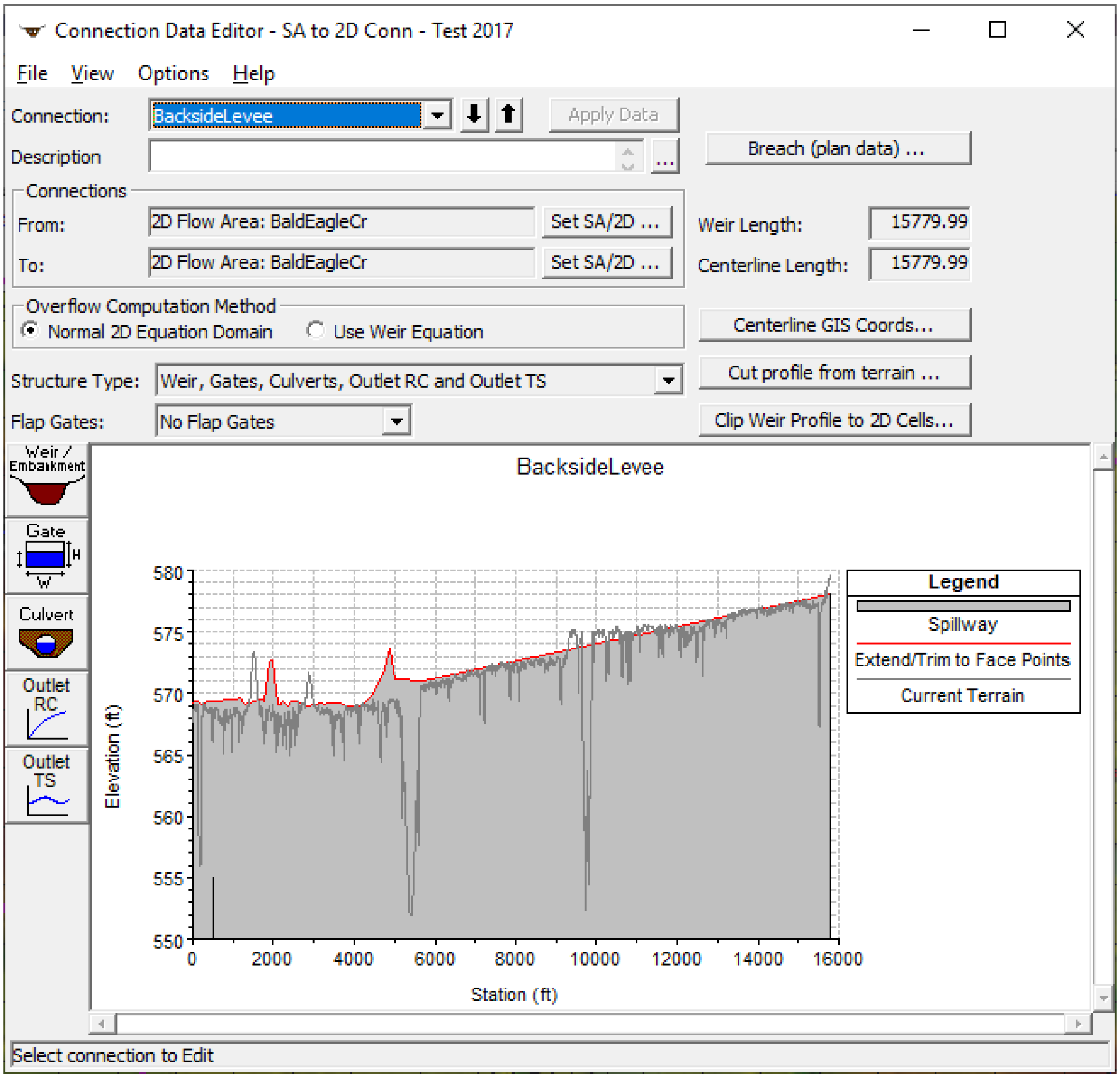


Figure 3-46. SA/2D Area Connection editor with Levee and Culverts

As shown in Figure 3-46, 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 3-45 and Figure 3-46, 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 3-47).

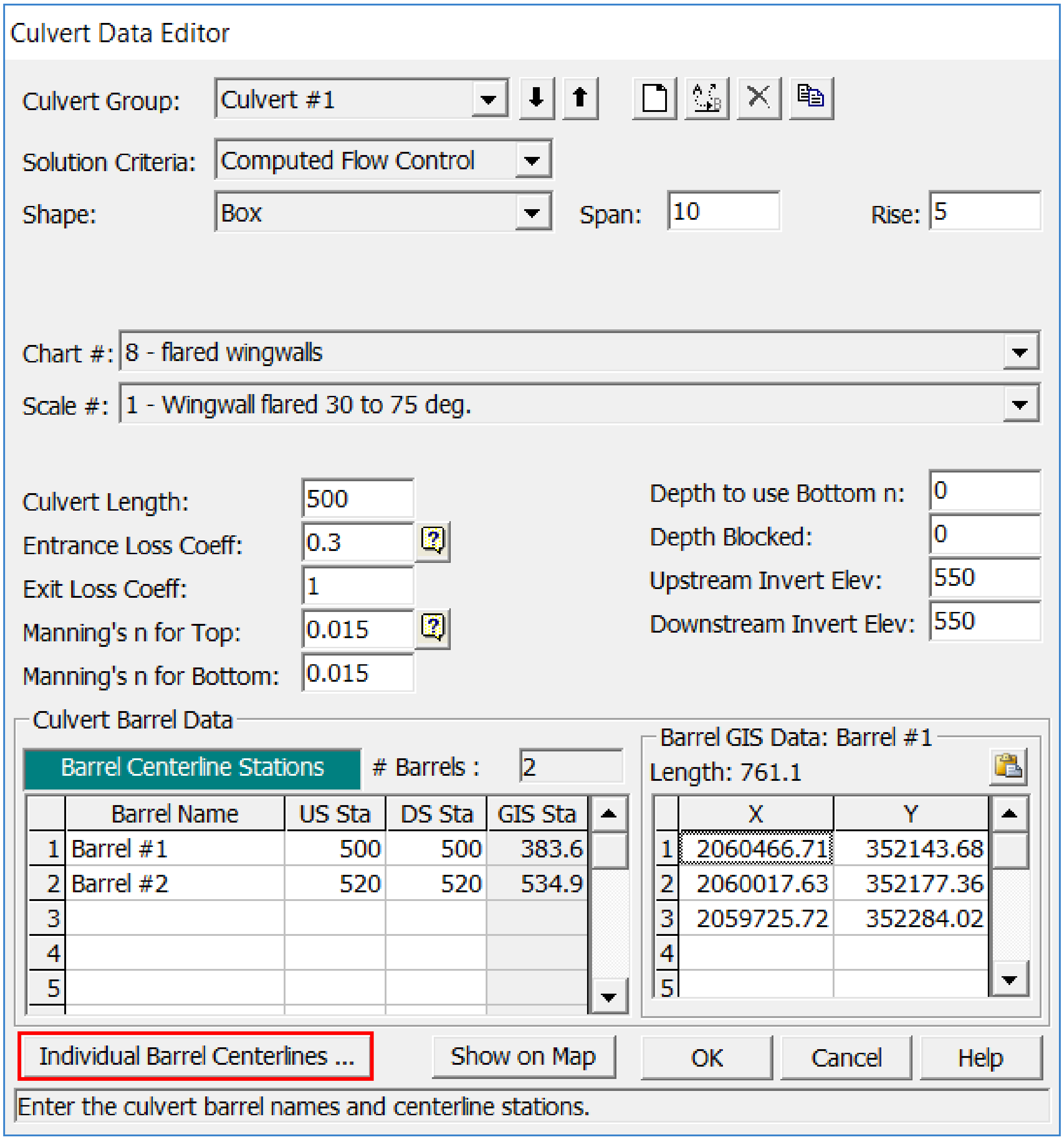


Figure 3-47. Culvert Data Editor for the SA/2D Area Connection.

To enter spatial coordinates for a culvert, press the button on the lower left portion of the editor labeled "**Individual Barrel Centerlines**" (r*ed box* in Figure 3-47). This button opens the **Edit GIS Data Table** editor (Figure 3-48), 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 3-46, 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 **Edit GIS Data Table** centerline coordinate editor opens (Figure 3-48).

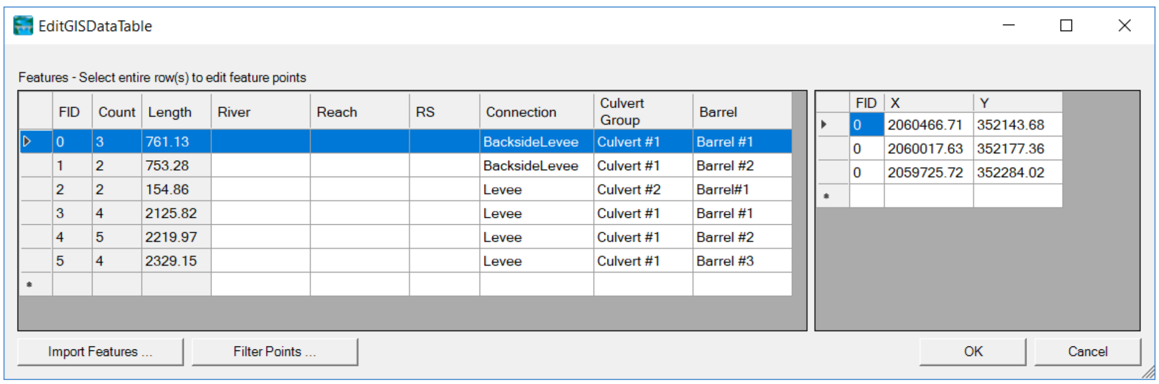


Figure 3-48. Hydraulic Structure Centerline Data Table

As shown in Figure 3-48, 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 3-45. Additionally, there is an option to import the culvert X and Y centerline coordinates at the bottom of the editor.

> **Note:** In HEC-RAS, 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.

Modeling Bridges inside 2D Flow Areas

Users can model bridges directly inside of 2D Flow Areas. Bridges inside of 2D flow areas can handle the full range of flow regimes, from low flow to pressure flow, and combined pressure flow and flow going over top of the bridge deck/roadway. The bridge data is entered very similar to the modeling of bridges in a 1D model. Additionally, users have the same low flow (energy, momentum, and Yarnell) and high flow (energy and pressure/weir) bridge modeling approaches they had for 1D bridge modeling (Except the WSPRO low flow method is not available for 2D modeling).

The software takes the user input bridge data and modeling approaches, then develops a family of rating curves for the bridge, just as is does for 1D modeling. However, for 2D modeling, the bridges curves are used to obtain a water surface difference through the bridge for each set of cells being used to model the bridge. This water surface difference is then equated to a force. That force is distributed and put into a special version of the momentum equation for each set of cells spanning the bridge centerline. So instead of calculating friction forces; pressure forces, and spatial acceleration forces, these forces are obtained from the bridge curves. Then the 2D equations are solved as they are normally solved at any cell/face in the model. This approach allows for equivalent forces to be computed for low flow, pressure flow, and combined pressure flow/weir flow, or even low flow/weir flow. The amount of force given to each cell is based on the percentage of the total flow passing through that particular set of cells. This approach allows for varying flow, water surface, and velocity at each of the cells around the centerline of the bridge opening. So the flow is still computed as two-dimensional flow through and over top of the bridge. Flow can pass at any angle through the bridge opening based on the hydraulics of the flow and the number of cells being used to represent the bridge opening. This allows for modeling of highly skewed bridges with no special options for bridge skew required by the user.

To model a bridge inside of a 2D Flow Area, the **SA/2D Area Conn** is used. The basic steps for adding a bridge to a 2D model are the following:

1. Draw a centerline for the bridge opening/embankment using the **SA/2D Area Conn** drawing tool in the Geometry editor, or by using the editing tools in RAS Mapper. **The bridge centerline must be drawn from left to right looking downstream.**
2. Develop an appropriate mesh (cell size and orientation) for the bridge, using the structure mesh controls. Some hand editing may be required depending on the bridge and what else is near the bridge (i.e. levee, another bridge, railroad tracks, road, etc…)
3. Enter the bridge data (Deck/roadway; distance from upstream bridge deck to outside cross section' piers; abutments; bridge modeling approach; Manning's n values for the 1D bridge cross sections; and hydraulic tables controls (HTAB) into the **SA/2D Area Conn** editor.
4. Pre-process the geometry in order to create the bridge curves. Review the bridge family of rating curves for hydraulic accuracy.

5. Run the model and review the results. Make any necessary changes to the data in order to improve the results.

Draw the Bridge Centerline

To add a bridge inside of a 2D flow area, go the Geometric Data editor and select the **SA/2D Area Conn** drawing tool from the tool bar across the top of the editor. Draw the centerline of the bridge from left to right looking downstream. See Figure 3-49 below. In the example shown in Figure 3-49, the bridge centerline is only laid out for the bridge opening. The user has the choice of just modeling the main bridge opening with this structure (including the bridge abutments), or the entire road embankment (left and right of the bridge opening) can also be included. If the entire roadway approaches are included in the structure, then the family of rating curves will include the flow over the left and right roadway approaches, as well as the flow through and over the bridge. If only the bridge opening is modelled with the structure, then the bridge curves that get developed are only for flow going through and over the main bridge opening. The left and right embankments can be modelled as either, normal 2D flow cells and faces (solving the general 2D equations), or separate hydraulic structures can be laid out for the left and right roadway approaches. Using separate hydraulic structures for the roadway approaches allows the user to change the elevation of the roadway, instead of using the raw terrain elevations, and it also allows for using the weir equation to compute the overflow (if desired) and breaching of the embankments. Additionally, these separate SA/2D hydraulic connections can be used to breach the roadway embankments if desired.

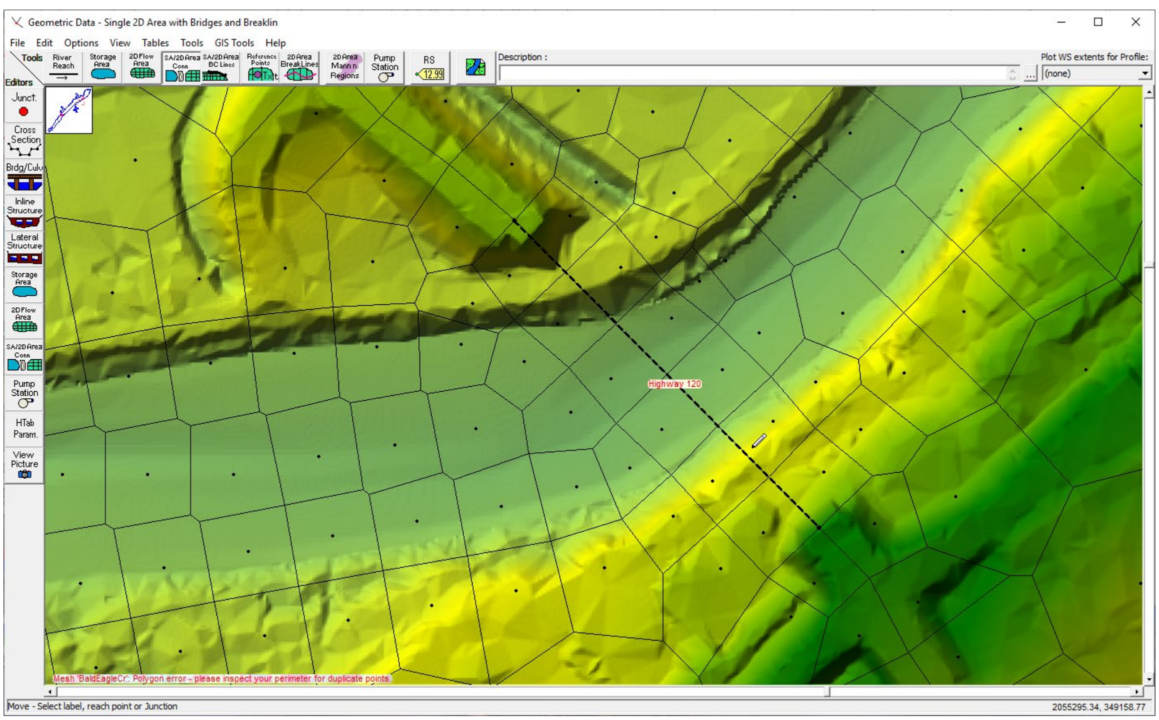


Figure 3-49. Example Bridge Centerline for Bridge Opening Only.

Entering the Bridge Data

To enter the bridge data, either left click on the bridge centerline and choose **Edit Connection**, or open the **SA/2D Area Conn** editor and select the correct structure. The first time you open the editor for a new SA/2D Area Conn it will look like what is shown in Figure 3-50 below.

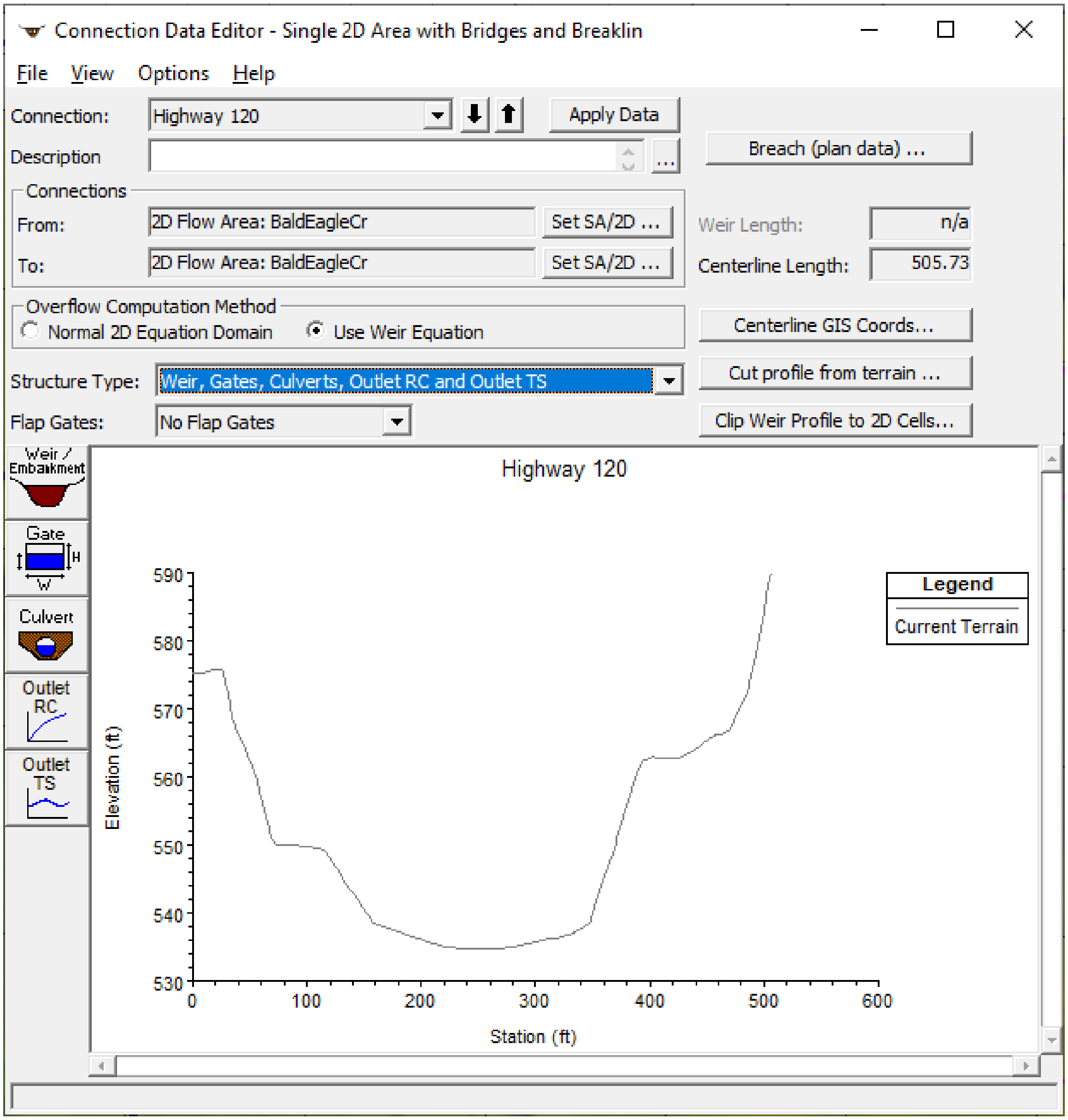


Figure 3-50. Bland SA/2D Area Conn Editor.

The first step is to change the **Structure Type** to a **Bridge**. Once this is done, the editor will now look like what is shown in Figure 3-51 below:

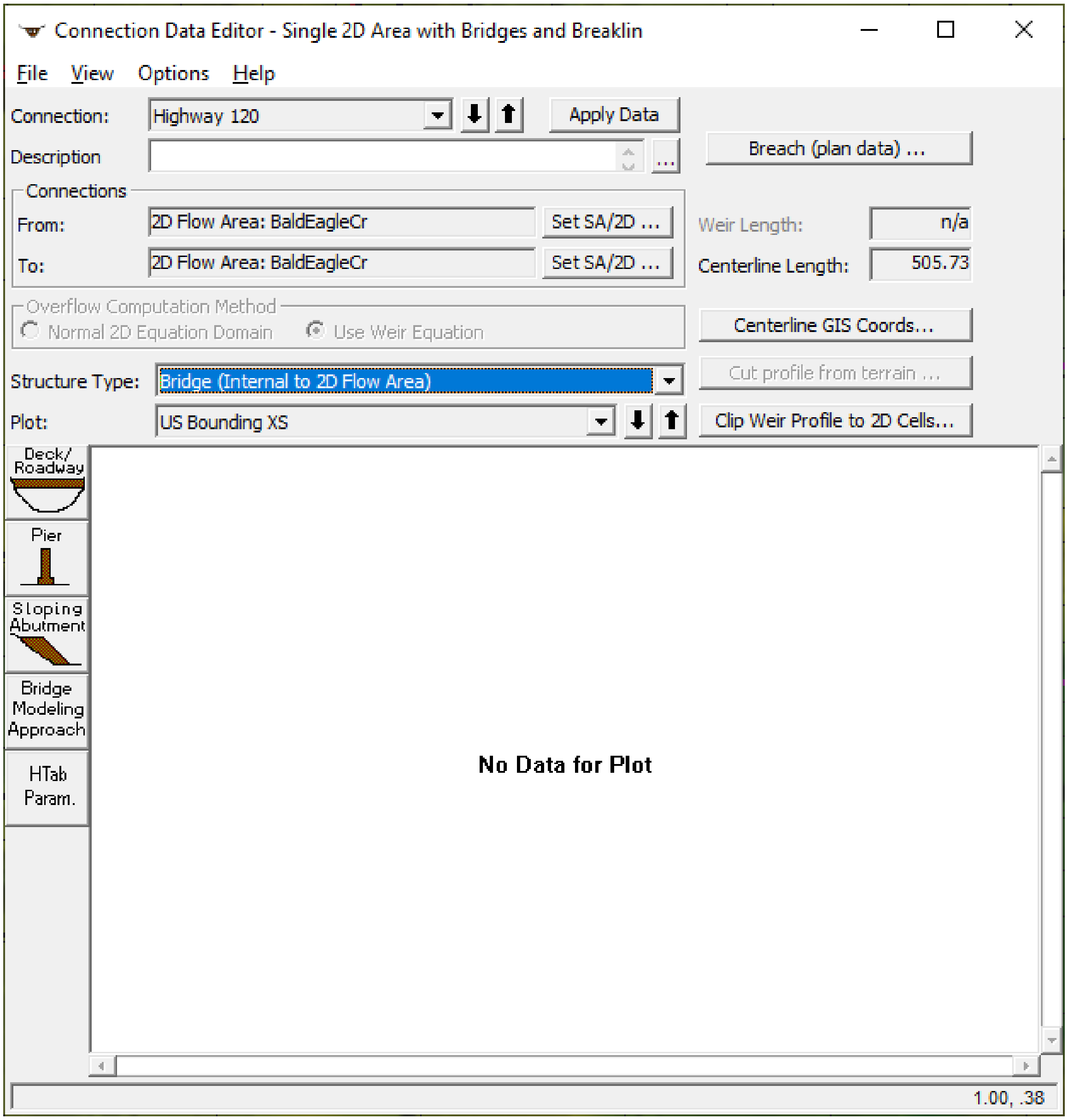


Figure 3-51. SA/2D Area Conn with Structure Type of Bridge Selected.

Once the Bridge Structure Type is selected, the editor will morph into what will look very similar to the traditional 1D bridge editor. On the left hand side of the window are data editors for entering the **Deck/Roadway; Piers; Abutments; Bridge Modeling Approach;** and the **HTAB** parameters.

The first step is to enter the Bridge **Deck/Roadway** data. When this button is selected the following window will appear (Figure 3-52):

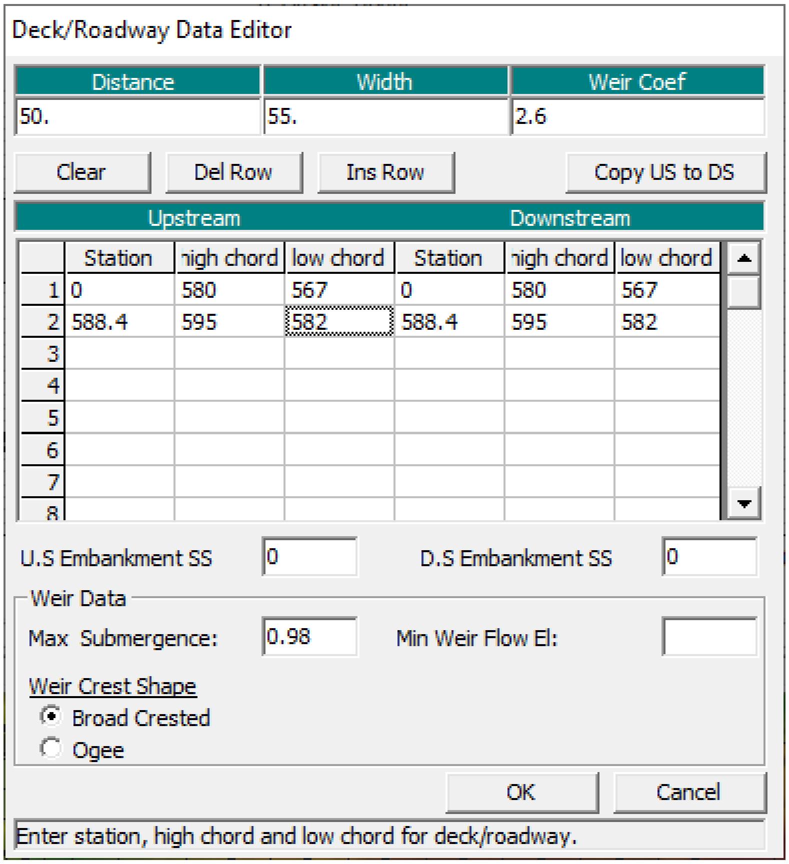


Figure 3-52. Bridge Deck/Roadway editor.

As shown in Figure 3-52, the user must enter the **Distance** (this is the distance from the upstream side of the bridge deck to the cross section upstream outside of the bridge; the **Width** of the bridge deck in the direction of flow; a **Weir Coefficient** for flow going over the road way; and the **Station** (distance from left to right along the bridge deck/roadway)**, High Chord,** and **Low Chord** elevations for the upstream and downstream side of the bridge deck.

The upstream and downstream Embankment side slopes are optional, as they are only for plots. The **Weir Crest Shape** is set to “Broad Crested” by default which is appropriate for a bridge deck/roadway. The **Max Submergence** is the submergence percentage at which the program will stop using the weir equation and transition to using the energy balance method for developing the bridge curves under high submergence.

The next step is to enter any piers or abutments that are inside of the bridge opening. When the user presses the pier editor button, the following window will appear:

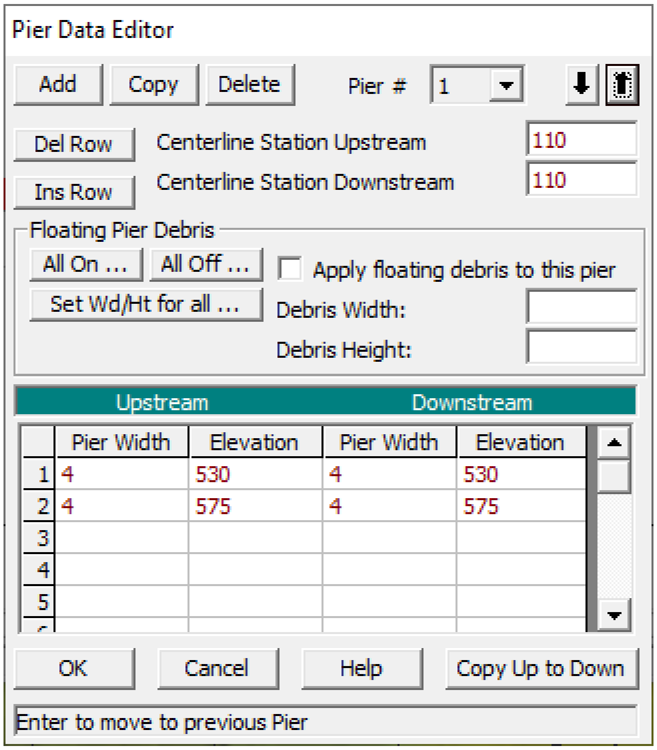


Figure 3-53. Bridge Pier Editor.

As shown in Figure 3-53, the user enters pier data in the same exact manner as they would for a 1D bridge. A **Centerline Station** is required for both the upstream and downstream side of the bridge pier. The pier is formed by entering pairs of elevations vs widths, starting below the ground and going up past the low chord of the bridge deck. This must be done for both the upstream side and downstream side of the bridge, but if they are the same, then just fill in the upstream side, then use the **Copy Up to Down** button to copy the data downstream.

The abutment editor allows you to add abutments inside of the bridge opening that are different than the natural ground. For example, “Spill through Abutments” are abutments that have a slope to then and often a rounded or angled approach to guide the flow through the opening. The abutment editor is the same as for 1D bridges, and works the same way. The user enters station and elevation data going from left to right, for each abutment in order to modify the terrain through the bridge opening.

After the bridge deck/roadway, piers, and any abutments are entered, the editor will show the bridge information graphically. See Figure 3-54 below.

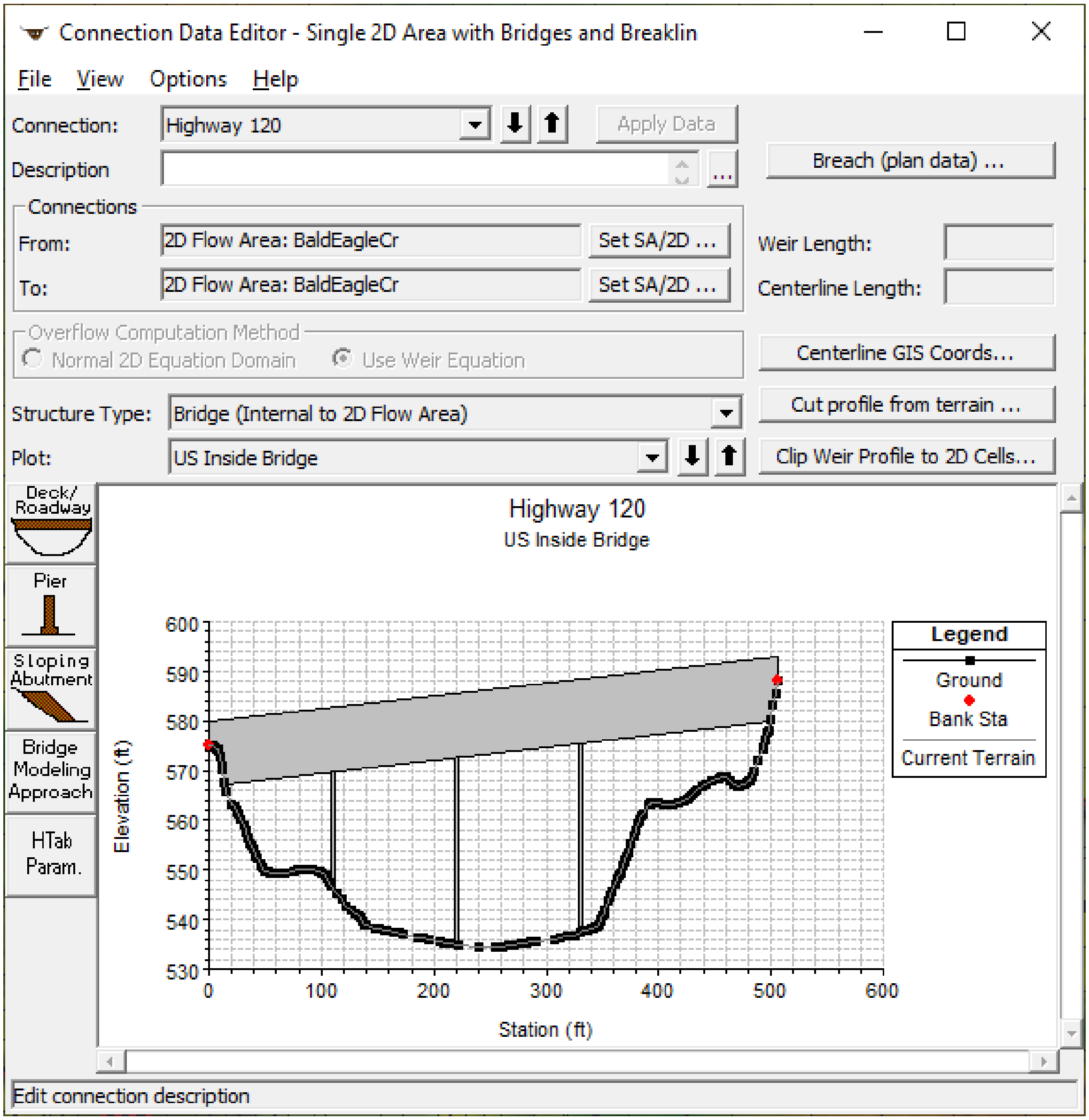


Figure 3-54. 2D Bridge Editor with Bridge Deck/Roadway and Piers.

The next step is to Enter Manning's n values for all of the 1D cross sections that are automatically formed by the user entered bridge data. To enter Manning's n values for the 1D cross sections, go to the **Options** menu and select **External and Internal Bridge Cross Sections…** When this option is selected the user will see an editor as shown in Figure 3-55 below.

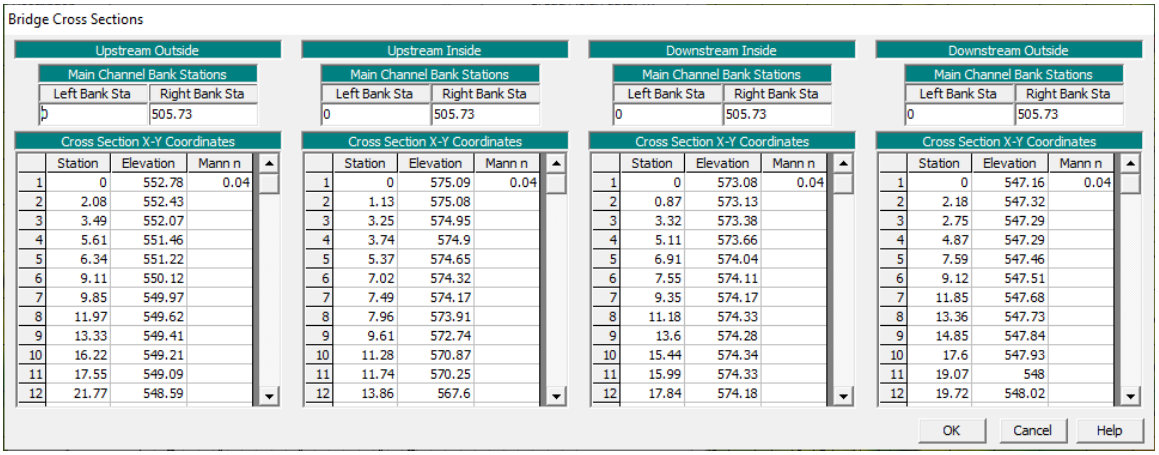


Figure 3-55. Internal Bridge Cross Sections Editor.

As shown in Figure 3-55, the user can must Enter Manning's n values for each of the four bridge cross sections (upstream Outside, Upstream Inside the bridge deck, Downstream Inside the bridge deck, and Downstream outside the bridge). Manning's n values are entered as horizontally varying values, starting with the very first station within the cross section. At least one n values must be entered for each cross section. Users can modify the cross-section station elevation data here also. However, the length of the cross section must stay the same as what is spatially laid out form the bridge centerline data and other bridge information. The left and right main channel bank stations can also be changed, by default they are set to the first and last point of each cross section.

The next step for entering the bridge data is to define the **Bridge Modeling Approach**. When selecting the Bridge Modeling Approach button, the editor will look like this:

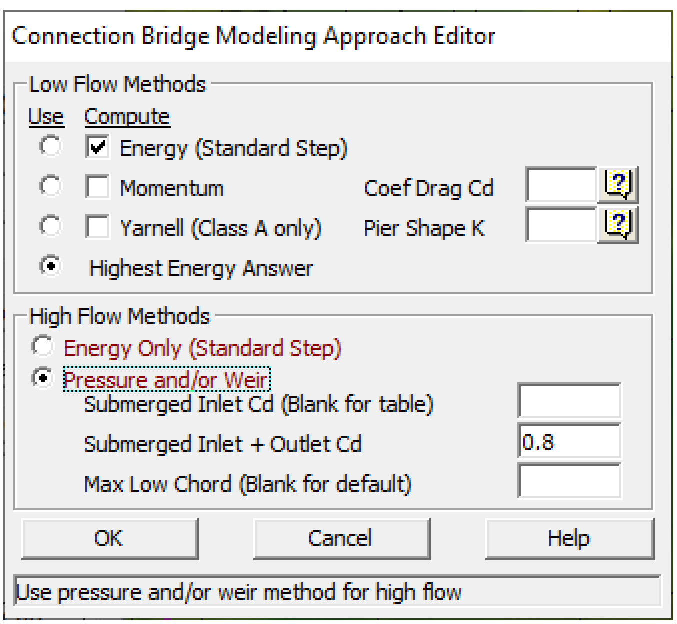


Figure 3-56. 2D Bridge Modeling Approach Editor.

This editor is very similar to the bridge modeling approach editor for 1D bridges. The user can select one or more low flow (water stays below the low chord of the bridge deck and does not pressurize the bridge opening) bridge hydraulic methods, and take the higher answer as the selected answer. Available low flow methods are: Energy, Momentum, and Yarnell. The WSPRO low flow bridge modeling method was removed as it requires approach and exit cross sections, which are not available inside of a 2D flow area.

The high flow methods are: Energy and Pressure/Weir flow. Pressure/Weir flow should be used when the bridge deck blocks a significant amount of the flow area, and the resulting upstream water surface will be significantly higher than the downstream water surface. This will cause the flow going over the roadway to pass through critical depth, much like a normal weir. If the deck is a smaller portion of the flow area, and the upstream and downstream water surface will not be significantly different, then generally the Energy based method is a better solution choice.

The last step for entering the bridge data is to define the **HTAB Parameters**. When the **HTab Param** is pressed, the following editor will appear:

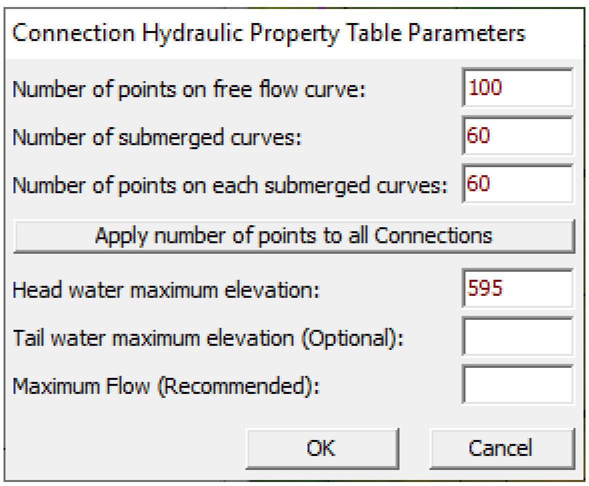


Figure 3-57. HTAB Parameters Editor for 2D Bridges.

As show in Figure 3-57, the user must enter the following information: **Number of points on free flow curve** (maximum is 100); **Number of submerged curves** (maximum is 60); **Number of points on each submerged curve** (maximum is 60); and **Head water maximum elevation.** The **Tail water maximum elevation** is optional, as is the **Maximum Flow**. However, the maximum flow is recommended as it will help control the limits of the table.

Once all of the bridge data are entered, and the user closes the SA/2D Area Conn editor, the bridge data will look like what is shown in Figure 3-58. As shown in Figure 3-58, the grey area is the bridge deck/roadway. The Black/Red line is the bridge centerline. There is also four Red dotted lines. These represent the four cross sections that will be extracted and used buy the geometric pre-processor to perform the bridge hydraulic computations in order to develop the Family of Rating curves for the bridge. The user can view all of these cross sections from the SA/2D Area Conn editor.

For the modeling of bridges inside of a 2D flow area, HEC-RAS will automatically create the four needed cross sections for pre-processing the bridge hydraulics into a family of curves. These four locations are:

1. Upstream just outside the bridge deck, normally at the toe of the upstream embankment. This cross section is automatically generated upstream of the bridge deck based on the user entered **Distance** field.
2. Upstream just inside the bridge deck/roadway.
3. Downstream just inside the bridge deck/roadway.
4. Downstream outside of the bridge deck/roadway, normally at the toe of the downstream embankment. This cross section is automatically generated downstream of the bridge deck a distance equal to what the user entered for the upstream **Distance** field.

For the two cross sections inside the bridge, they follow the edges of the deck/roadway. However, for the outside ones, it was decided to automate their location by simply creating cross sections that are parallel to the inside cross sections, and the distance upstream and downstream from the bridge deck is based on what the user entered for the **Distance** field in the Deck/Roadway editor. If the user changes the Deck/roadway data (Bridge width or distance field), then the location of the 1D cross sections will change. The user can view each of the four cross sections, as well as the centerline data (The centerline terrain data I what is applied to the 2D cell faces inside of the bridge). If the bridge data is changes, the user will see that the terrain under the current 1D cross section line is different than what they currently have entered. The user can recut any of the 1D cross sections by simply pressing the **Cut profile from terrain** button while viewing a specific cross section.

The user entered bridge deck/roadway, piers, and abutments are applied to the inside 1D cross sections. However, this data is averaged (interpolated between the two inside cross sections) in order to modify the terrain data of the 2D cell faces that are at the centerline of the bridge. The bridge centerline ground is interpolated from the two cross sections inside the bridge, at the upstream and downstream end. This is done because the bridge curves are based on those cross sections, and the terrain at the centerline was not used to develop the bridge curves. Additionally, the user can edit the internal bridge curves, say for example to better represent the terrain inside the bridge. So, interpolating the centerline from the two cross sections inside the bridge is necessary to be consistent with the cross sections used to make the bridge curves, and to maintain model stability. Make sure the Bridge centerline profile looks correct, as that is what is being used for the terrain data of the bridge centerline faces for the 2D cells.

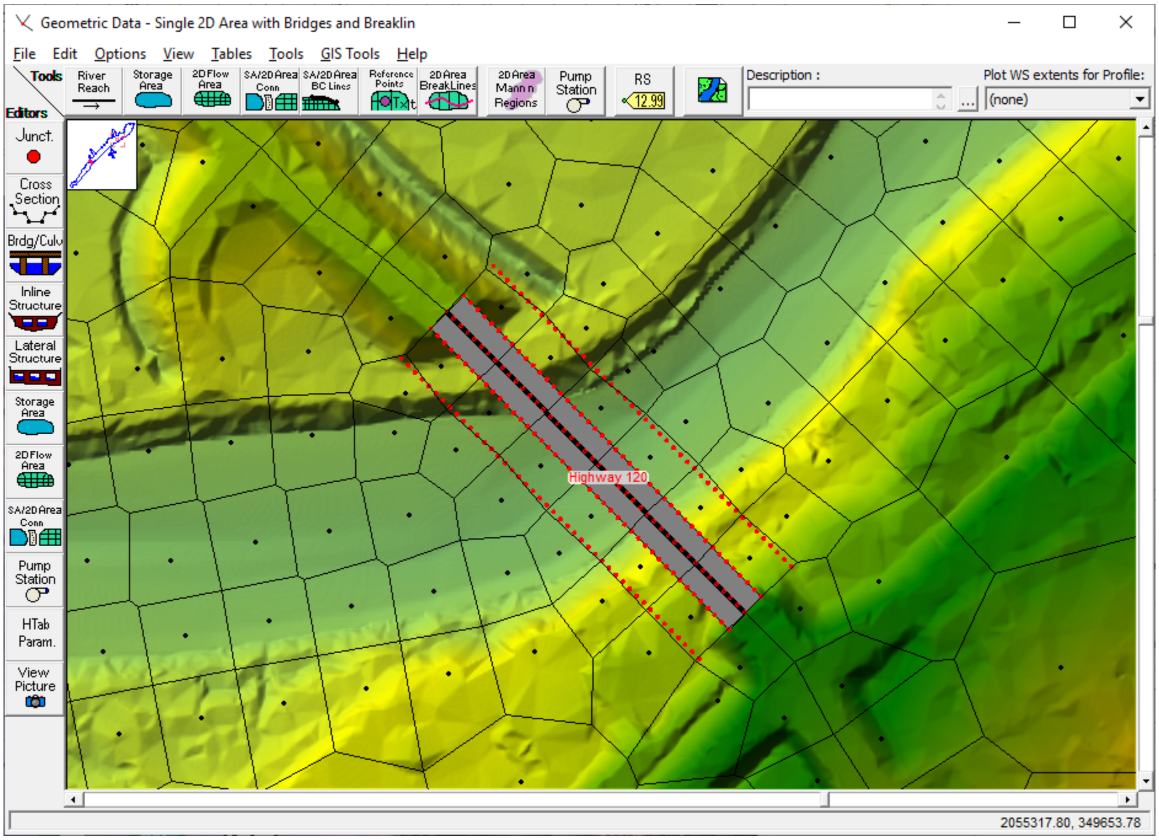


Figure 3-58. Plan view of a Bridge Inside of 2D Flow Area with Four Cross sections shown in red.

As shown in Figure 3-58, this bridge was laid out to only represent the bridge opening and abutments. The road approaches are being modelled with cell faces aligned with the high ground of the roadway. This is an option and not a requirement. The roadway approaches could have been included as part of the bridge, and the deck/roadway could have been used to refine the elevations of those roadway approaches. If the roadway approaches were included as part of the bridge model, then **Ineffective Flow Areas** would need to be entered for the two cross sections outside of the bridge. There is an option to add Ineffective Flow Areas for these cross sections under the **Options** menu of the SA/2D Area Conn editor.

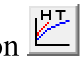
For the bridge modeling approach shown in Figure 3-58, there are six cells going across the bridge. The SA/2D Area Conn (Bridge in this case) allows the user to define a cell spacing along the centerline, then enforce that cell spacing, such that the cells start and end exactly at the beginning and end of the bridge deck/roadway (this is a requirement for all structures modelled with the SA/2D Area Conn). When forming the cells around the bridge, it is a good idea to use the mesh/cell option for “Near repeats”. This means, not only will it form the cells around the bridge centerline, but it will repeat those same exact cells outside of the initial cells. This will allow the cells around the bridge centerline to be formed as nice squares. Keep in mind, these four cross sections are only used to develop the bridge curves. But the flow through and over the bridge is still solved with the 2D flow equations.

There are several **Options** available for Bridges inside of 2D Flow Areas. From the **Options** menu at the top of the SA/2D Area Conn editor, the following options are available:

1. **External and Internal Bridge Cross Sections** – This option allows the user to edit the four cross sections that will be used for the bridge in computing the bridge curves. The user can change the station/elevation data, Manning's n values, and the main channel bank station locations. Also, if the bridge deck/roadway data is changed, and therefore the location of the four cross sections will change, the user also has the option to re-cut the cross sections from the terrain using the button labeled "**Cut Profile from Terrain**". This button will re-cut the current cross section being viewed in the editor.
2. **Bridge Ineffective Regions** - This option allows the user to define ineffective flow areas for the upstream and downstream cross sections outside of the bridge. If the user has included the left and right roadway approaches as part of the bridge, then it may be necessary to define ineffective flow areas for the outside cross sections, in order to compute accurate headwater and tailwater elevations for the bridge curves.
3. **Momentum Equation** – This option allows the user to control which components of the low flow Momentum bridge hydraulic method are turned on or off. For more details on this review the Chapter on Bridge Hydraulics in the HEC-RAS Hydraulic Reference manual.
4. **Class B Defaults** – This option allows the user to control where critical depth will be calculated/used for Class B low flow. For more details on this review the Chapter on Bridge Hydraulics in the HEC-RAS Hydraulic Reference manual.
5. **Pressure Flow Criteria** – This option is used to define when the program will start checking for the flow to transition from low flow to pressure flow. By default it uses the energy gradeline of the upstream outside cross section. This is conservative, and can often lead to the program jumping to pressure flow for bridges that have high velocities. The user can change this to the "water surface elevation" if they feel the bridge hydraulics are jumping to pressure flow too soon.

Pre-Processing the 2D Bridge into Curves

After all of the bridge data are entered, and the user has ensured that the mess/cells and the 1D cross sections are well formed around the bridge, the 2D **Geometric**

Preprocessor must be run in order for the bridge family of curves to be computed for each bridge. This will automatically happen before an unsteady flow simulation if the software detects that it needs to be run. However, the user can just run the Geometric Preprocessor without performing the full Unsteady Flow computations if they want to view the bridge curves before the computations are performed. Either way, once the geometric preprocessor is run, the user can view the Family of Curves for each of the bridges in the model. To view the bridge curves, press the Hydraulic Tables button  on the main HEC-RAS window. Then select **SA/2D Area Connections** from the **Type** menu on the plot. Shown in Figure 3-59 are the bridge curves for the bridge entered in the example above:

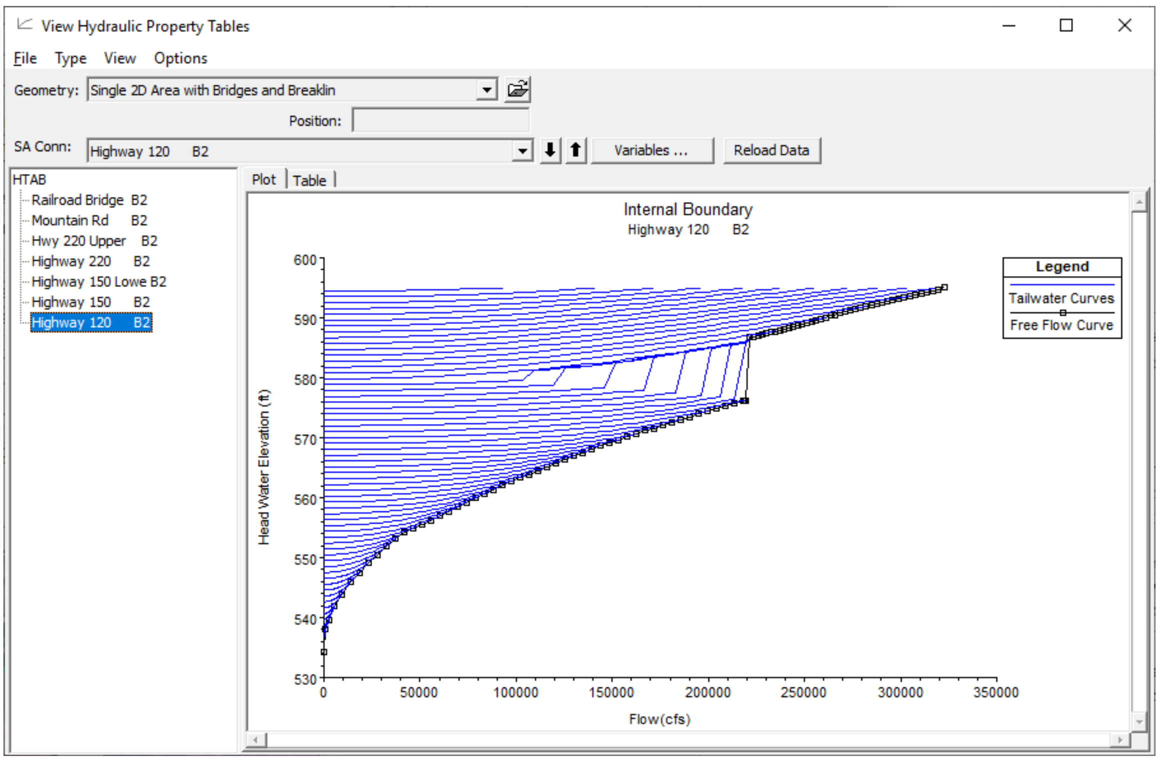


Figure 3-59. Family of Bridge Curves for Highway 120 2D Bridge Example.

Performing the Computations and Viewing the Results

Once you have entered all of the data and reviewed the preprocessed bridge curves, the model is ready to be run. There are no special options required for running the model with 2D bridges. So all that needs to be done is to run the normal Unsteady Flow simulation, just as you would for any 2D or combined 1D/2D model. Once the model has finished running the user can begin to view the output related to the 2D bridge hydraulics.

There are several ways to view output for 2D Bridges within HEC-RAS. These include:

1. Inundation maps in HEC-RAS Mapper, including water surface, velocity, etc…
2. Profile line plots inside of HEC-RAS Mapper
3. Stage and Flow Hydrograph plots from the main HEC-RAS Hydrograph plot button, or by left clicking on the structure in the Geometry editor and selecting "Stage and Flow Hydrograph".
4. Cross section plot of the SA/2D Area Connection. From the cross section plot window, select "SA/2D Area Conn" from the Type menu.
5. Tabular Output from the detailed output tables for the SA/2D Hydraulic Connection that represents the bridge.

Output from HEC-RAS mapper is available in many forms. The most common output is maps of water surface elevation and velocity. Shown in Figure 3-60 is a colored water surface elevation plot though the bridge area. This plot also has water surface contours turned on at the 0.2 ft level, which provides more context into how and where the water surface is changing.

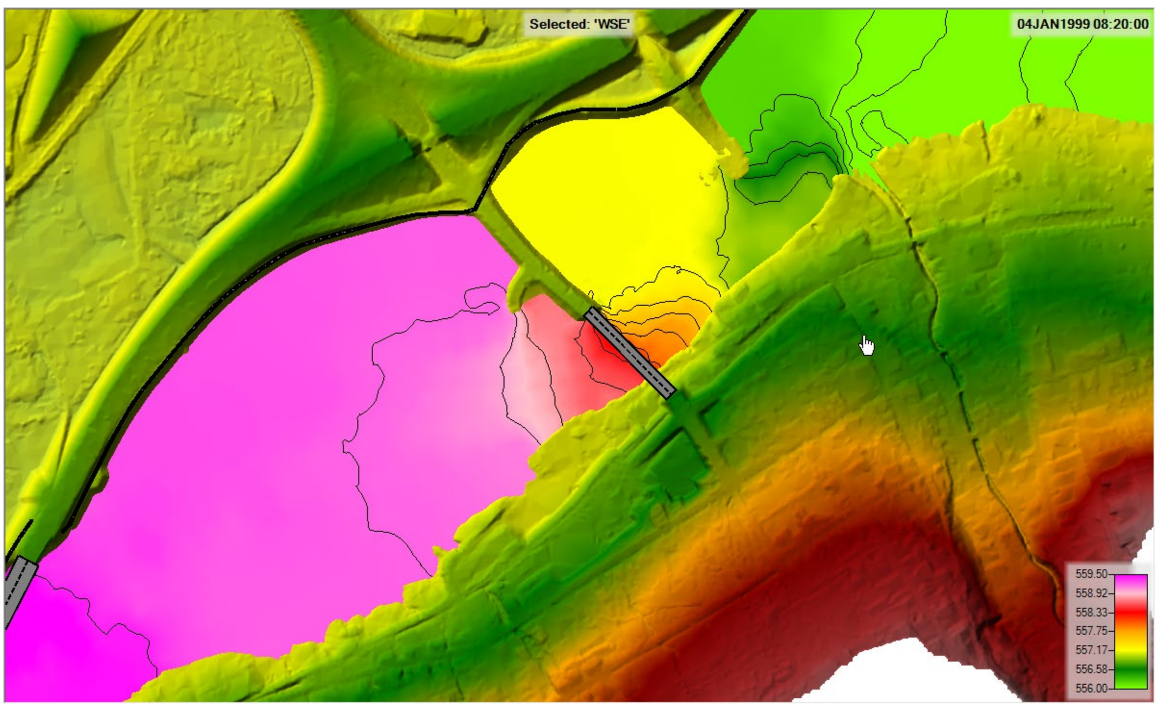


Figure 3-60. Colored Water Surface Elevation Plot with Contours, through a 2D Bridge.

Another very useful flow, especially in the area of Bridge, is the plot of colored velocity with the particle tracers turned on. Shown in Figure 3-61 is a velocity plot with particle tracers turned on.

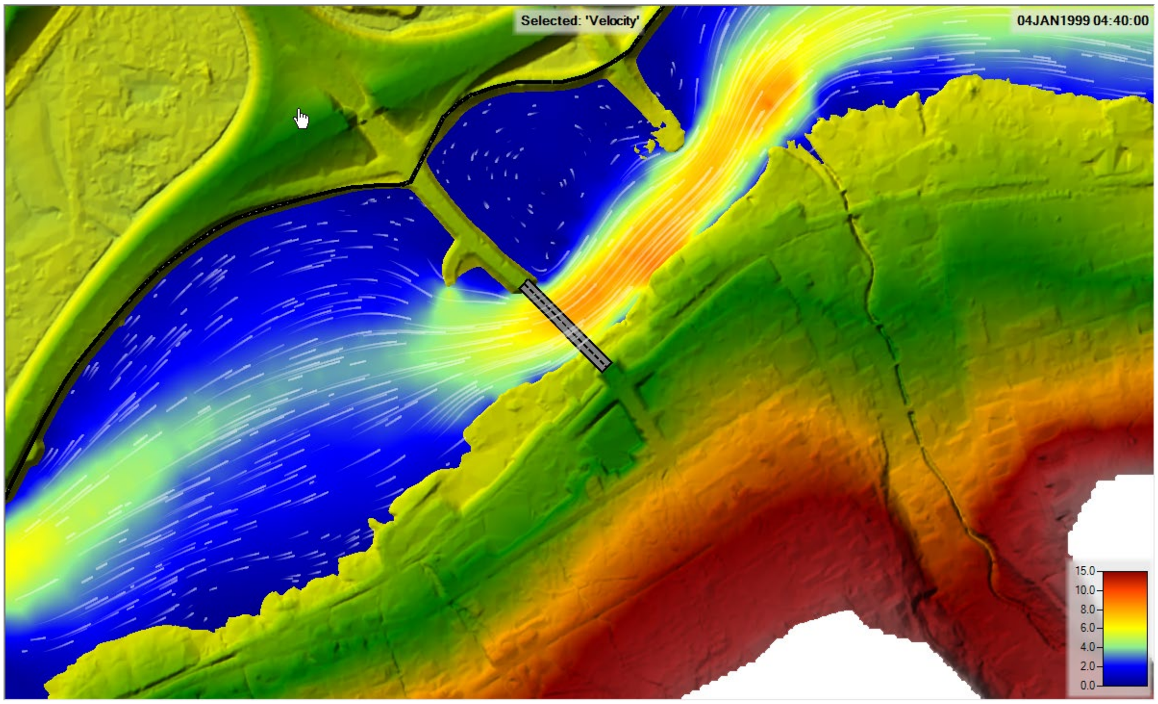


Figure 3-61. Colored Velocity Plot with Particle Tracers through a 2D Bridge.

One thing to make note of from both Figures 3-60 and 3-61. The flow is computed as true 2D flow through the bridge, even if it overtops the bridge. The 1D cross sections and bridge information are used to compute the family of rating curves, but the curves are only used to obtain the momentum force loss through the bridge opening. This approach does not force a 1D bridge water surface or velocity. As shown in the figures, the water surface varies across the channel upstream of the bridge, through the bridge, and downstream of the bridge. The velocity also varies, from cell to cell, based on the terrain, degree of constriction, and roughness coefficients.

Another useful output from HEC-RAS Mapper are Profile Line Plots. A common use of profile lines is to draw a line down the centerline of the channel and then request a plot of the terrain and the computed water surface elevation. Shown in Figure 3-62 is a terrain and water surface elevation plot through the area of the bridge.

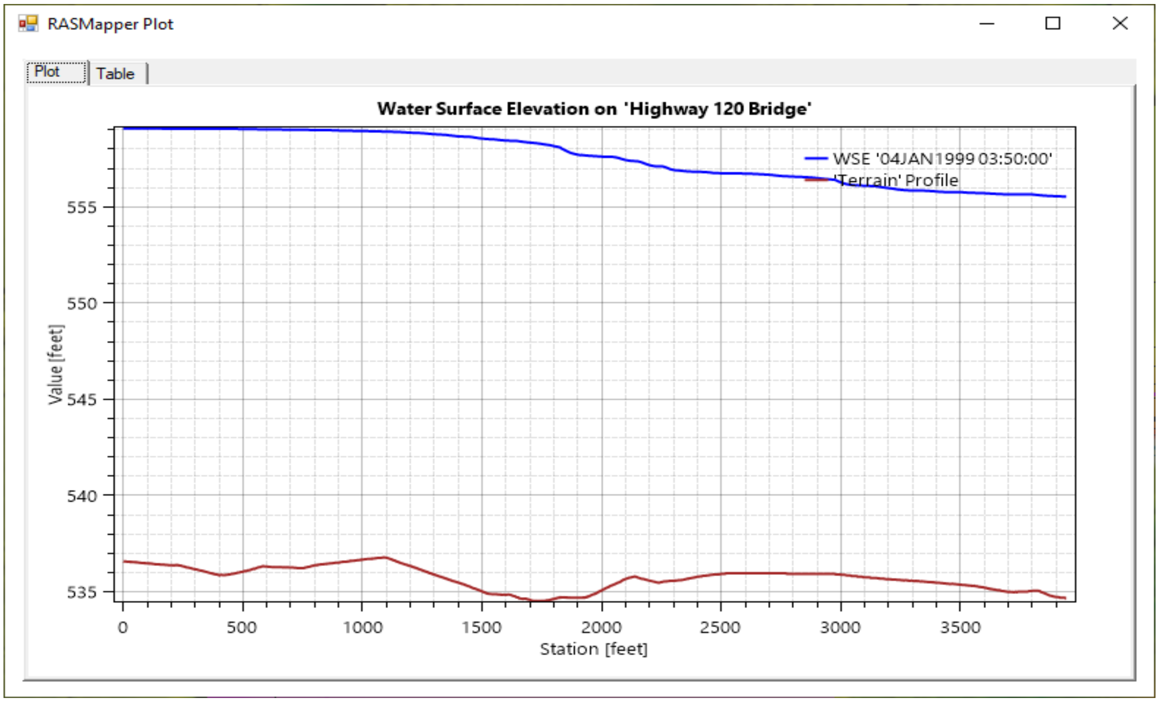


Figure 3-62. Profile line Plot of Terrain and Water Surface Elevation through the 2D Bridge Area.

Shown in Figure 3-63 is a Profile Line plot of a cross section view just upstream of the bridge.

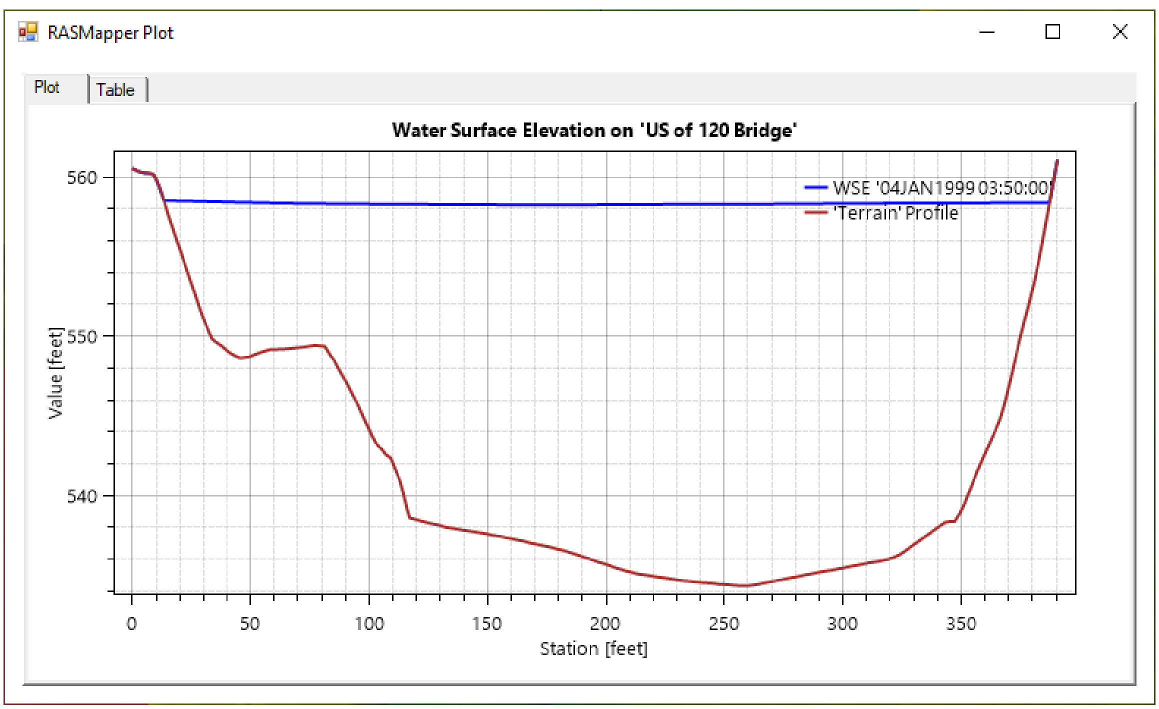


Figure 3-63. Cross Section Plot Upstream of 2D bridge using Profile lines Option.

Modeling Pump Stations inside 2D Flow Areas

A pump station can be used to pump water between two storage areas, a storage area and a river reach, between two river reaches, a river reach and a 2D Flow Area, a Storage Area and a 2D Flow Area, between two 2D Flow Areas, or from one cell to another cell within the same 2D Flow Area. Each pump station can have up to ten different pump groups, and each pump group can have up to twenty identical pumps.

To add a pump station to the system, select the **Pump Station** drawing tool at the top of the geometric data editor. When this button is pressed, move the mouse to the location that represents where the pump station will be located, and click the left mouse button. An editor will pop up asking you to enter a name for the Pump Station. This will establish a pump station location and Icon. Shown in figure 3-64 is an example of connecting a 2D flow area to a 1D cross section (TestPump) and a 2D flow area to a storage area (SAPump).

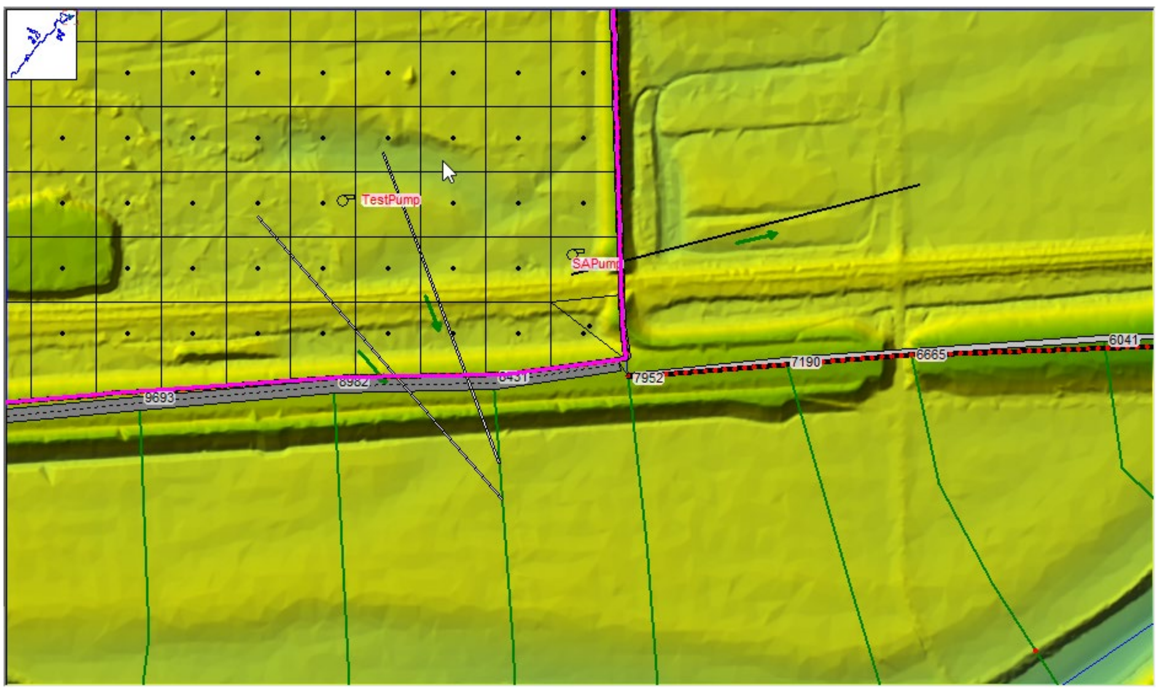


Figure 3-64. Two example pump station connections.

Once a pump station is added to the system, the user must edit the pump station and fill in the required data. To bring up the pump station editor, select the pump station editor button on the left hand side of the geometric data editor, or move the mouse over the pump station icon on the schematic, press down on the left mouse button, then select **Edit Pump Station**. When the Pump Station editor is selected, the following window will appear:

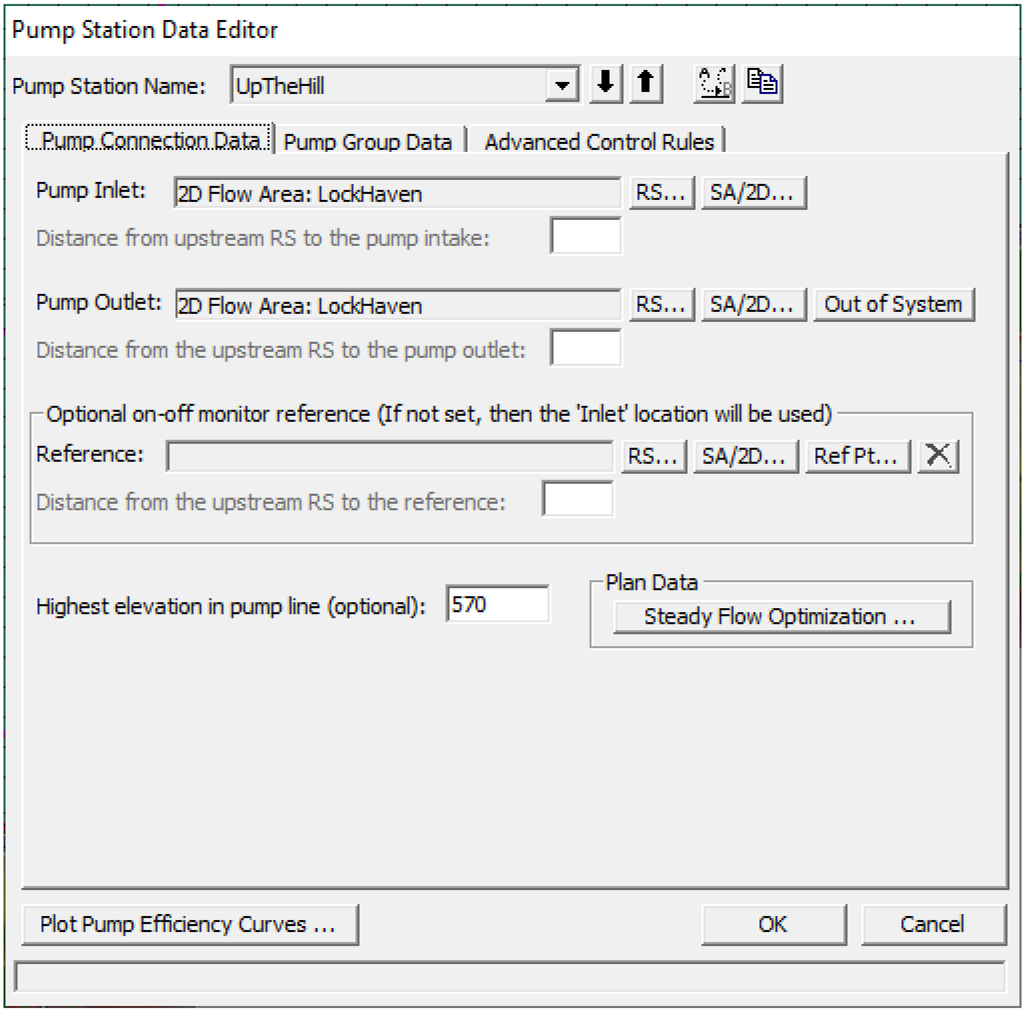


Figure 3-65. Pump Station Editor with Pump Connection Data

As shown in Figure 3-65, there are three tabs on the pump station editor, the first is for the pump connection data, the second is for the pump group data, and the third is for applying advanced rule controls over the pump station. The Pump Connection Data contains the following data:

Rename Pump Station button: This option allows the user to rename the pump station to something other than the default.

Pump Inlet: This is the location of where the pump station is pumping from. This can be either a storage area, 2D Flow Area, or a river station from a river reach. The **Set RS** button allows the user to connect a pump from a river station of a reach, the **Set SA/2D** button allows the pump to be connected from a Storage Area or a 2D Flow Area.

Pump Outlet: This is the location of where the pump station is pumping to. This can be either a storage area or 2D Flow Area (use Set SA/2D button) or a river station from a river reach (use Set RS button).

Optional On-Off Monitor Reference: By default the program uses the “Pump Inlet” location to determine when the pump should turn on or off. However, the user has the option to set a different location to be used as the monitor point for determining whether the pump should be turned on or off. This optional monitor location can be a storage area, 2D Flow Area (if you are using a 2D Flow Area, you must add a reference point into the geometry data within that 2D flow Area); or a river station within a river reach.

Highest elevation in pump line: This option allows the user to enter an elevation to be used as the highest elevation in the pump line. One example of where this may be useful is if a pump station was being used to pump water over top of a levee. In this situation, the too and from water surface elevations does not completely quantify the required head to pump the water over the levee. So it is necessary to enter the elevation of the highest point in the pump line (top of the levee) in order to accurately compute the flow going through the pump.

Steady Flow Optimization: This option is for steady flow modeling only. If water is being pumped from or to a river reach, the amount of flow going into or out of the reach should be accounted for when computing the water surface profiles. However, the water surface profiles will affect the computation of the amount of flow through the pumps. Therefore, to calculate this accurately, the pump flow and water surface profiles must be calculated iteratively until a balance is found between the river flows and the pump flows. This optimization feature is not done automatically by the steady flow program, however, the user can have the program do this by selecting **Steady flow optimization**. When this option is selected, a window will appear allowing the user to turn the pump flow optimization on.

In addition to the pump connection data the user must fill out the pump group data. Select the **Pump Group Data** tab and the editor will look like the following (Figure 3-66):

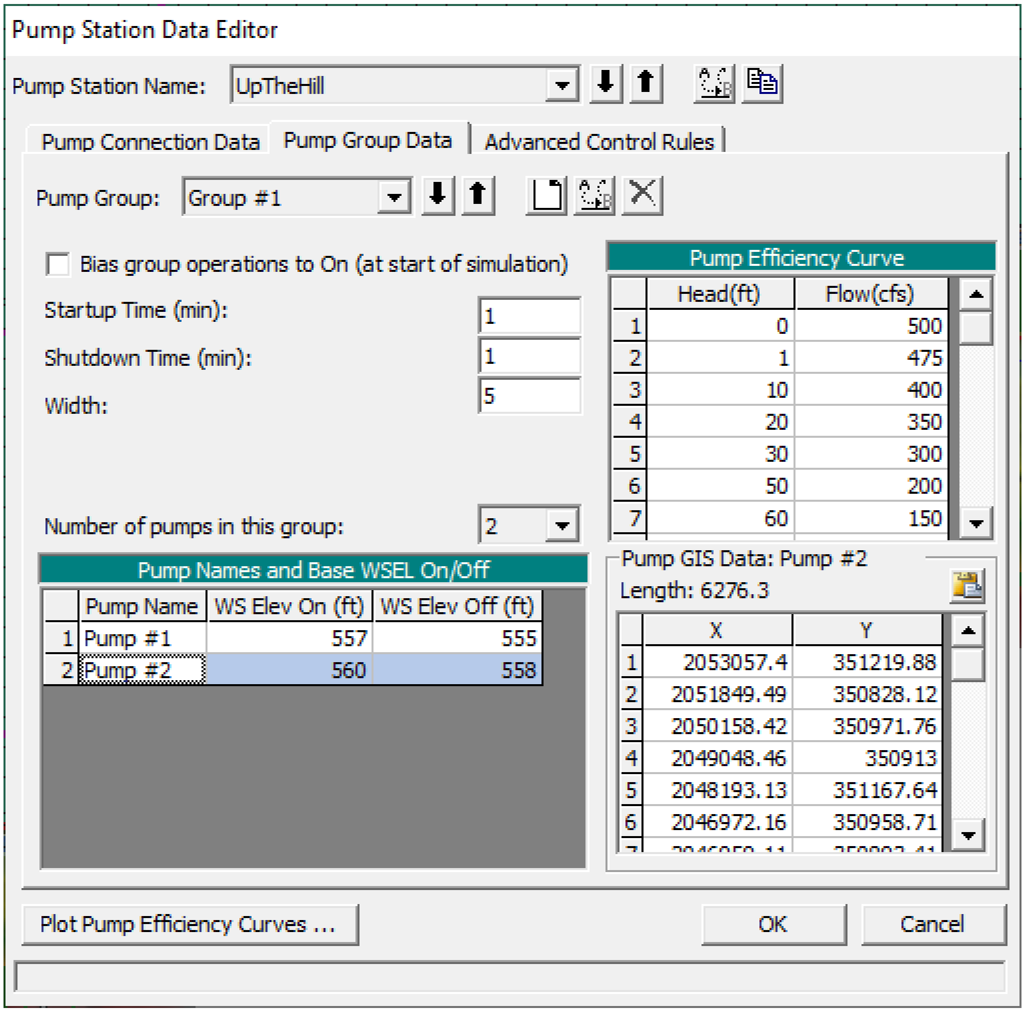


Figure 3-66. Pump Station Editor with Pump Group Data

As shown in Figure 3-66, the pump group data consists of the following:

Pump Group Name: By default, the first pump group is called "Group #1", and the second would be "Pump Group #2", etc. The user has the option to rename any pump group to whatever they would like. This is done by pressing the "Rename Group" button.

Add Group: This button is used to add another pump group. If you have pumps that have different flow capacities and use different pump efficiency curves, they must be entered as a separate pump group.

Rename Group: This allows you to rename a pump group.

Delete Group: This button is used to delete the current pump group.

Bias group operations to on (Steady Flow Only): This option is only relevant for a steady flow run. When this option is selected, and a particular water surface profile is between the on and off elevation for a pump, the program will assume the pump is turned on. If this option is not checked, then the program will assume the pump is off when the water surface is between the on and off elevations.

Startup (min): This option is used for unsteady flow only. When a pump is triggered to turn on, the default operation is that the pump turns on instantly and starts pumping to full capacity the very next time step. This option allows the user to enter a startup time in which the pumps will transition from zero flow to full capacity over the user entered time step in minutes. This option is very useful to prevent the unsteady flow computations from going unstable when to large of a flow change is experienced from a pump turning on.

Shutdown (min): This option is used for unsteady flow only. When a pump is triggered to turn off, the default operation is that the pump turns off instantly and stops pumping the very next time step. This option allows the user to enter a shut down time in which the pumps will transition from full capacity to zero flow over the user entered time step in minutes. This option is very useful to prevent the unsteady flow computations from going unstable when to large of a flow change is experienced from a pump turning off so abruptly.

Pump Width: This field is only used for drawing the width of the pump line onto the geometric data window.

Number of Pumps in Group: This field is used to enter the number of identical pumps in the current pump group. Identical pumps must use the same pump efficiency curve but can have different on and off trigger elevations.

Pump Efficiency Curve: This table is used to enter the pump efficiency curve, which is a table of static heads versus flow rates. The head represents the total head in the system, which is normally the difference in the water surface elevations between the from and the to location. **Note: The entered flow is the pump rate capacity at that particular head. In HEC-RAS, the flows entered for a given head difference, must already account for all energy losses in the pump line (friction, bends, junctions, etc…).** Do not enter the rated pump curve from the manufacturer, that curve does not account for losses in the pump line. An example of how to compute a pump efficiency curve is shown in Figure 3-67 below. As shown in Figure 3-67, the user must compute all energy losses in the system, between the two static pools. The energy losses in the line are subtracted from the manufacturer pump efficiency curve to get the curve for use in HEC-RAS. The pump efficiency curve can be plotted for visual inspection by pressing the **Plot Pump Curves** button at the bottom of the window.

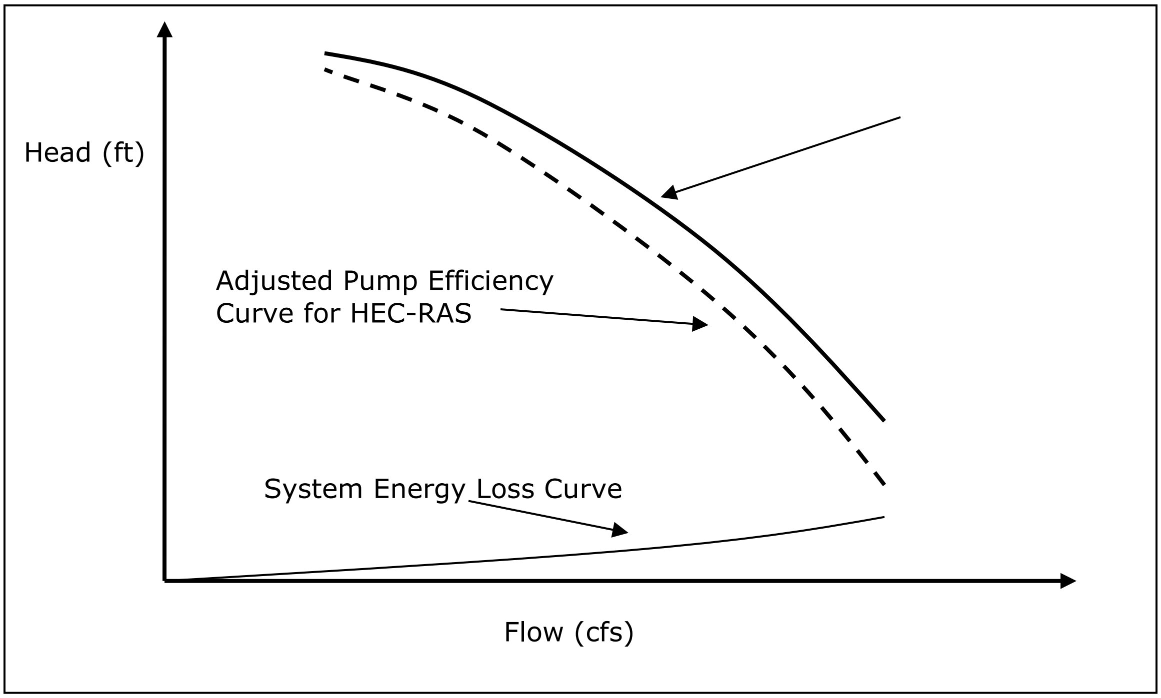


Figure 3-67. *Pump Efficiency Curve for HEC-RAS*

The monitor location for triggering a pump on or off is by default the inlet location, unless otherwise specified in the Optional On-Off Reference field. In general, the pump on elevation must be higher than the pump off elevation. Trigger elevations must be specified for all of the pumps. If the user puts the pump off elevation higher than the pump on elevation, then the pump turns on when the water surface goes below the on elevation, and the pump remains on until the water surface gets higher than the pump off elevation. This would be for example, pumping water up to a storage tank. When the pump off elevation is lower than the pump on elevation (typical way of using it), the pump turns on when it goes above the on elevation, and the pump turns off when it goes below the off elevation. This is the typical use of the pumps for interior ponding areas.

The bottom half of the window shows a table with all the individual pumps in the group. The table contains the following:

Pump Name: This field contains the name of each of the individual pumps. Pumps are automatically names “Pump #1”, then “Pump #2”, etc. The user can double click in the Pump Name field and change these names to be more specific to the location.

WS Elev On: This is the elevation at which the pump will be turned on. This is based on the elevation of the water surface at the “Inlet” location connected to the pump.

WS Elev Off: This is the elevation at which the pump will be turned off. This is based on the elevation of the water surface at the “Inlet” location connected to the pump.

Pump GIS Data: Another table directly to the right of the individual Pump Name table, is a table containing the GIS coordinates of the individual pumps. If the user clicks on a pump row in the Pump Names table (ex, if Pump #1 was selected), then the X, Y coordinates in the Pump Coordinates table are for Pump #1. If another Pump is selected, then the X, Y coordinates will be for that pump. Entering X, Y coordinates for a pump is only need when connecting pumps to 2D Flow Areas. This is required in order to figure out which cell(s) the pump is connected to in the 2D Flow Area. User must draw a spatial line that goes from the Pump Inlet to the Pump Outlet. This line will be used to show the Pump connection spatially in the Geometric Data editor, as well as establish which cell the pump is connected to. X, Y coordinates are not required if you are not connecting pumps to a 2D Flow Area. But if even one end of the pump is connected to a 2D Flow Area, the X, Y coordinates are required.

The final tab, labeled **Advanced Control Rules**, is an optional tab used to specify rules that will override the physical pump data. When this tab is selected the editor will appear as follows:

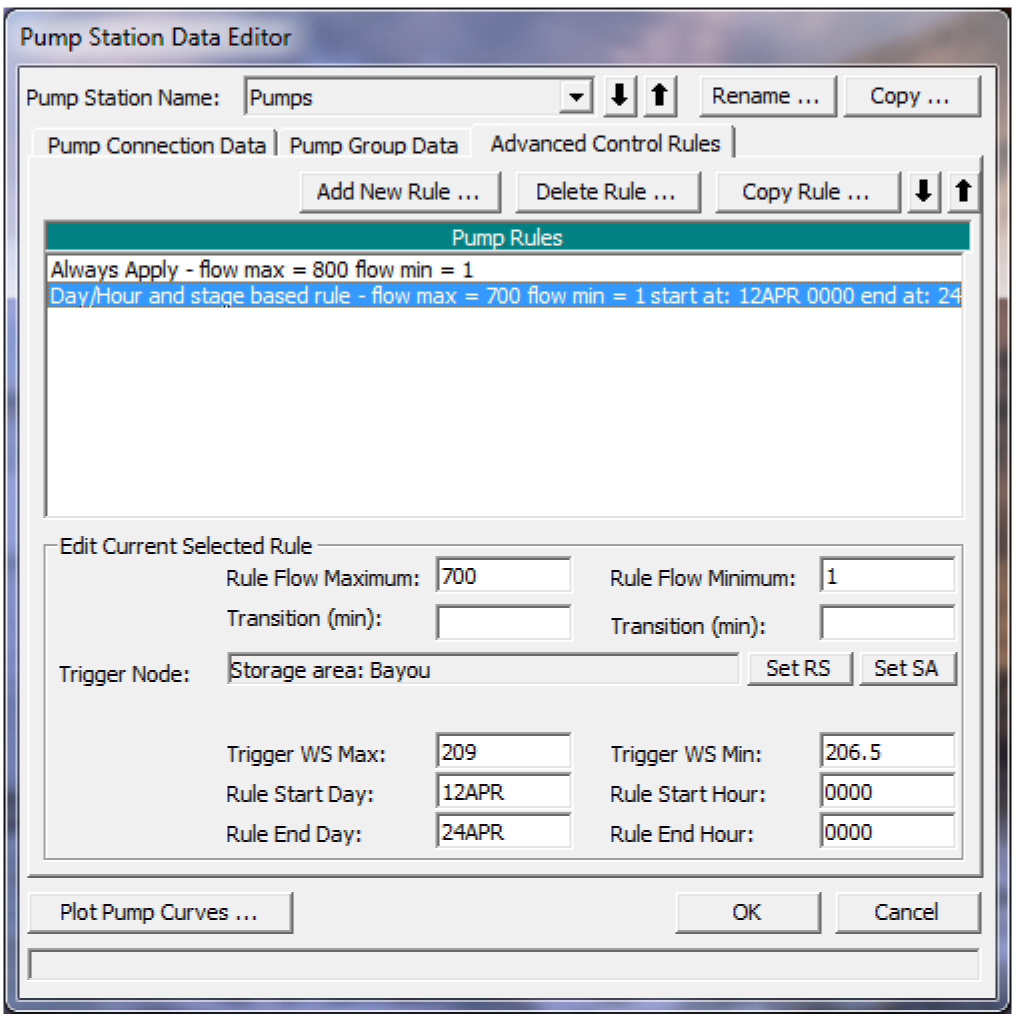


Figure 3-68. Pump Editor with Advanced Control Rules Tab Selected.

As shown in the Figure 3-68 the Advanced Control Rules tab has three buttons at the top of the editor, **Add New Rule; Delete Rule**, and **Copy Rule**. The Delete Rule button will delete the currently selected rule from the list of pump rules shown in the text box labeled **Pump Rules**. The Copy Rule button makes a copy of the currently opened rule. The Add New Rule button allows the user to enter a new rule. When this button is selected an editor will appear as shown below:

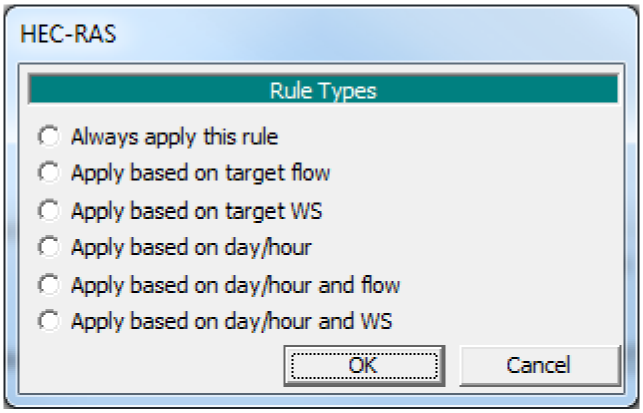


Figure 3-69. Rule Types Editor.

As shown in Figure 3-69, there are six types of rules that can be applied to a pump station. Each of the six rule types allow the user to specify a minimum and maximum flow for the entire pump station. This minimum and maximum flow will narrow the range of possible flows that have been computed for the pump station based on the physical pump data. The rule types only differ in how and when the minimum and maximum flow range gets applied.

The first rule type, **Always apply this rule,** is applied at for all time steps in the solution. The second rule type, **Apply Based on Target Flow**, is applied only when a target minimum and/or maximum flow is exceeded (flow is greater than specified maximum or less than specified minimum) at a user specified flow monitoring location. The flow monitoring location can be a cross section within a river reach, or a storage area. The third rule type, **Apply based on target WS**, is applied only when a target minimum and/or maximum water surface elevation is exceeded (stage is greater than specified maximum or less than specified minimum) at a user specified stage monitoring location. The fourth rule type, **Apply based on Day/Hour**, is only applied only during a user specified time window. The user enters a starting day and time, and an ending day and time. The specified maximum and minimum flows are then applied to the pump station only during the user specified time window. The fifth rule type, **Apply based on day/hour and flow**, is a combination of a user specified time window, and a maximum and/or minimum flow target at a user specified flow monitoring location. The last rule type, **Apply based on day/hour and WS**, is a combination of a user specified time window, and a maximum and/or minimum stage target at a user specified stage monitoring location.

The user can also apply a transition time in minutes to the maximum and minimum flow for each of the rules. Therefore, if a rule will change the flow from the currently computed value to a user entered maximum, the transition time is used to allow for the flow change to occur over a user specified time. This same concept is used for the minimum flow rate also.

The user can specify as many rules as they want for each pump station. The rules will be applied to the pump station in the order that they have been entered (which is also the order in which they appear in the editor). The user can move a rule up or down in the list by highlighting a rule, then using the up and down arrow buttons to move the rule.

After all the pump data are entered, press the **OK** button to have the data excepted by the program. This does not save the data to the hard disk; it only allows it to be used in the current execution of the program. To save the data permanently, you must save the geometry data from the File menu of the Geometric Data Editor.

CHAPTER 4

Boundary and Initial Conditions for 2D Flow Areas

Overview

HEC-RAS has a wide range of boundary and initial conditions that can be applied to a model. Boundary conditions consist of **external boundary conditions** along the perimeter of the 2D area, **internal boundary conditions**, and **global boundary conditions (**Meteorological Data**)** that are applied to the entire model (i.e. precipitation, wind, etc…).

There are four types of external boundary conditions that can be linked directly to the boundary of 2D flow areas, these are:

- **Flow Hydrograph**
- **Stage Hydrograph**
- **Normal Depth**
- **Rating Curve**

The **Normal Depth** and **Rating Curve** boundary conditions can only be used at locations where flow will leave a 2D flow area. The flow and stage hydrograph boundary conditions can be used for putting flow into or taking flow out of a 2D flow area. For a **Flow Hydrograph**, positive flow values will send flow into a 2D flow area, and negative flow values will take flow out of a 2D area. For the **Stage Hydrograph**, stages higher than the ground/water surface in a 2D flow area will send flow in, and stages lower than the water surface in the 2D flow area will send flow out. If a cell is dry and the stage boundary condition is lower than the 2D flow area cell minimum elevation, then no flow will transfer.

There are two types of internal boundary conditions that can be used in HEC-RAS:

- **Flow Hydrograph**
- **Precipitation**

The **Precipitation** boundary condition can be applied directly to any 2D flow area as a time series of rainfall excesses. This is the original way of putting precipitation into HEC-RAS. This method is very limiting, in that it applies the same precipitation spatially to the entire 2D Flow Area. A newer way of applying precipitation has been added to HEC-RAS. The new precipitation method is described next.

Global boundary conditions, are boundary conditions that get applied to the entire model. The current global boundary conditions type in HEC-RAS are:

- **Precipitation**
- **Evapotranspiration**
- **Wind**

The new methods for entering global precipitation and wind boundary conditions can be in the form of gridded data or point gage data. If point gage data are used, then the information is interpolated between the gages using various optional methods.

External Boundary Conditions

As mentioned previously, there are four types of external boundary conditions that can be applied to the outer boundary of 2D Flow Areas. These boundary condition types are: flow hydrograph; stage hydrograph, normal depth, and rating curve. External boundary condition locations can be added to the geometry either in **HEC-RAS Mapper**, or in the **Geometric Data editor**. The following explains how to do this from the Geometric Data editor. To use HEC-RAS Mapper to do this, please review the HEC-RAS Mapper manual.

To add external boundary conditions to a 2D flow area, go to the **Geometry Data editor** and select the tool (button) called **SA/2D Area BC Lines** (Figure 4-1). Once the button called **SA/2D Area BC Lines** is selected, the user can draw a line along the outer boundary of the 2D Area to establish the location of the boundary condition. To create the external boundary condition, click the left mouse button one time at the location along the outside perimeter of the 2D Area where the boundary condition should start. Next, add points by single clicking along the perimeter, then double click to end the boundary condition line at the location where it should end. Once the user double clicks to end the boundary condition line, the interface will pop up a window and ask the user to enter a name for this boundary condition. In the example shown in Figure 4-1, two 2D flow area boundary condition lines were entered at the right hand side of the 2D flow area. These boundary condition locations were given the name “DSNormalDepth1” and “DSNormalDepth2”, however, the user can use any name desired.

The user can add any number of external boundary conditions to a 2D flow area. For example, HEC-RAS allows one or more locations where a **Flow Hydrograph** boundary condition (or other types) can be connected to a single 2D flow area. The user can also have one or more **Stage Hydrographs** linked to the same 2D flow area. The user can have **Rating Curves** and **Normal Depth** boundary conditions hook up at multiple locations to allow flow to leave the 2D area.

WARNING! Two different external boundary conditions cannot be attached to the same cell face. The user must start or end at the adjacent cell face.

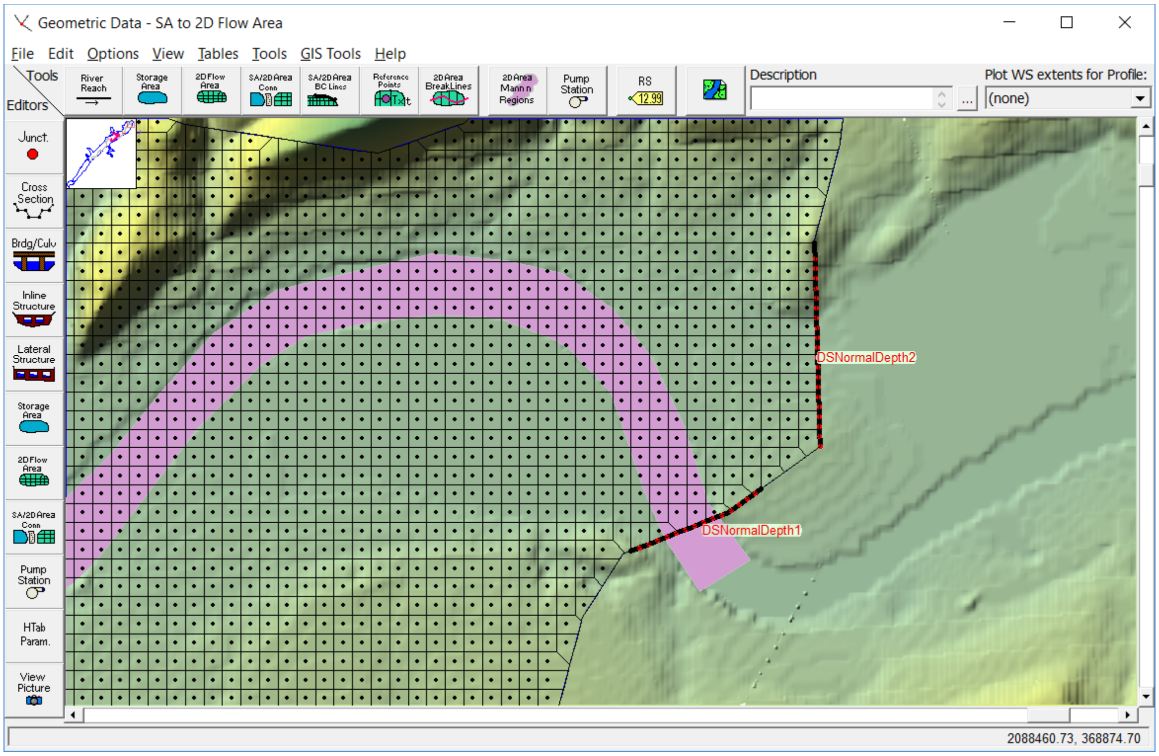


Figure 4-1. Example of adding an External 2D flow area boundary condition location.

Once all of the 2D flow area boundary conditions have been identified (drawn with the **SA/2D Area BC Lines** tool), the boundary condition type and the boundary condition data are entered within the **Unsteady Flow Data** editor. The **Unsteady Flow Data** editor is where the user selects the type of boundary condition and enters that boundary conditions data (see Figure 4-2).

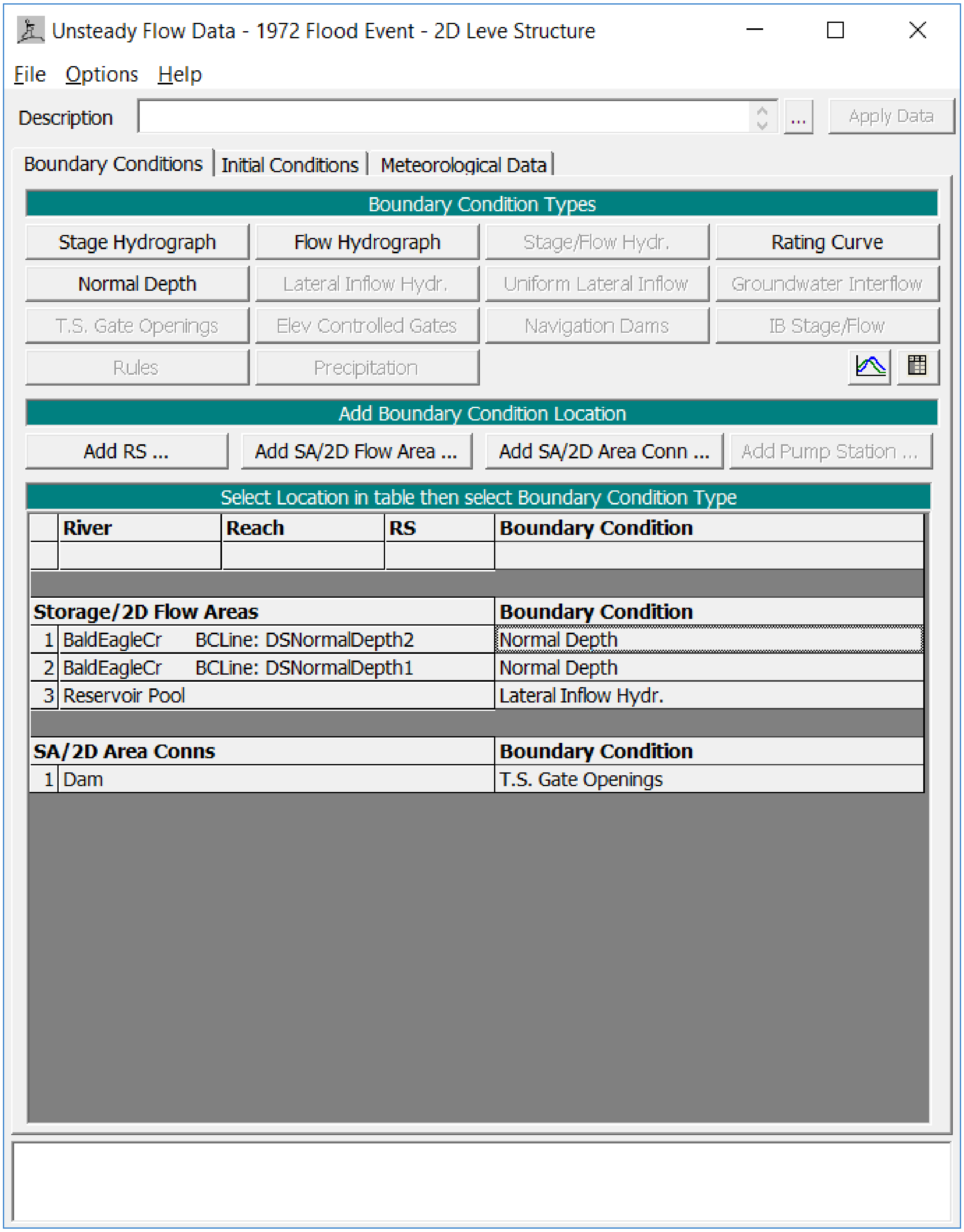


Figure 4-2. Example of adding external boundary conditions directly to a 2D flow area.

As shown in Figure 4-2, the lower table on the Boundary Conditions tab will contain any of the 2D flow area Boundary Condition locations that were entered in the Geometric

Data editor. To enter a 2D flow area boundary condition, select the appropriate field in the "Boundary Condition" column for a particular location, then select the boundary condition type from the active boundary conditions types at the top of the window. When a 2D flow area is selected, there are only four types of boundary conditions available: Stage Hydrograph; Flow Hydrograph; Rating Curve; and Normal Depth. For the example shown in Figure 4-1, two boundary condition lines were established for the 2D flow area. These two boundary conditions lines are being used to allow flow to leave the 2D flow area using the **Normal Depth** (Manning's equation) boundary condition method. Boundary condition lines can also be placed along other parts of the 2D flow area to allow flow to come in. In this case, the **Flow Hydrograph** (to bring flow directly into the 2D Area), or the **Stage Hydrograph** boundary condition type can be used.

The following shows what information is required for each boundary condition type connected directly to a 2D flow area.

Flow Hydrograph

A flow hydrograph is generally used to bring flow into a 2D flow area, however, it can also be used to take flow out (negative flow values). The required data for this boundary condition type is:

1. Flow hydrograph (Q vs time)
2. Energy Slope (for computing Normal Depth)

The Energy Slope is used to compute Normal Depth from the given flow rate and the cross section data (underlying terrain data) along the **Boundary Condition Line** for each computational time step. A flow distribution in the cross section is then computed (based on the normal depth water surface and the conveyance in the cross section) and this flow distribution is used to appropriately distribute the flow to the cells along the boundary condition line that are wet. At any given time step, only a portion of the boundary condition line may be wet, thus only the cells in which the water surface elevation is higher than their outer boundary face terrain will receive water. However, if the computed Normal Depth water surface is higher than all the boundary face elevation data along the **Boundary Condition Line**, then all the cells will receive water based on a conveyance weighting approach.

Stage Hydrograph

A Stage Hydrograph can be used to bring flow into or take flow out of a 2D flow area. If the water surface elevation in the Stage hydrograph is higher than the cell water surface elevation (or dry elevation), flow will go into the 2D Cells. When the water surface elevation of the Stage Hydrograph is lower than the water surface in the 2D flow area, flow will go out of the 2D area. If a cell is dry and the stage boundary condition is lower than the 2D flow area cell minimum elevation, then no flow will transfer. The flow is computed on a per cell basis. For instance, if the Stage Hydrograph water surface is higher than the water surface of some of the 2D boundary cells and lower than that of other 2D boundary cells, water will simultaneously enter and exit the Stage Hydrograph boundary. The Stage Hydrograph boundary condition also has an option called "**Use Initial Stage**". When this option is turned on, the first stage in the hydrograph will be used to fill the 2D area as an initial condition. The filling starts at the boundary condition and then fills any cell to that water surface as long as it is hydraulically connected. The filling stops when it runs into faces that are higher than that elevation.

Normal Depth

The **Normal Depth** boundary condition can only be used to take flow out of a 2D flow area. When using the **Normal Depth** boundary condition, the user is required to enter a friction slope for that area, just like they would do for a 1D cross section location. The friction slope should be based on the land slope in the vicinity of the 2D flow area boundary condition line. The Friction Slope is used in Manning's equation to compute a **Normal Depth** water surface elevation for each given flow, based on the cross section underneath the 2D Boundary Condition line. Just like the **Stage Hydrograph** boundary, the **Normal Depth** boundary is computed on a per cell basis.

Rating Curve

The **Rating Curve** option can only be used to take flow out of a 2D flow area. The user is required to enter a Stage (Water Surface Elevation) versus flow relationship for this option. The rating curve is also applied on a per cell basis.

Internal Boundary Conditions

Internal Flow Hydrograph

The user has the option to add a flow hydrograph as an internal boundary condition. To put in the Internal Boundary condition line, from the **Geometric Data** editor, select the the **SA/2D Area BC Lines** 

tool, then draw an internal line inside of the 2D Flow Area (Figure 4-3). 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 4-3 provides an example of a created internal BC line named "*Internal BC*", as well as an example of an external BC line (labeled "*edge bC*").

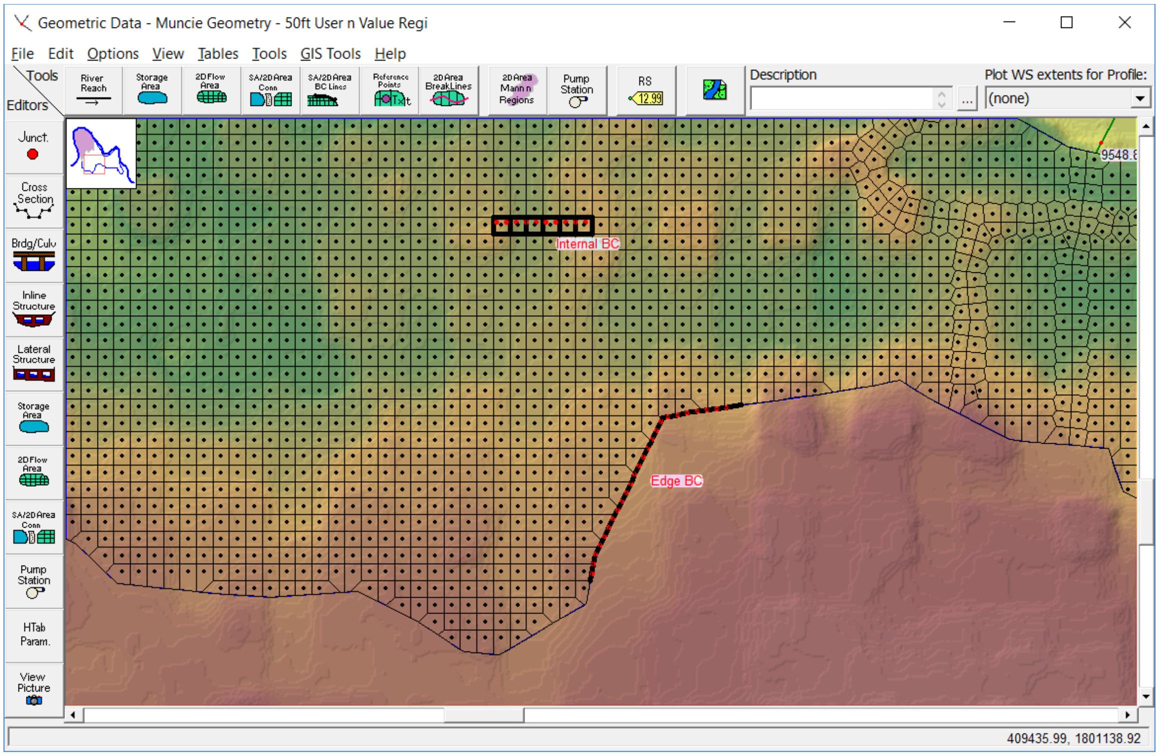


Figure 4-3. Example Internal BC Line for Attaching a Flow Hydrograph inside a 2D Area.

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 4-4) that were laid out in the **Geometric Data** editor.

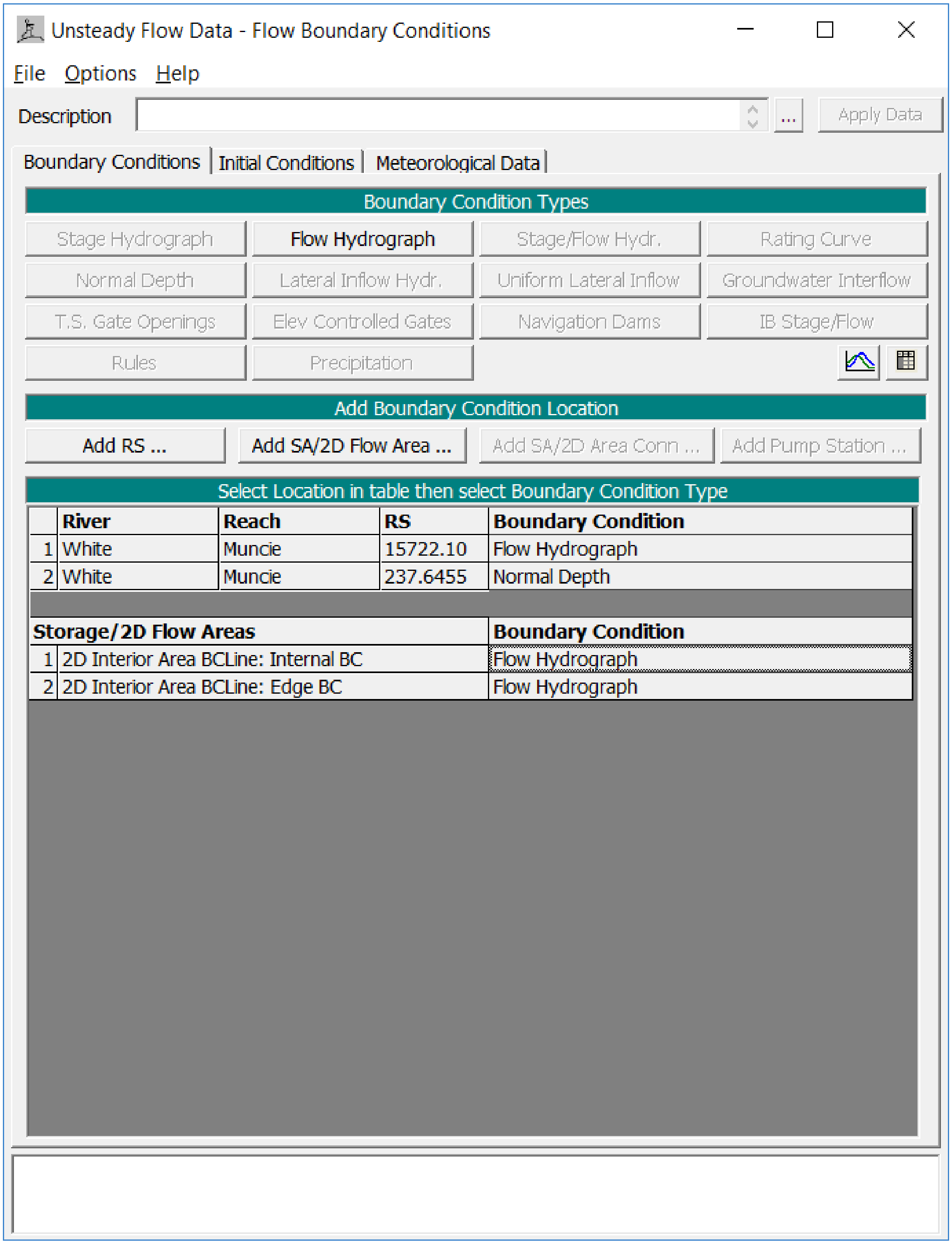


Figure 4-4. Unsteady Flow Data Editor with Example Internal BC Line.

As shown in Figure 4-4, 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.

Precipitation

This **Precipitation** option can be used to apply rainfall excess (Rainfall minus losses due to interception/infiltration) directly to a 2D flow area. To apply a precipitation boundary condition to a 2D flow area (or storage area), go to the **Unsteady Flow Data** editor and select the button that is labeled **Add Storage Area** from the Boundary Conditions Locations table. Select the 2D flow area of interest to add to the table. Once the 2D flow area is in the table, select the blank field under the Boundary Condition column, then select the **Precipitation** boundary Condition Type. This will bring up an editor that will allow you to either read the precipitation data from HEC-DSS, or enter the data as a time series into a table directly. Precipitation is applied equally to all cells within the 2D flow area.

NOTE: This is the older method for adding precipitation data into HEC-RAS. Newer capabilities have been added that allow for spatially distributed rainfall and infiltration. See the section below on "Global Boundary Conditions"

Global Boundary Conditions

HEC-RAS now has the capability to have spatially varying precipitation, evapotranspiration, and wind. Spatial precipitation, evapotranspiration, and wind data are added into the Unsteady Flow Boundary Conditions editor from the **Meteorological Data** Tab.

Spatial Precipitation and Evapotranspiration

Spatial precipitation and evapotranspiration can be entered into HEC-RAS as either gridded data, point gage data, or a constant rate. The spatial precipitation and evapotranspiration data are added into HEC-RAS through the Unsteady Flow Boundary Conditions editor. When that editor is open, you will find a tab on the main window called "**Meteorological Data**". When that tab is selected the editor will change as shown in figure 4-5. A shown in Figure 4-5, the user can enter precipitation data in either gridded or point gage formats.

Gridded Precipitation

To enter gridded precipitation data, first the user must select "**Enable"** from the "**Precipitation/Evapotranspiration:**" selection box at the upper left of the tab. Next the user must then select **"Gridded"** from the **Mode** selection box under the **Meteorological Variables** area of the window. Once "Gridded" data is selected, then the user should press the **Edit** button (button with three dots) on right hand side of the precipitation area (Figure 4-5).

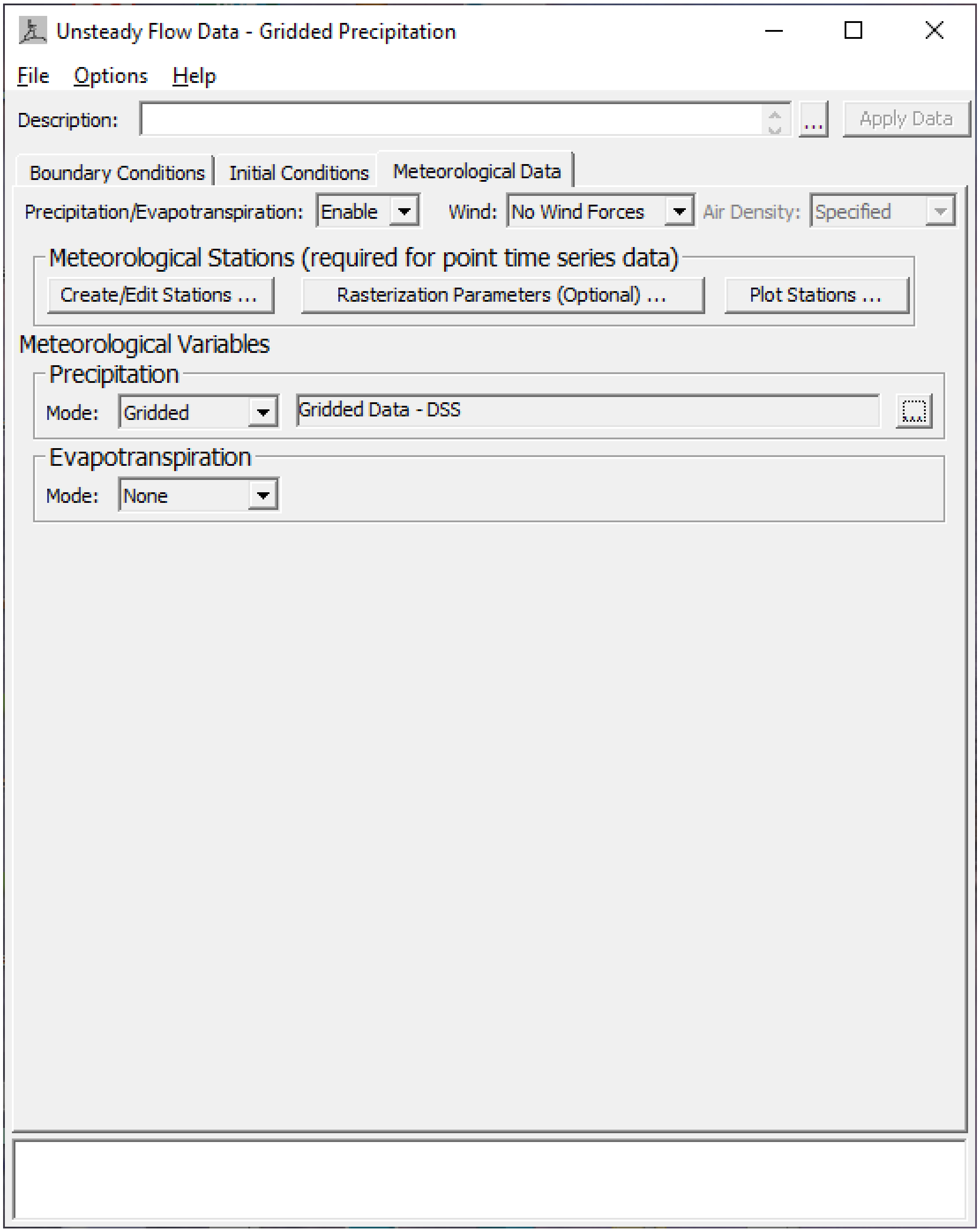


Figure 4-5. Example Unsteady Flow Boundary Condition editor with Meteorological Data tab open.

When the **Edit** button is selected, the Unsteady Flow Data will change by adding additional information under the Meteorological Variables area (Figure 4-6).

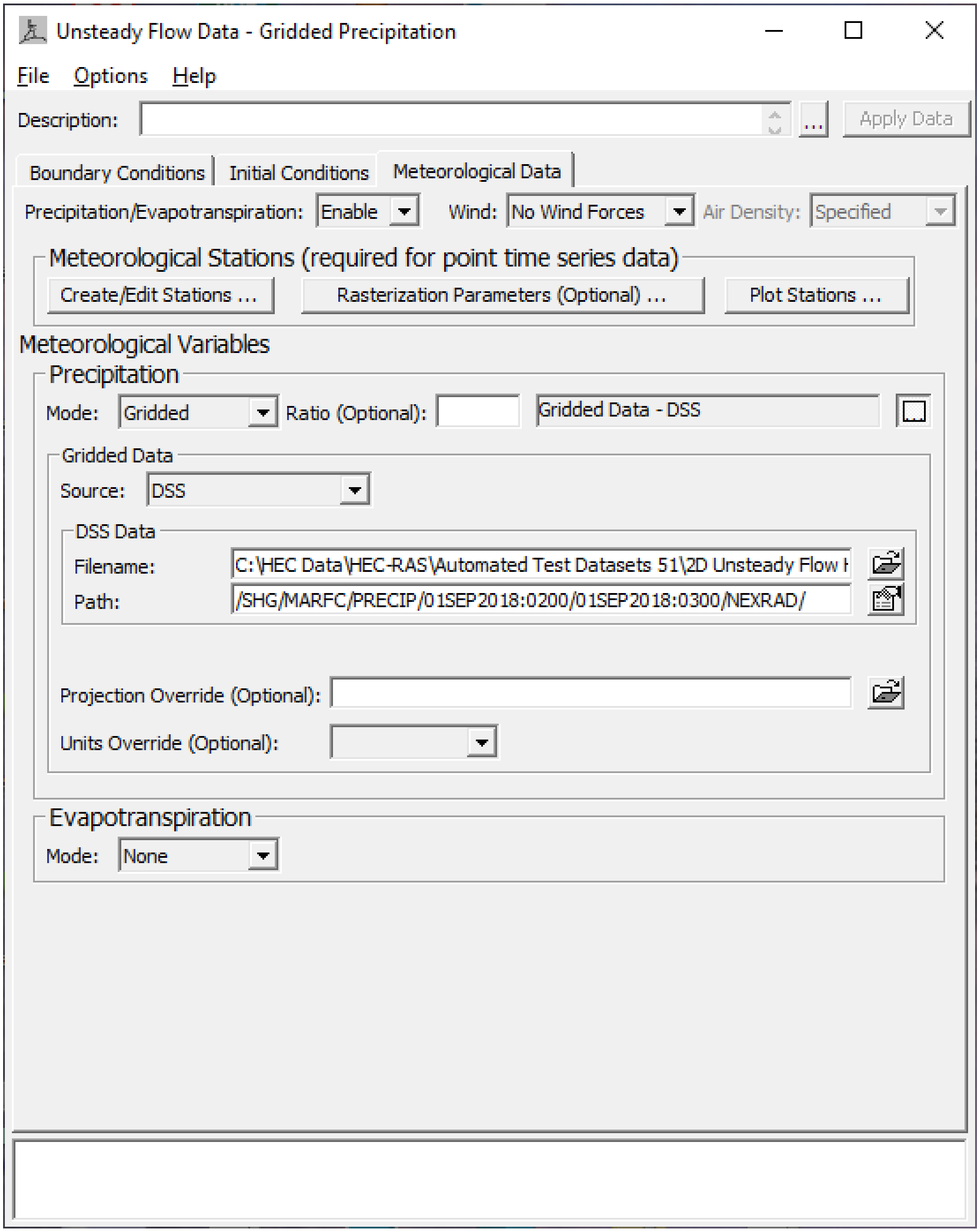


Figure 4-6. Unsteady Flow Data Window with Gridded Precipitation Data from HEC-DSS.

As shown in Figure 4-6. The user must select a **Source** for the gridded data. The current options for gridded data sources are: HEC-DSS and GDAL Raster Files (Single Raster File or Multiple Raster Files). If the “GDAL Raster Files” option is selected, that data must be in either the **NetCDF** or the **GRIB** file formats. These are National Weather Service file formats for their gridded data. Once the user selects a data source, then the user should use the Open File icon next to the filename field to select the source file. If a **DSS** file type is selected, a DSS Pathname chooser will open once the DSS file is selected. The user must choose the DSS pathname that want to use for reading the gridded precipitation data from DSS. If the data is being read from either the NetCDF or GRIB file formats, the user must go through an import process to bring the data in before it can be used. This import process is explained below under the Wind data section (it is the same process for precipitation data).

Additional options for the gridded data are: Ratio; Projection Override; and Units Override. The **Ratio** option allows the user to multiple all of the precipitation by a user specified ratio. For example, if the user enters “5” in the ration field, all of the precipitation values will be multiplied by 5 before being used. The **Projection Override** option allows the user to override the horizontal projection found in the file being read in. This is generally only need if the software is unable to understand the projection information stored in the gridded data file. The **Units Override** option allows the user to override what the software thinks the units of the data are. This is also only needed if the software has trouble detecting the units of the data from the meta data contained within the gridded data file.

Once the gridded precipitation is defined in the Unsteady Flow Data editor, and saved, the user can visualize the precipitation data in HEC-RAS Mapper. Gridded precipitation can be viewed as spatial maps in both an incremental (ex. Hourly) and a accumulated form. This data is available from the **Event Conditions** portion of the tree in HEC-RAS Mapper. Shown in Figure 4-7 is an example incremental precipitation map. These maps can be animated over time by pressing the green **Play** button at the upper left of the window.

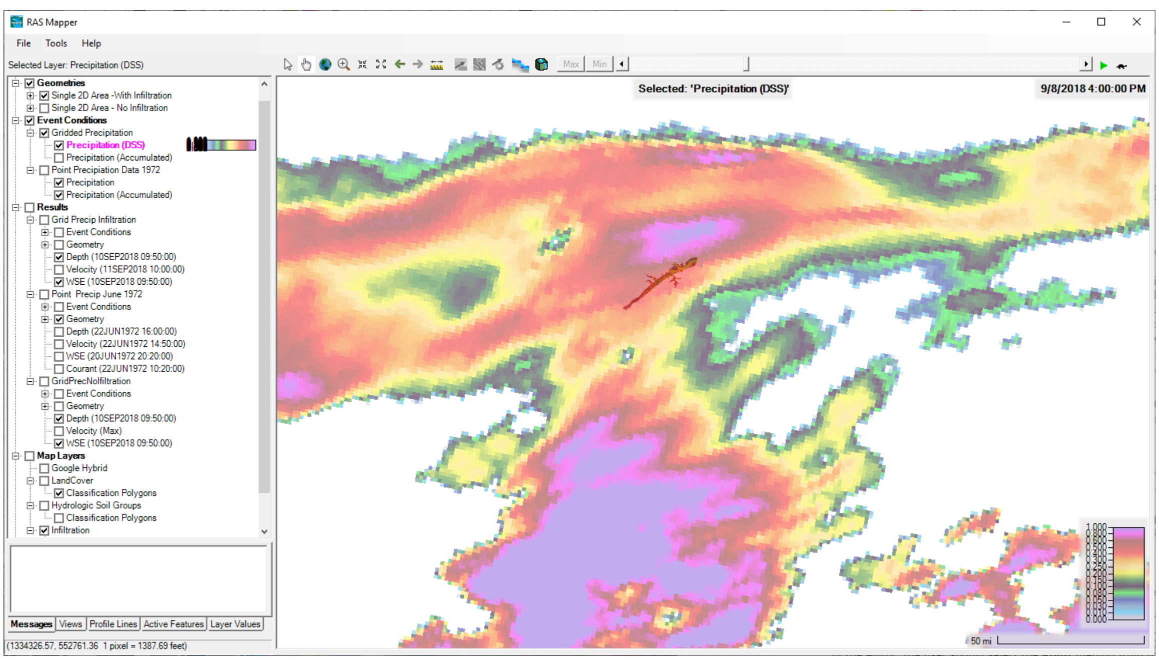


Figure 4-7. Example Incremental Precipitation Map.

Shown in Figure 4-8 is an example of an accumulated precipitation map.

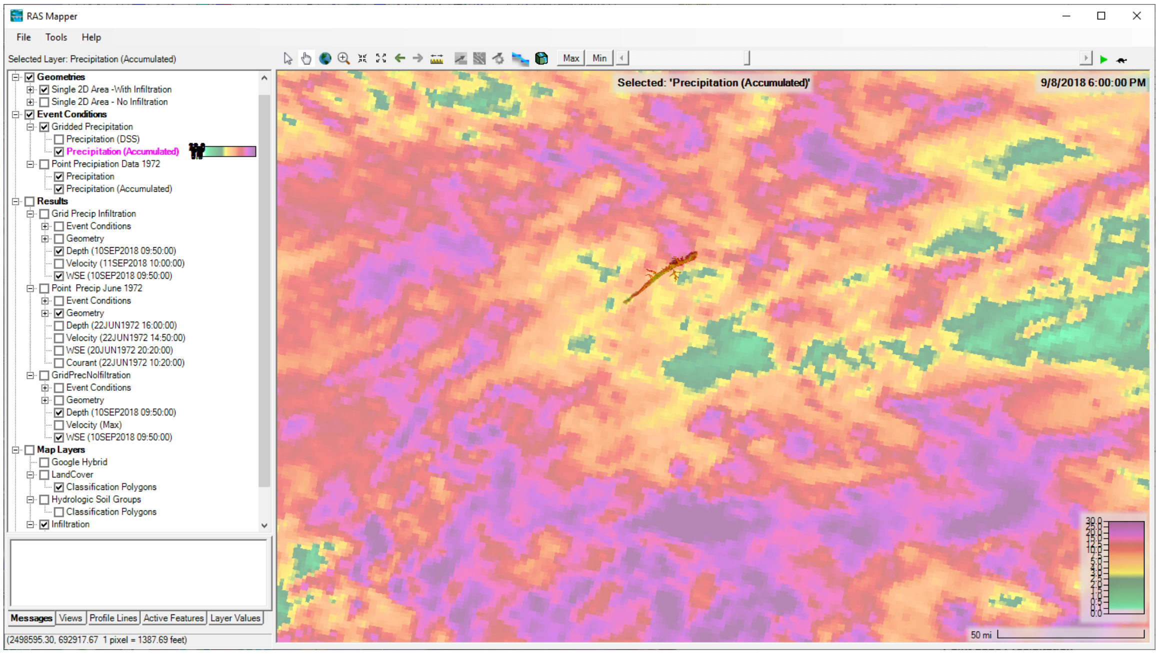


Figure 4-8. Example Accumulated Precipitation Map.

In addition to spatial maps, precipitation can be plotted at a point location for both incremental and accumulated precipitation data. With the two map types turned on, right click at any location and a popup menu will appear allowing you to select either: **Plot Precipitation** or **Plot Accumulated Precipitation.** An example of a plot of incremental precipitation at a point location is shown in Figure 4-9.

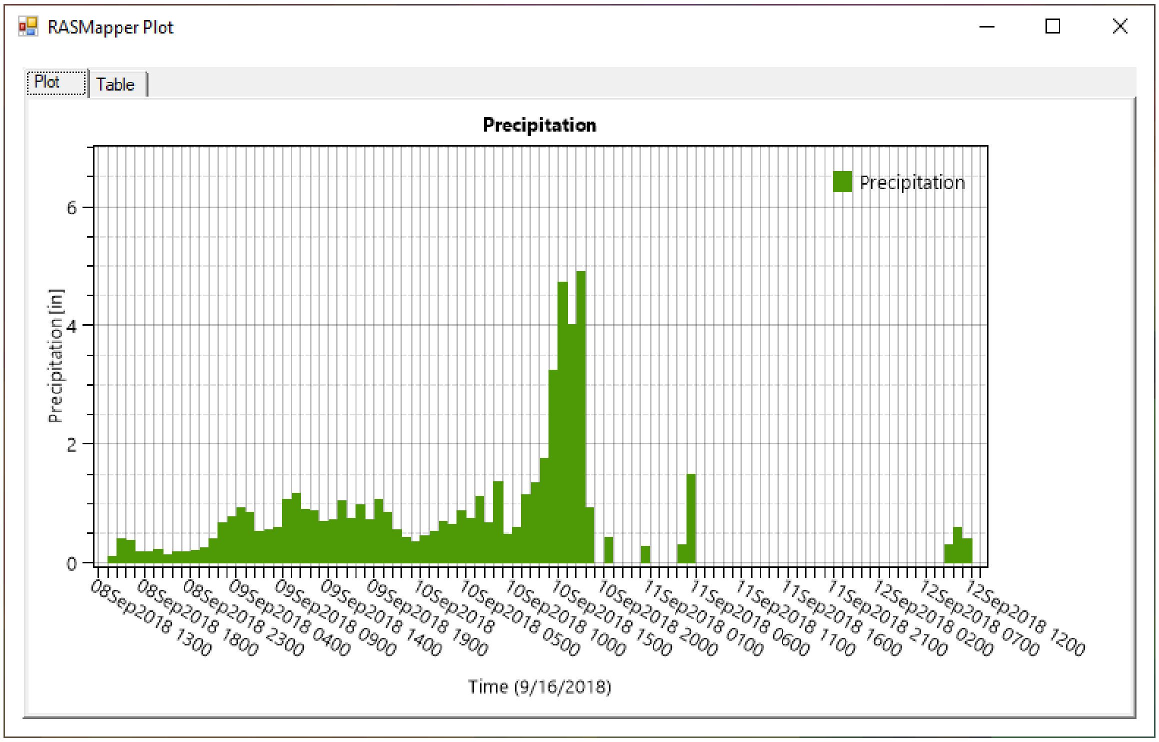


Figure 4-9. Example incremental precipitation plot at a point location.

An example of an accumulated precipitation plot from a point location is shown in Figure 4-10.

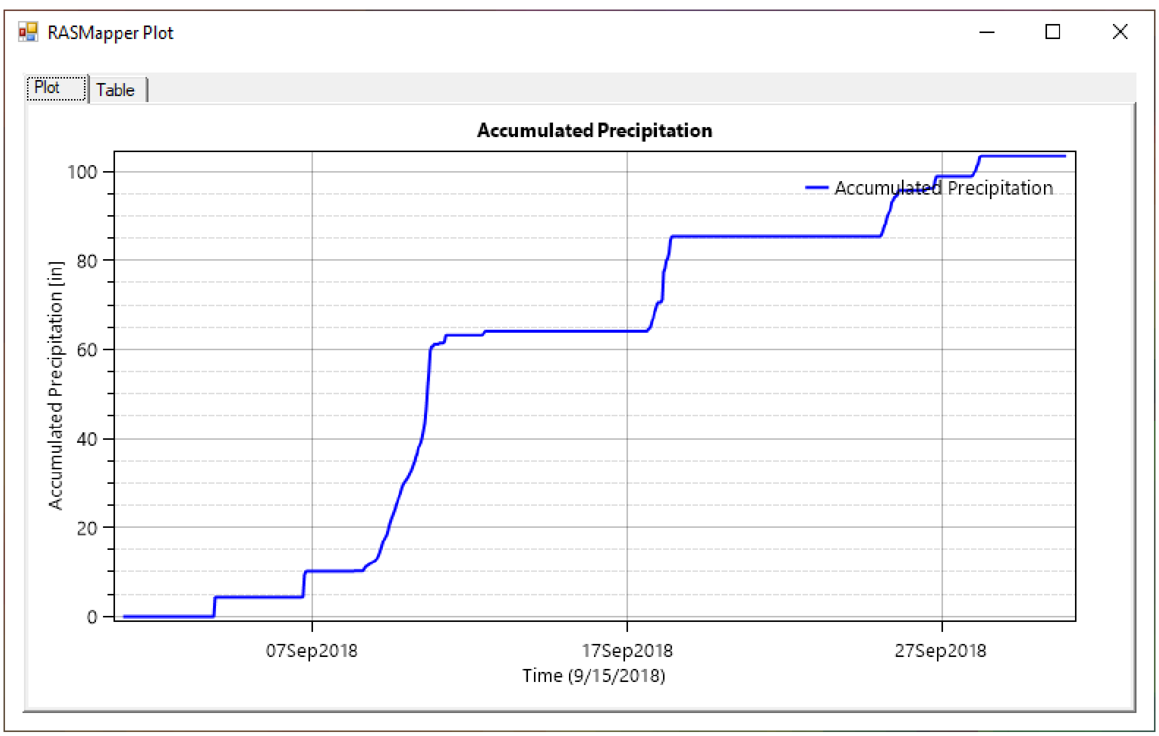


Figure 4-10. Example of an Accumulated Precipitation Plot from a Point Location.

Once the User makes a model run, this same type of information and plotting is available under the **Results|Event Conditions** area for a specific Plan.

Point gage Precipitation

User's can also enter precipitation data as point gage data in HEC-RAS. This data is then interpolated in order to convert it to a gridded data form, which is how the computational engines use it.

To enter point gage data into HEC-RAS, open up the **Unsteady Flow Data** editor, then select the **Meteorological Data** tab. The editor will look as shown in Figure 4-11 below.

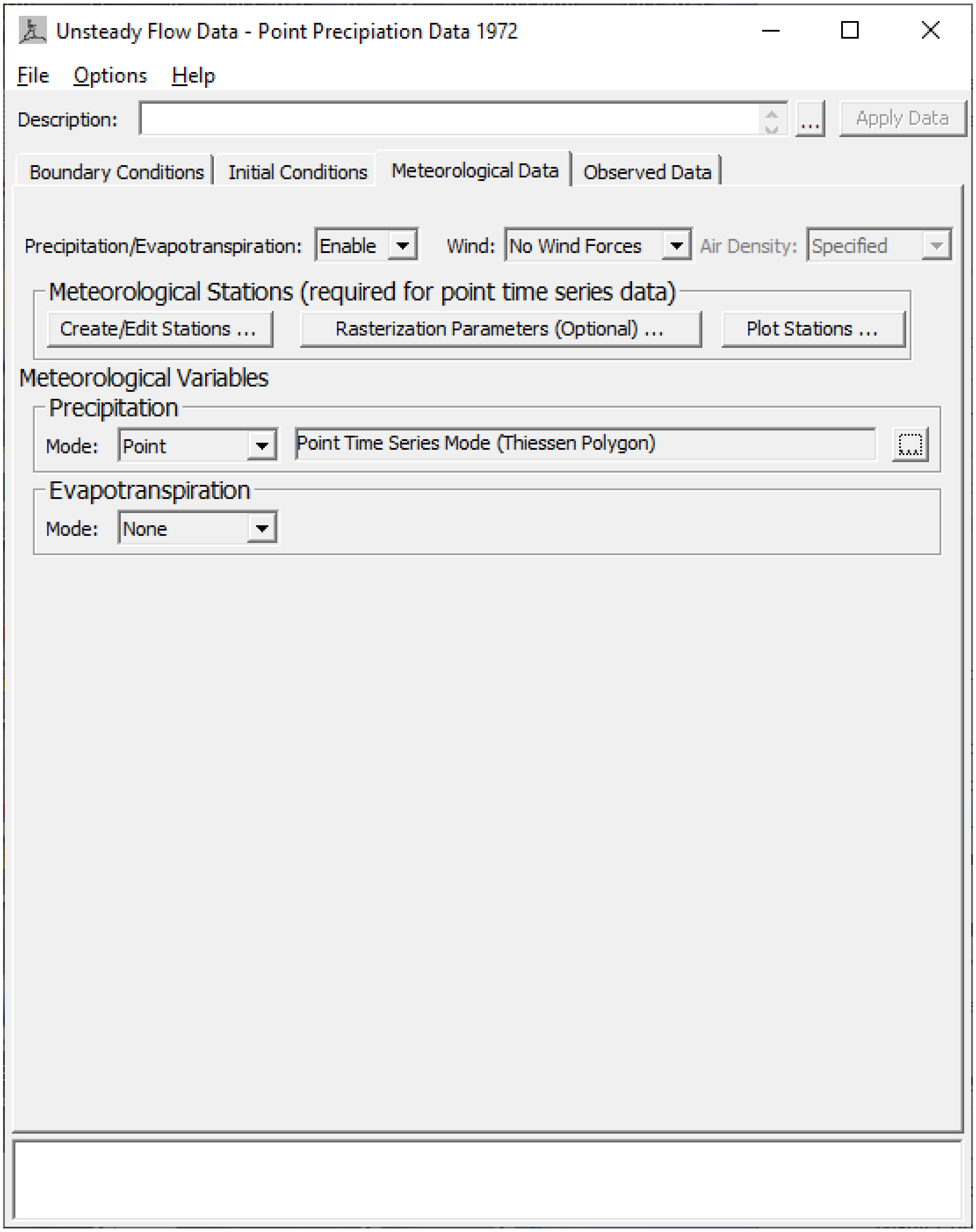
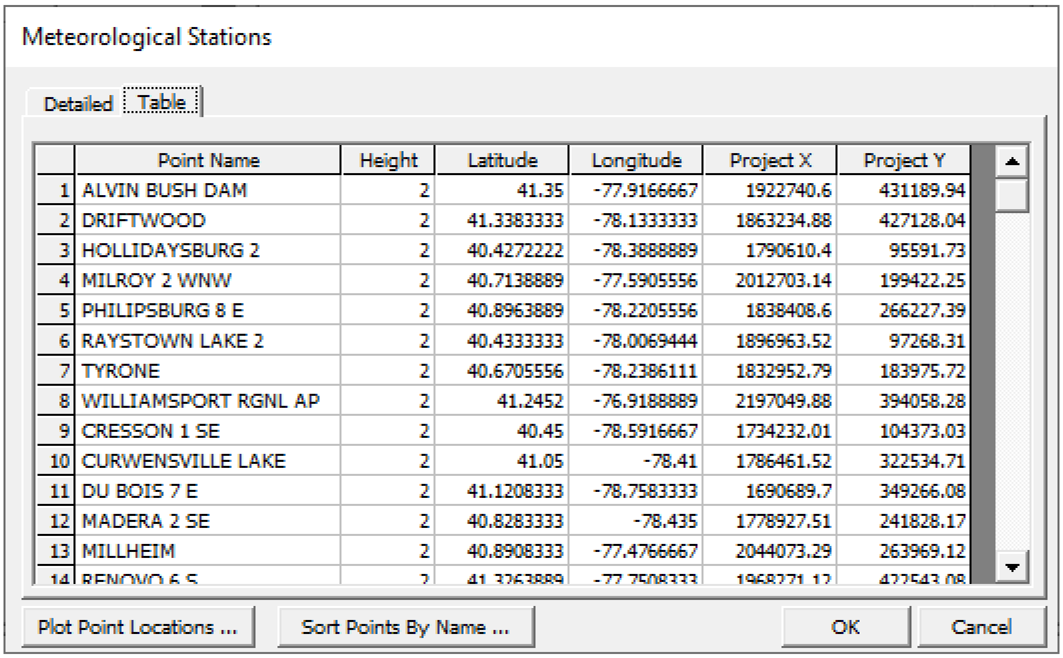


Figure 4-11. Meteorological Data tab with point gage data being used.

To enter/edit gaged precipitation data, the user must press the **Create/Edit Stations** button first. This will bring up a table as shown in Figure 4-12. The Meteorological Stations table is used to enter the **Name** of the gage, and the **X, Y coordinates** of the gage location. Users can enter the gage location in either Latitude/Longitude, or in the project X, Y coordinate system (This is based on what you set up in HEC-RAS Mapper for the horizontal coordinate system of the project). Users must first fill out all of the gage names/locations. The Tab labelled "**Detailed**", allows the user to create one gage at a time, rename a gage, and delete a gage. Then, under the **Meteorological Variables** section of the editor, the user should select the **Point** method from the Precipitation **Mode** drop down. Next press the **Expand/Edit** button to view the additional point data information. The **Point Time Series Data** will show up, allowing the user to select one of the **Interpolation Methods** and to **edit/Enter** data for each of the individual point gages. See Figure 4-13 below.



	Point Name	Height	Latitude	Longitude	Project X	Project Y
1	ALVIN BUSH DAM	2	41.35	-77.9166667	1922740.6	431189.94
2	DRIFTWOOD	2	41.3383333	-78.1333333	1863234.88	427128.04
3	HOLLIDAYSBURG 2	2	40.4272222	-78.3888889	1790610.4	95591.73
4	MILROY 2 WNW	2	40.7138889	-77.5905556	2012703.14	199422.25
5	PHILIPSBURG 8 E	2	40.8963889	-78.2205556	1838408.6	266227.39
6	RAYSTOWN LAKE 2	2	40.4333333	-78.0069444	1896963.52	97268.31
7	TYRONE	2	40.6705556	-78.2386111	1832952.79	183975.72
8	WILLIAMSPORT RGNL AP	2	41.2452	-76.9188889	2197049.88	394058.28
9	CRESSON 1 SE	2	40.45	-78.5916667	1734232.01	104373.03
10	CURWENSVILLE LAKE	2	41.05	-78.41	1786461.52	322534.71
11	DU BOIS 7 E	2	41.1208333	-78.7583333	1690689.7	349266.08
12	MADERA 2 SE	2	40.8283333	-78.435	1778927.51	241828.17
13	MILLHEIM	2	40.8908333	-77.4766667	2044073.29	263969.12
14	RENOVO 6 S	2	41.3263889	-77.7508333	1968271.12	422543.08

Figure 4-12. Point gage Meteorological Station Table for entering Gage Names and locations.

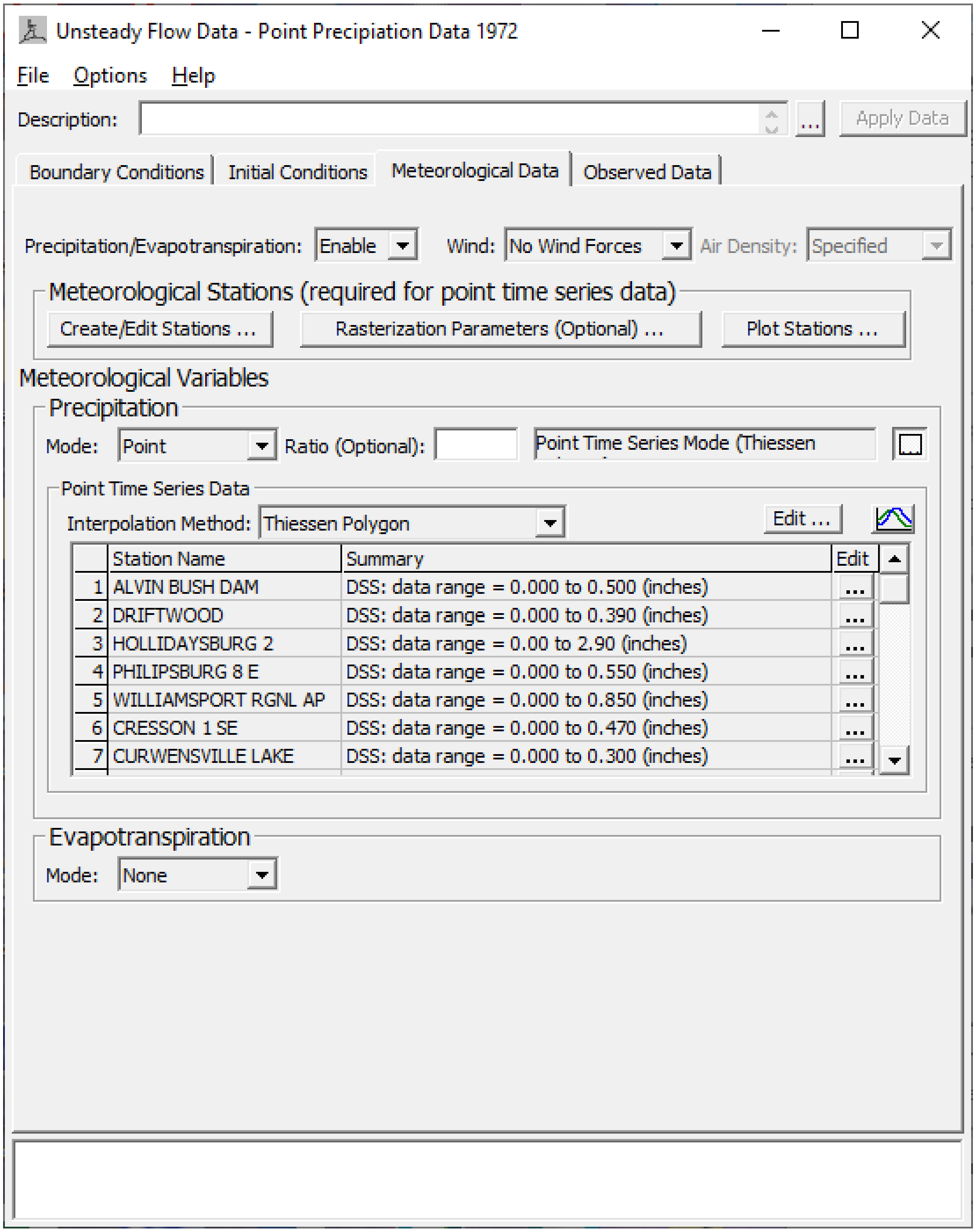


Figure 4-13. Meteorological Data tab with Point Precipitation data shown.

For each of the gages that the user would like to use for the event being modelled, the user must **Edit** each of the individual point gages in order to define the precipitation and turn that gage on for use. When the **Edit** button next to a gage is pressed, an edit window will appear for that gage as shown in Figure 4-14.

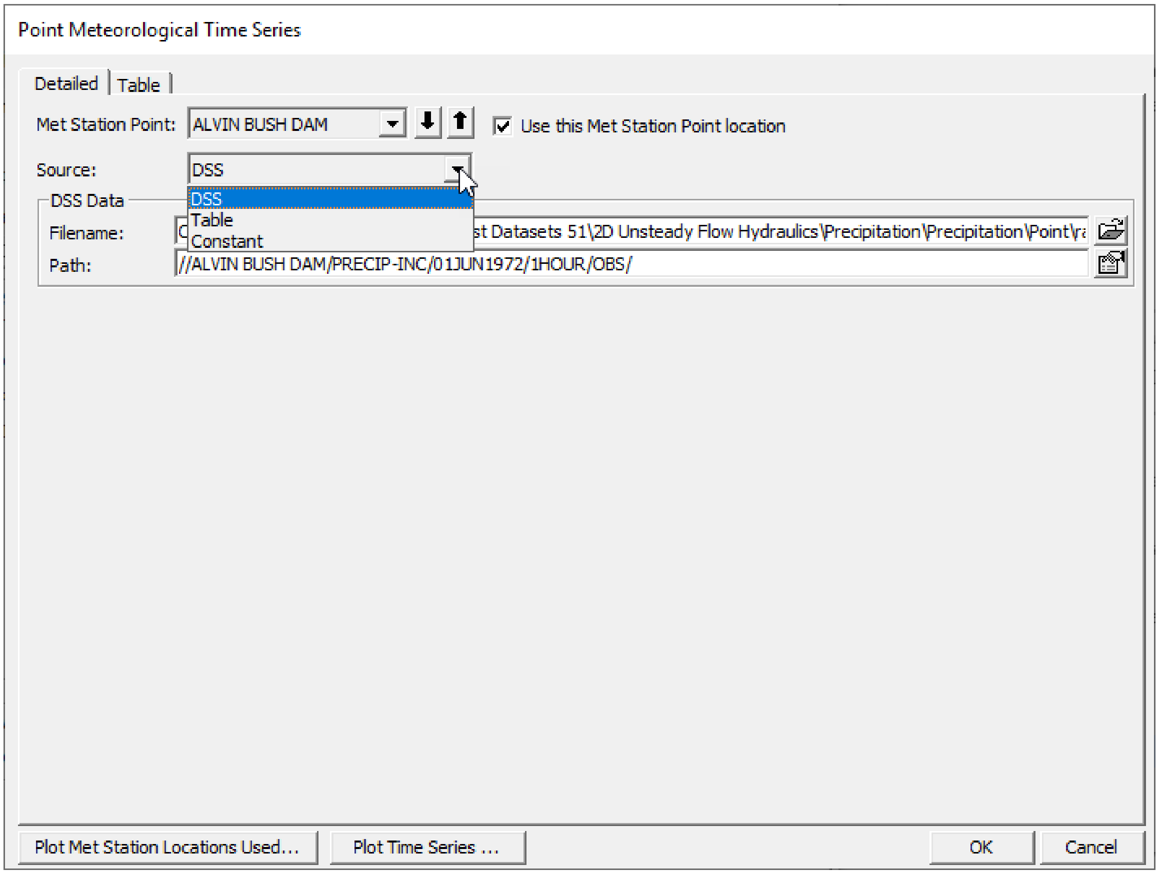


Figure 4-14. Point Precipitation Gage editor.

As shown in Figure 4-14, the user must select the "Source" for the data. The data source can be: DSS (HEC-DSS); Table; or a Constant value (ex. 1.5 in/hr). If the user chooses DSS, they must select the appropriate HEC-DSS file to connect to, and also the specific HEC-DSS pathname that contains the correct data for the specific gage and the event being modelled.

For the point gage data to be used in the analysis, the user MUST check the box labeled "**Use this point location**". If this box is not checked, it will not be used in the analysis. This option allows user to turn on/off various gages in order to experiment with the effect of different gages being included or not. The user must do this for all of the gages they wish to use in the analysis.

Once you have entered data for all of the desired point gages, and you checked the box to use those gages, the last steps are to set the **Rasterization Parameters** and to select the **Interpolation Method**. To set the Rasterization parameters, press the button labeled **Rasterization Parameters (Optional)** from the Meteorological Data tab. This will bring up the following window:

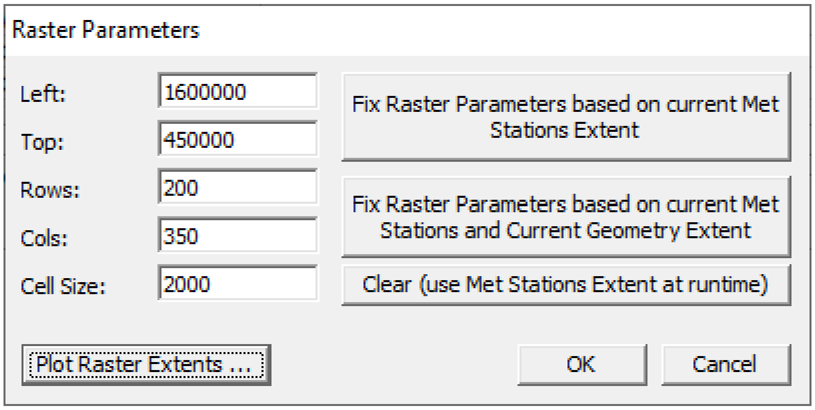


Figure 4-15. Editor for controlling Rasterization of point gage data.

As shown in Figure 4-15, the user can define the coordinate extents for how the point gage data will be rasterized. There are several options for doing this. The first option is that the user can completely define the extents and the cell size to use on their own. This is accomplished by entering a **Left** and **Top** coordinate (in the coordinate system established for the model in RAS Mapper). Then entering the number of **Rows**, **Cols** (columns), and the **Cell Size.** The Cell Size is entered in ft when working in English units, and in meters when working in metric units. The second approach is to use one of the three buttons on the right hand side of the window. These three options are: **Fix Raster Parameters based on the current Met Station Extents**; **Fix Raster Parameters based on the current Met Stations and Current Geometry**; and finally **Clear (use Met Stations at runtime).** The last option is often used for real time forecasting, in which the number of available gages that are recording may change.

To understand exactly what you are going to get for the rasterization of the point gage data, press the botton labeled **Plot Raster Extents...** This will bring up a plot so you can see what the extents would be for each of the possible options. Se an example below in Figure 4-16.

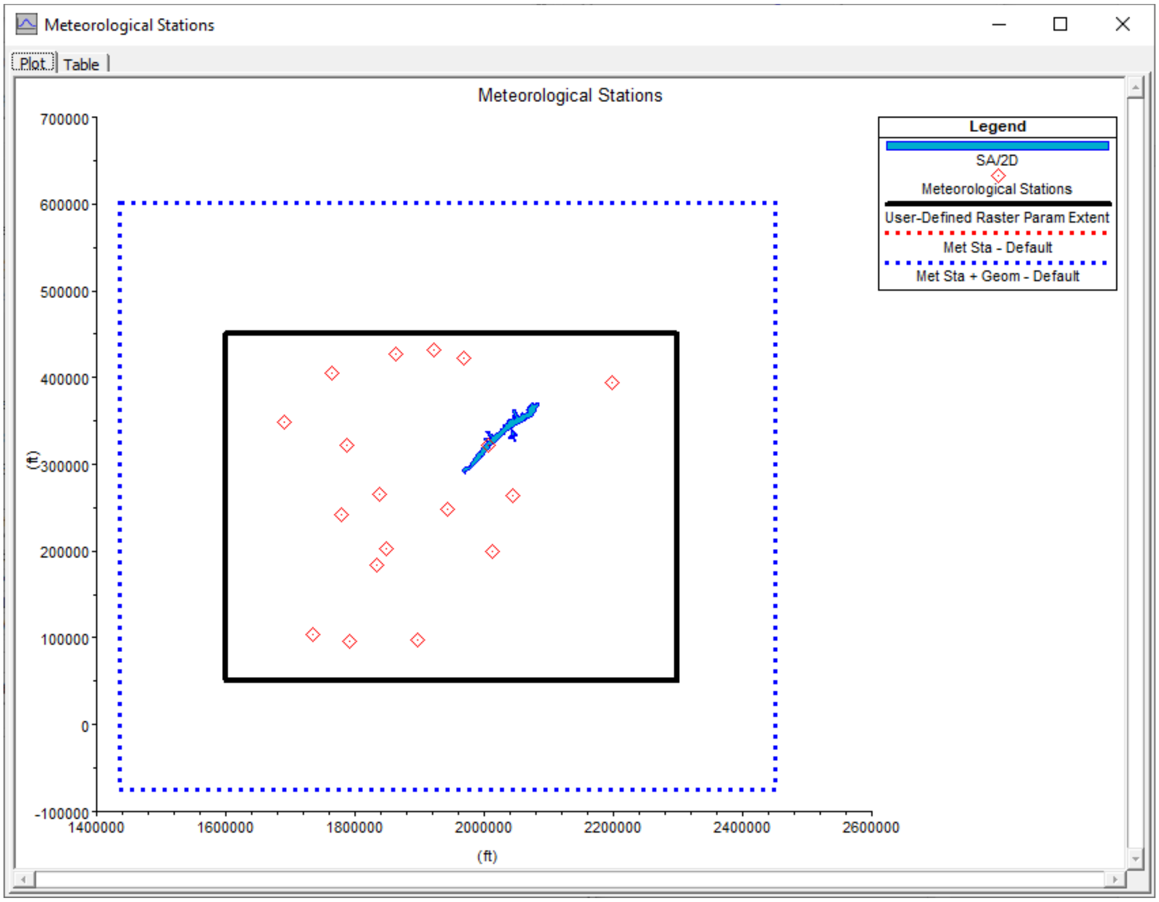


Figure 4-16. Plot showing all the possible options for rasterizing the point gage data.

The final thing that the user must do is select the **Interpolation Method.** There are four point gage Interpolation Methods available to the user, they are: Thiessen Polygon; Inverse Distance Squared; Inverse Distance Squared (Restricted); and Peak Preservation.

The **Thiessen Polygon method** is the traditional method often use in Hydrology. This method draws A line between the two point gages, and finds half the distance between each gage. This is done for all of the gages, and then polygons are formed by the linear bisection lines between the gages. HEC-RAS only does this in order to figure out which gage should be used for the pattern of the rainfall event. If a 2D cell is closet to a specific gage (Inside that gages polygon), then that gage will be used for the pattern (Intensity vs time) for the rainfall event. However, for the storm total rainfall at each cell, the inverse square of the distance method is used in order to get a weighted storm total rainfall for each cell. That storm total is then applied to the nearest gages pattern in order to define the rainfall for that specific cell.

The **Inverse Square of the Distance** method computes a weighted depth of rainfall for each time period individually, by using the inverse square of the distance weighting method for all of the gages near a specific cell. If the rainfall data is hourly data, then this is done individually for each one hour time step.

The **Inverse Distance Squared (Restricted)** method does the same thing as the Inverse Square of the Distance method, except it first triangulates all of the gages. Then, if a cell lies inside of a specific triangle, then only the three gages that form the triangle are used in the storm weighting process. This prevents gages that are far away from being used, and limits the interpolation to the three closest gages to the cell.

The **Peak Preservation** method is an attempt to retain the intensities within a storm as it moves across a watershed. The Inverse Square of the Distance methods have the problem of diminishing the rainfall intensities when the timing of the rainfall is different at each of the gages being used for the interpolation. The peak preservation method, takes all of the gages being used for a particular cell, then it finds the center of mass of the precipitation for each of the gages. Next it lines up the gaged data by the center of mass. Then the data is interpolated on a time step by time step basis. Finally, the interpolated data is then shifted back to the correct time, to account for the location of the cell between the gages.

In order to test out the different interpolation methods, we have added the ability to visualize the interpolation of the precipitation data inside of HEC-RAS Mapper. HEC-RAS Mapper now contains a Layer called **Event Conditions**. This new layer will allow you to view/animate precipitation data (gridded and point data), and wind data. To view the Point Precipitation data and its resulting interpolated values, turn on the precipitation layer that corresponds to the data you entered into the Unsteady Flow Data editor. An example of this for Point Precipitation, using the Thiessen Polygon interpolation method is shown in Figure 4-17 below.

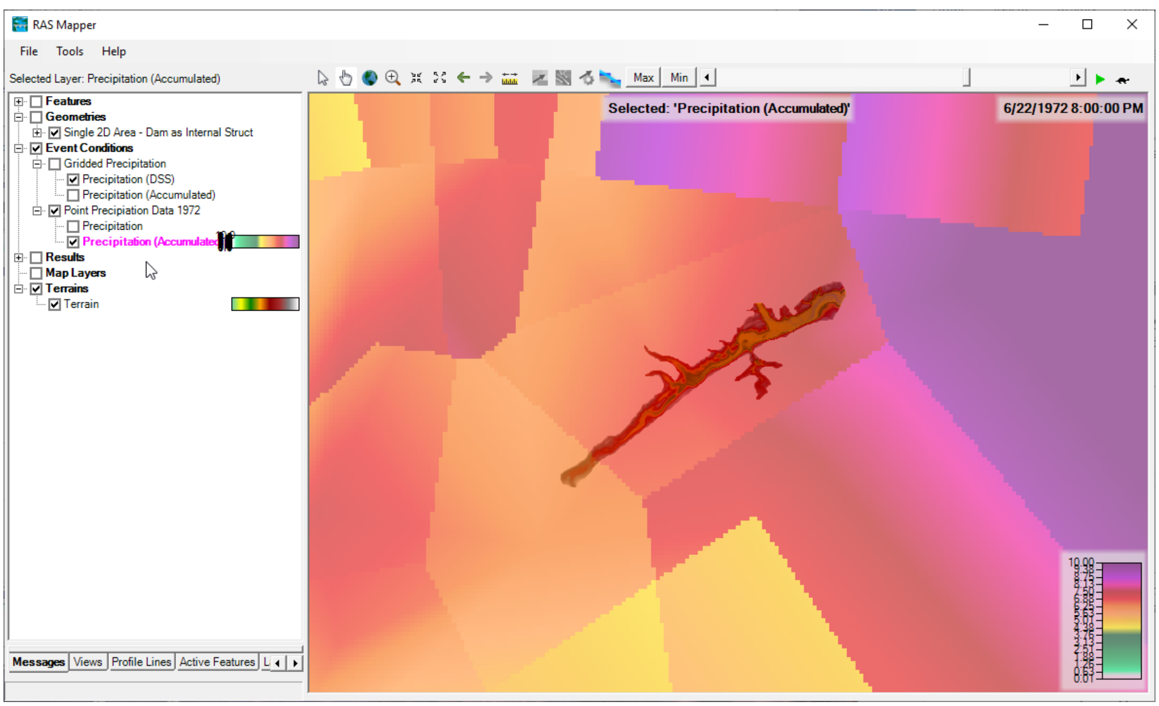


Figure 4-17. Point Precipitation with the Thiessen Polygon Interpolation Method.

HEC-RAS Mapper has a layer for the incremental precipitation, called **Precipitation**. There is also a layer for the cumulative precipitation, called **Precipitation (Accumulated)**. User can turn on ether of the precipitation layers. These layers can be animated, or a specific point in time can be selected. Additionally the user can right click on the map and request one of three different time series plots. The time series plotting options are: **Plot Precipitation** (this is the incremental precipitation vs time plot); **Plot Accumulated Precipitation** (accumulated precip vs time); and **Compare Precipitation Methods (accum)** (this option plots all four of the point precipitation methods for the accumulated rainfall vs time). Shown in Figure 4-18 is an example of an Incremental Precipitation time series plot.

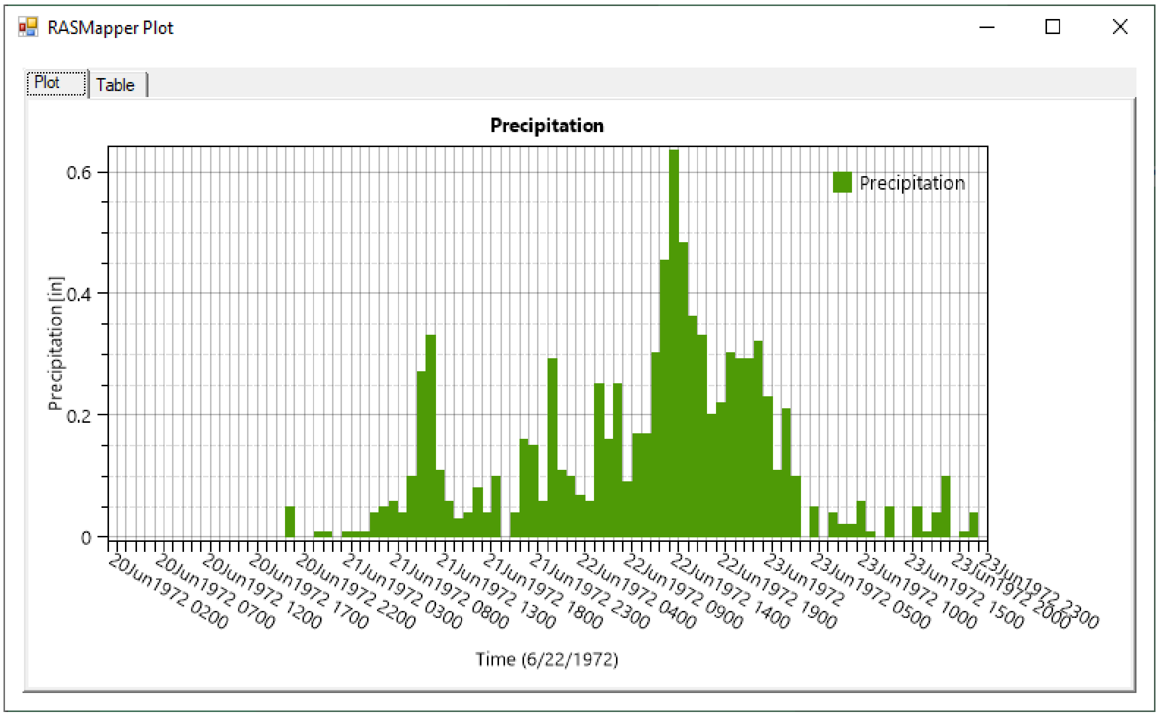


Figure 4-18. Example Incremental Prcipitation Time Series Plot.

Shown in Figure 4-19 is an example Accumulate Precipitation vs Time plot.

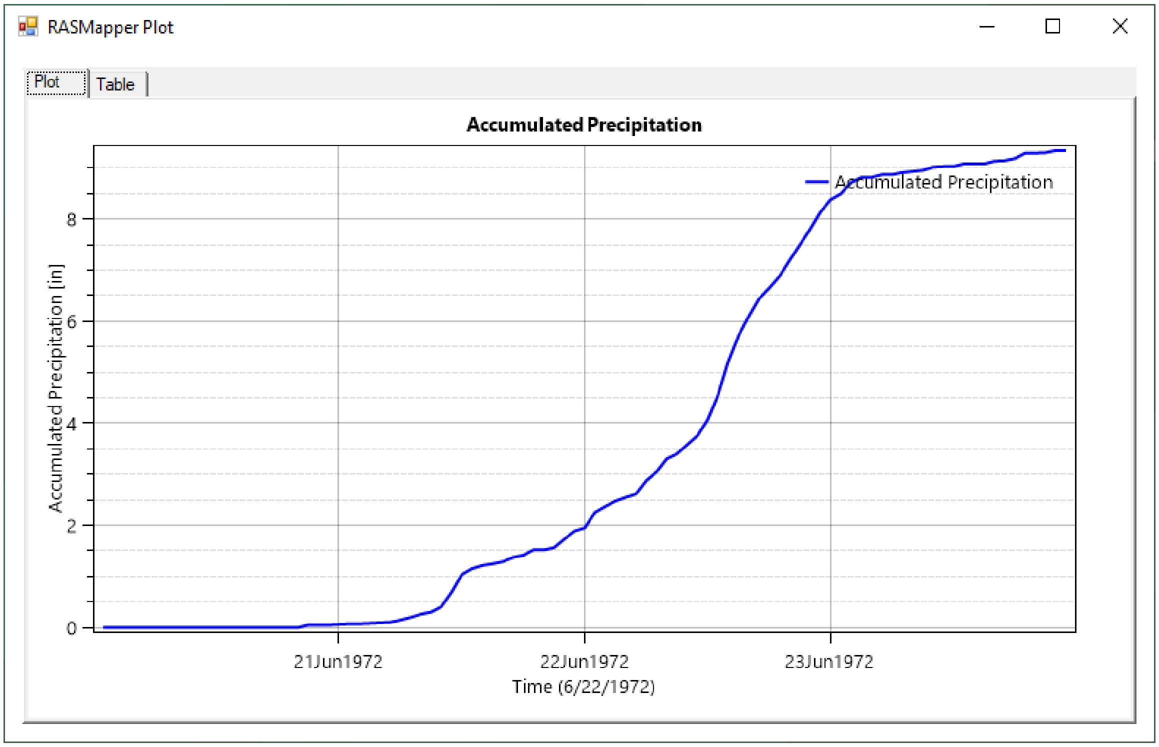


Figure 4-19. Example of Acumulated Precipitation vs Time Plot.

Shown in Figure 4-20 is an example of Accumulated Precipitation versus time for all four available precipitation interpolation methods.

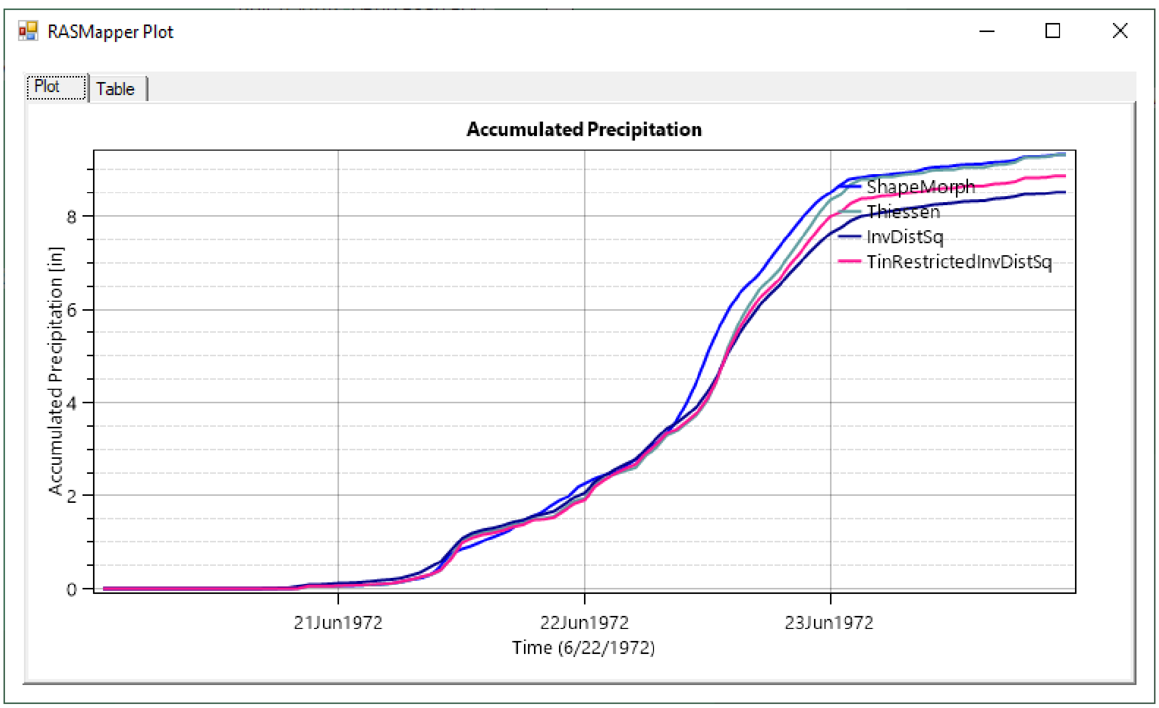


Figure 4-20. Example Accumulated Precipitation versus time for all four Interpolation Methods.

Additionally, users can right click on the **Precipitation** layer and select the menu option called: **Rasterization Parameters**. This menu option will bring up a sub menu, allowing the user to select one of the four point precipitation interpolation methods; define how the program extrapolates outside of the point gage data; select a resampling technique; and also selecting the cell size to use when rasterizing the precipitation data (default is cell size is 2 kilometers). Users can select a smaller cell size for rasterizing the precipitation data, however, there must be enough point gages close enough together in order to justify using a smaller precipitation cell size.

NOTE: While HEC-RAS Mapper allows you to select different interpolation methods, and then visualize the result of using those interpolation methods, this is available to allow the user to experiment with the different methods. Once the user decides which interpolation method they actually want to use in a simulation, the must have that method selected in the Unsteady Flow Data editor, on the Meteorological Data tab. The method selected in the Unsteady Flow Data editor is the actual method that will be used in the simulation.

Evapotranspiration Data

Evapotranspiration data is only needed if the user is utilizing either the Deficit Constant or Green and Ampt infiltration methods. Additionally, Evapotranspiration data is an option for these two methods. It is only needed if the user wants to account for evapotranspiration during dry periods between rainfall events.

Evapotranspiration data can be entered as gridded data, point gage data, or just a constant rate. The method of entering the data is similar to precipitation, as described above.

Wind Forces

Wind forces can be incorporated in both 1D and 2D unsteady flow modeling. Wind data (speed and direction) can be entered as either gridded data or point gage data, just like precipitation data described above. If the user enters wind data into the unsteady flow data editor, that data will be used and wind forcing will be applied to all 1D river reaches and 2D flow areas that are using the Full Shallow Water Equations (SWE).

Gridded Wind Data

To enter gridded wind data (speed and direction), open the Unsteady Flow Data editor and select the **Meteorological Data** tab. From the **Wind Forces** drop down select from one of the three options: No Wind Forces; Speed/Direction; and Velocity X/Y. If you select **No Wind Forces** (Default), then no wind forces are applied to the data set. If you select **Speed/Direction**, then you must enter the wind data as separate speed data and separate direction data, based on degrees from north (North is zero degrees, East is 90 degrees, etc…). If the user selects **Velocity X, Y**, then the wind data is entered as a velocity X component and a velocity Y component for each grid. This approach will define the magnitude and direction. An example with wind defined by Speed/Direction is shown below in Figure 4-21.

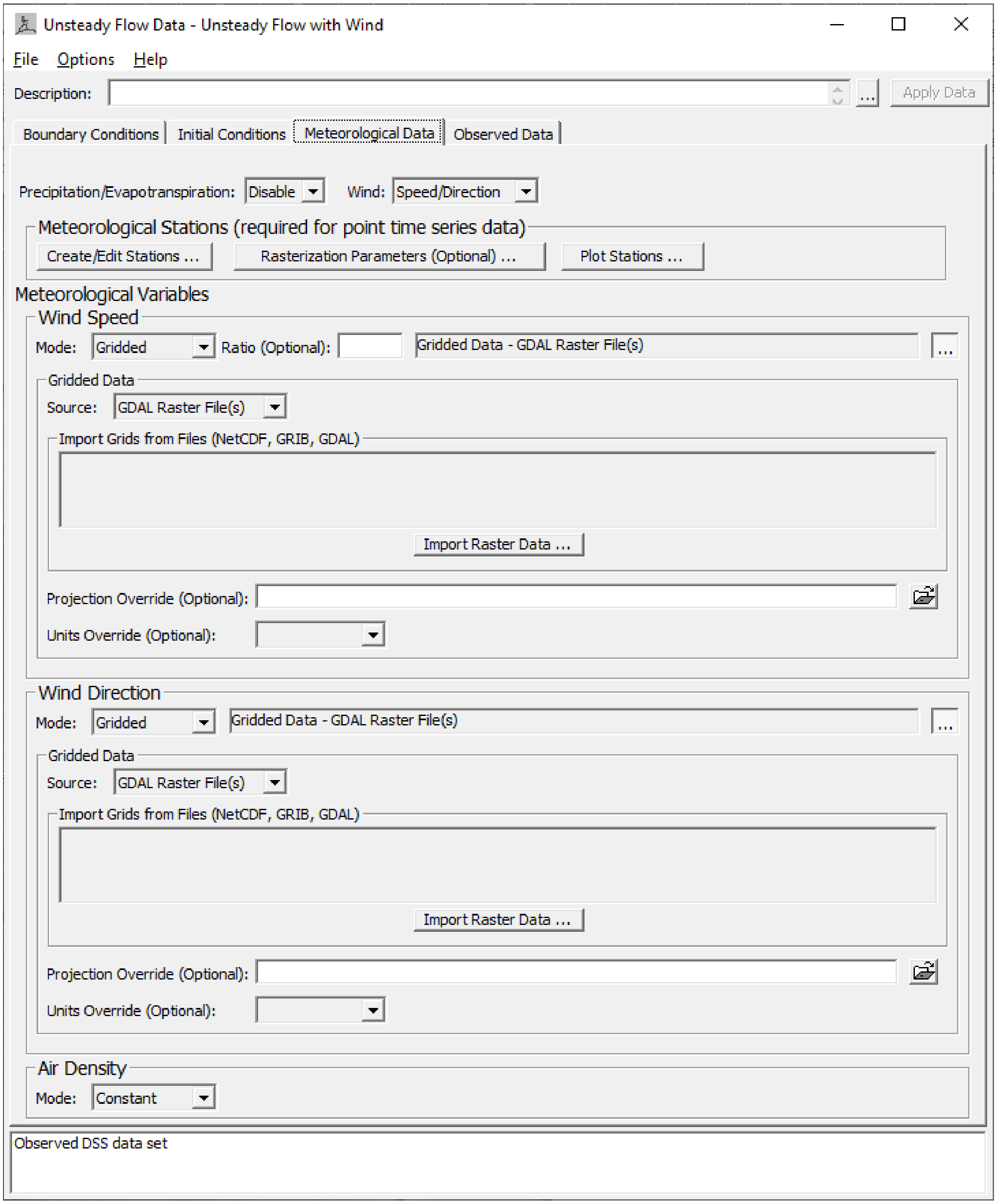


Figure 4-21. Example of Wind data entered as Speed/Direction.

As shown in Figure 4-21, the user must first define the **Mode** for entering the data, such as: **Point; Gridded;** or **Constant.** In the example shown the wind data is being entered as Gridded data. This requires a separate grid for the Wind Speed and the Wind Direction. The options for the **Source** of the wind data are: **DSS; Single Raster File;** and **Multiple Raster Files**.

If DSS is selected the user must specify the HEC-DSS file containing the data, and the DSS pathname pointing to the gridded information. If a Single Raster File is selected, that means a single Raster file contains all of the spatial data for the entire time window of the event being modelled. The Raster files can be either a NetCDF file or a GRIB file (National Weather Service formats). If the Multiple Raster Files Source is selected, that means that more than one files contains the data for the entire time window being modelled. The same two file formats are available, NetCDF and GRIB.

For the Raster file data source, the user must specify the location and name of the file(s), as well as which group within the file contains the specific data (speed or direction; velocity X and velocity Y). As shown in Figure 4-21, once the user has specified "Gridded" data and the source as "GDAL Raster Files", then the user must press the **Import Raster Data** button to go through the import process. When the **Import Raster Data** button is pressed, then a window will appear as shown in Figure 4-22.

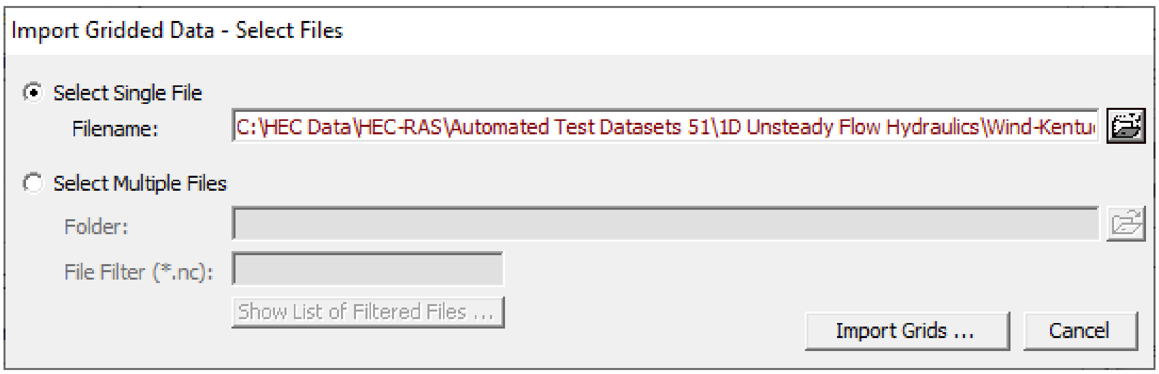


Figure 4-22. Gridded Rastewr Data Importer.

As shown in Figure 4-22, the user selects from importing either a Single Raster that contains all the data or Multiple Raster files. Once either the Single or Multiple Raster files are selected, the user presses the **Import Grids** button to import the data. If the data is from a NetCDF file, a new window will appear, asking the user to select from the discovered data sets in the file, which one to import. For example, if the user is trying to import the Wind Speed, data, they should select the header/data set that is the wind speed data (usually labeled "US"). If the user is trying to import the wind direction data, they need to select the wind direction header/data set (usually labeled "UD").

Because the NetCDF and GRIB file formats can contain Meta data in different formats, there are two options to override certain pieces of needed information. These options are: **Projection Override** and **Units Override**. These fields are optional and should only be used if HEC-RAS is having trouble reading the projection and/or units of the data.

Once the gridded data is entered into the Unsteady Flow Data editor, and saved, the user can open HEC-RAS Mapper and visualize the gridded wind data from the **Event Conditions** layer. This data can be animated in time. Users can also right click on the map and ask for a time series of the wind speed and direction at that location (this will bring up two separate plots, one for speed and one for direction). An example plot of a gridded wind field is shown in Figure 4-23 below.

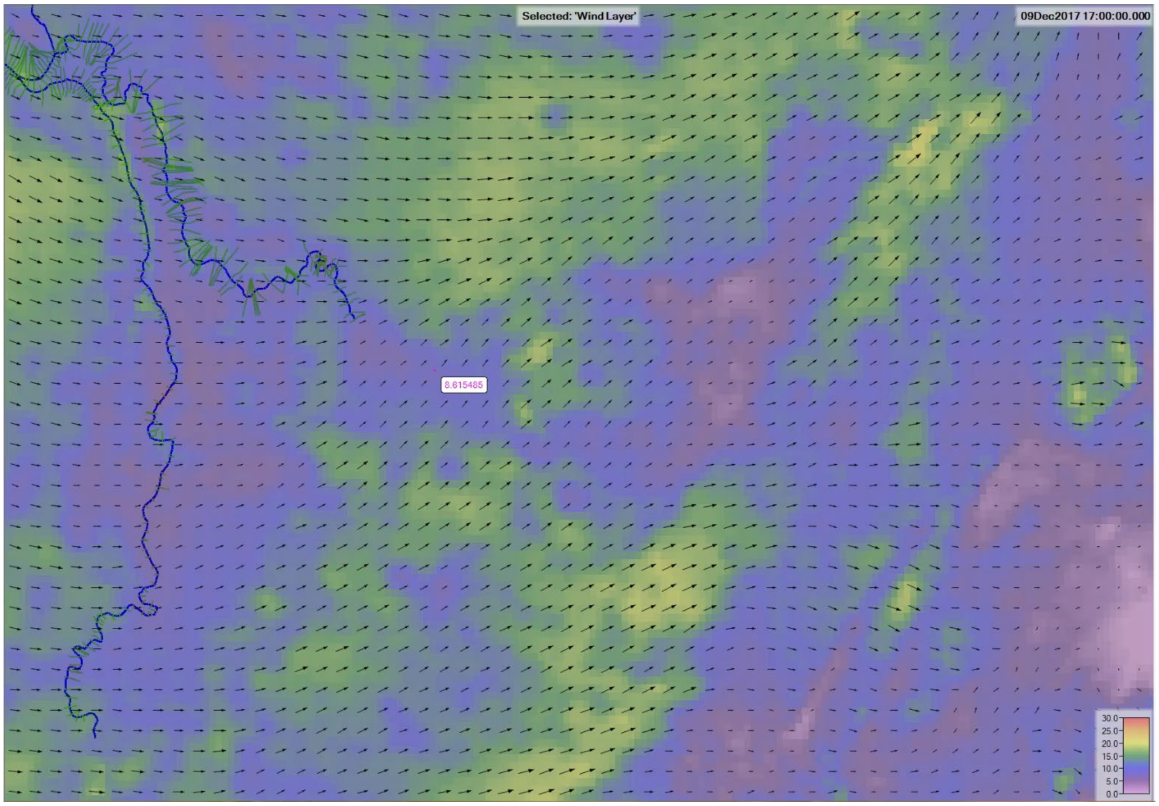


Figure 4-23. Example gridded wind field for 1D and 2D modeling.

Point Gage Wind Data

User's can also enter wind data as point gage data in HEC-RAS. This data is then interpolated in order to convert it to a gridded data form, which is how the computational engines use it. Point gage data for wind is entered in the same manner as point age precipitation data, which is document above. Please refer to the point gage precipitation data to understand how to enter point gage data for wind gages.

Initial Conditions

Initial conditions for 2D flow areas can be accomplished in several ways. 2D flow areas can: start completely dry; be set to a single water surface elevation; set by using a **Restart File** from a previous run; set by **interpolating** results from an existing HEC-RAS **Results File** for a specific Plan, date, and time; or they can be established using the **2D Initial Conditions Ramp up Time** option at the beginning of the run.

Dry Initial Condition

Nothing needs to be done to start a 2D flow area in a dry condition, this is the default option. The name of the 2D flow area will show up under the **Initial Conditions** Tab of the **Unsteady Flow Data** editor (See Figure 4-24). Just leave the initial condition elevation column blank, and this tells the software to start the 2D flow area dry. **Note: a 2D area connected directly to the upstream end or the downstream end of 1D reach, cannot start dry (see previous discussion).**

Single Water Surface Elevation

When the single water surface elevation option is used, every cell that has a lower terrain elevation than the user established water surface will be wet (with a water surface at that elevation), and cells with a terrain elevation that is higher than that water surface will be dry. To use this option, just put in the water surface elevation desired in the "Initial Elevation column of the **Unsteady Flow Data** editor/**Initial Conditions** tab, and in the row for the 2D flow area (see Figure 4-24).

Restart File

A **Restart File** can be used to establish initial conditions for an entire HEC-RAS model. This is a well-documented option under the **Unsteady Flow Data Editor** documentation in the HEC-RAS User's Manual. If a previous run has been made, and the option to write out a **Restart File** was used, then a Restart File can be used as the initial conditions for a subsequent run. The **Restart File** option has been modified to allow for restarting 2D flow areas in addition to all of the 1D flow elements in HEC-RAS. For 2D modeling, the Restart File will contain a water surface elevation for every cell in the model. Additionally, restart files can be generated using either of the 2D equation sets (full Saint Venant or Diffusion Wave), and can be used to start a model with a different equation set (i.e. the user can run the original run with the Diffusion Wave option and create a **Restart File**, then start up a model that uses the Full Saint Venant equations from that restart file). See the section on **Initial Conditions** in Chapter 8 of the HEC-RAS User's Manual for more information on how to use the **Restart File** option.

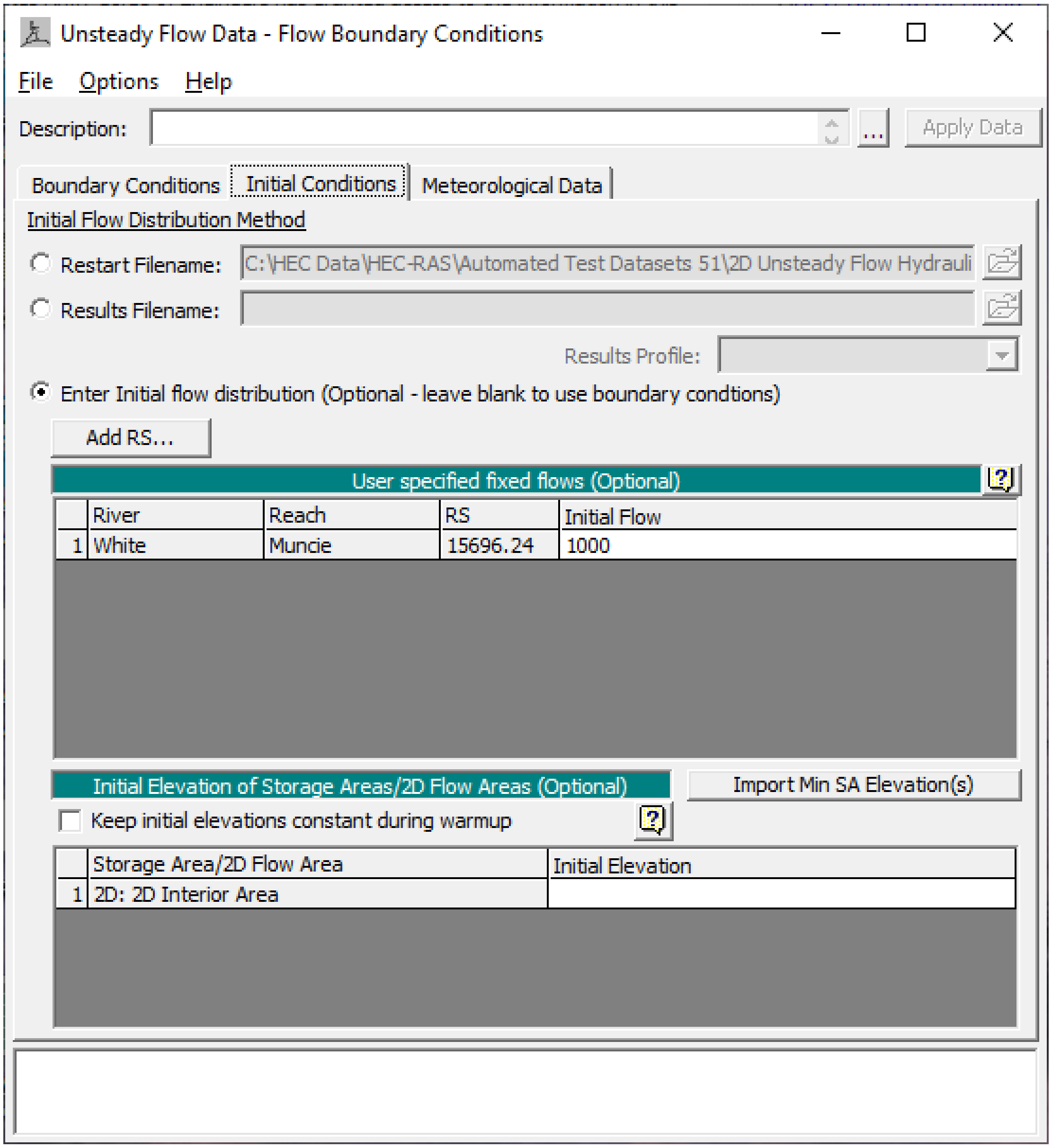


Figure 4-24. Unsteady Flow Data Editor with 2D flow area initial conditions.

Interpolation from Previously Computed Results

Another way to establish the initial conditions of a model (1D, 2D, or combined 1D/2D), is to use the interpolated results from a previously run Plan. This option is extremely flexible, in that it does not have to be from the Plan you are running, and you can change you geometry and it still works. The user selects a previously run Plan, Date, and time of an existing results file. At run time, HEC-RAS will interpolate water surface elevations, velocities, and flows, as needed from the HEC-RAS Mapper output data for the Plan. For example, 2D cells get an interpolated water surface elevation at the cell computation point if the cell is wet. 2D faces get and interpolated normal velocity (Perpendicular velocity) at the center of the face for each face that is wet. 1D cross sections get an interpolated water surface at the location where the river line crosses the cross section. 1D cross sections get an interpolated flow for that cross section. And 1D Storage Areas get and average water surface computed from all the water surfaces contained within the storage area polygon.

To use the option to "**Interpolate from Previously Computed Results File**", go to the **Initial Conditions** Tab on the Unsteady Flow Data editor. See Figure 4-25 below.

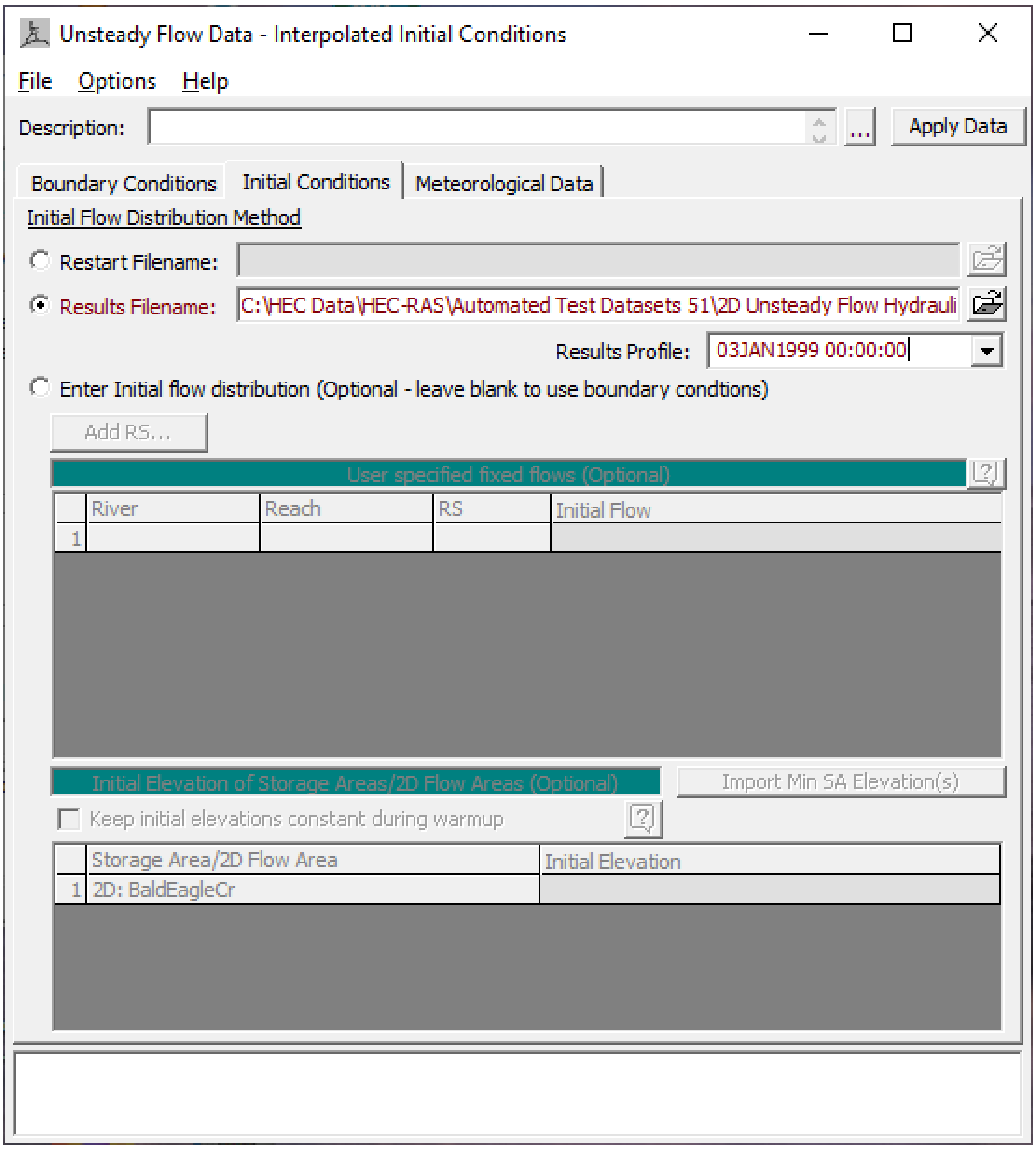


Figure 4-25. Example of Interpolating Initial Conditions from a Previously Run Plan/Date/Time.

As shown in figure 4-25, to use this method for initial conditions select the **Results Filename** option. Once this option is selected the user must first pick the hdf output file from the previously run Plan that they want to interpolate an initial condition from. The hdf output files will look like this: "projectname.p##.hdf". This is the output file that HEC-RAS Mapper uses to create all of the results maps. The next step is to pick an available **Results Profile** (Date and Time) from the selected output file.

As stated previously, at run time HEC-RAS will interpolate results from the computed results of the selected plan/date/time and set those results for the currently run Plan. This method will work even if you change the 2D mesh, 1D cross sections, or storage areas. So this is the most flexible way to establish an initial condition when you have 2D flow areas in a model.

2D flow area Initial Conditions Ramp Up Option.

The unsteady flow capability in HEC-RAS has always had an option to run a model warm-up period. The model starts with the initial conditions, it then holds all of the boundary conditions constant, based on their value at the beginning of the simulation, and then it runs a series of time steps with the constant inflow. This allows the model to settle down to water surface elevations and flows that are consistent with the unsteady flow equations being applied. If there are any lateral structures that have flow (based on the initial conditions) going across the structure, this flow will transition from a very small flow to the full computed flow over the duration of the warm-up period. This can reduce shocks to the system, especially in 1D river reaches.

2D flow areas have an additional option called **Initial Condition Ramp Up Time**. If a 2D area has external boundary conditions (flow hydrographs or stage hydrographs) or links to 1D elements, in which flow will be going into or out of the 2D area right from the start of the simulation, then the 2D flow area **Initial Condition Ramp Up Time** must be turned on to get flow through the 2D area in order to establish its initial conditions before the start of the simulation (or even before the start of the overall model warm-up time). The 2D flow area **Initial Condition Ramp Up Time** is a separate option for the 2D flow areas (separate from the overall model warm-up option). To use this option, select the **Options** menu from the Unsteady Flow Analysis window, then select **Calculation Options and Tolerances**. The window shown in Figure 4-26 will appear. Select the **2D Flow Options** tab. The user enters a total ramp up time in the **Initial Conditions Ramp Up Time** (hrs) field. Additionally, the user must enter what fraction of that time is used for ramping the 2D boundary conditions up from zero to their first value (i.e. a stage or a flow coming in). This is accomplished by enter the fraction in the column labeled **Boundary Condition Ramp Up Fraction (0 to 1).** The default value for the ramp up fraction is 0.1 (10 % of the ramp up time).

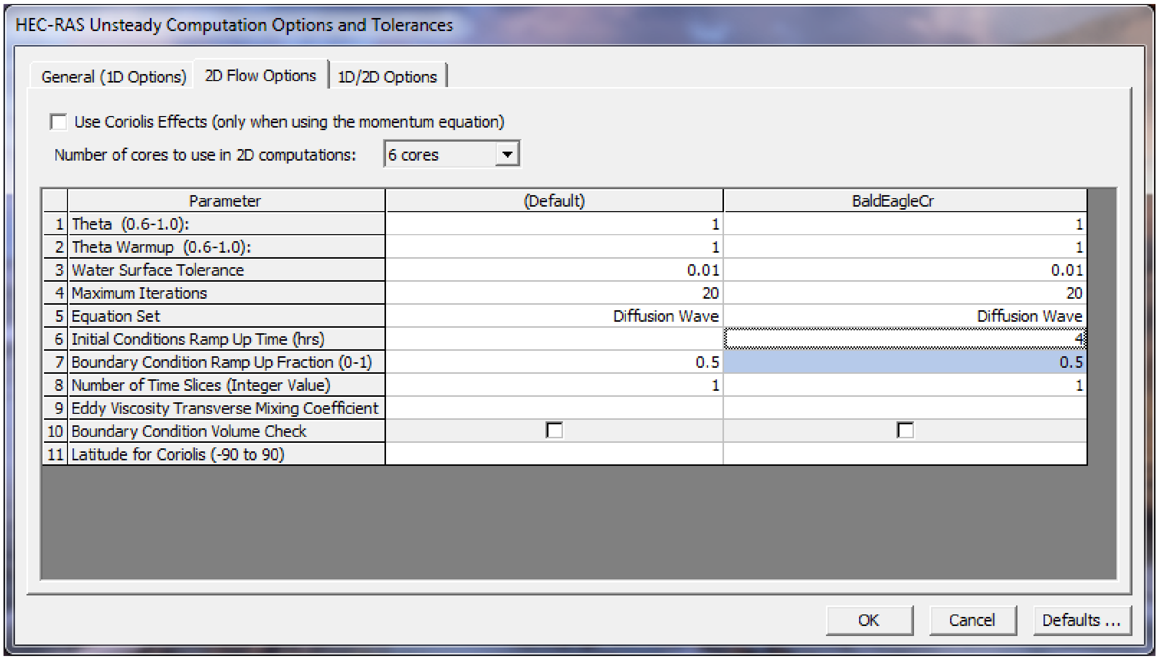


Figure 4-26. 2D flow area Computational Options.

Say, for instance, that a 2D area has an upstream flow boundary and a downstream stage boundary and the user has entered a two hour **Initial Conditions Ramp Up Time** with the **Boundary Fraction** at 0.5 (50%). Assume that the first flow on the flow boundary is 1000 cfs and the first stage of the downstream boundary has an elevation that corresponds to 10 ft of depth above the invert of the stage boundary (The invert is the lowest point along any part of the faces that make up the boundary). For the first hour of the initial conditions, the flow will increase linearly from 0 cfs up to 1000 cfs. The downstream stage boundary will transition from a depth of 0 ft up to a depth of 10 ft (and even though this is a "downstream" boundary, if the 2D area started out dry, then flow will initially come into the 2D area). For the second hour, the flow will held at 1000 cfs upstream and the depth at 10 ft downstream.

The initial conditions, if any, are computed separately for each 2D area (in a "standalone" mode). The flow and stages from any boundary conditions directly connected to the 2D area are taken into account. The flow and/or stage from any 1D river reach (that is directly connected) is taken into account to the extent possible. Flow from any lateral structure or storage area connectors is not taken into account during this part of the computations. (Flow crossing a hydraulic structure that is internal to the 2D area is computed.) If the user has entered a starting water surface for the given 2D area, then that water surface is used, before applying the 2D initial conditions ramp up time. Otherwise the 2D area starts out dry.

Warning: If you have 2D flow areas directly connected to 1D river reaches, you must use the 2D Initial Conditions Ramp Up option to get water all the way through the 2D flow area, such that the 1D/2D connection will be wet when the model starts up. If you do not do this, the 1D connection may go unstable right at the beginning of the simulation, because there is no water in the 2D area that it is connected to.

In addition to establishing the initial conditions within 1D river reaches and 2D flow areas, it is a good idea to turn on the overall model warm up option. By turning this option on, the program will hold all inflows constant, then solve the entire unsteady flow model together (1D and 2D), in order to get the unsteady flow equations and hydraulic connections to settle down to a stable initial condition before proceeding with the event simulation. The overall model warm up is turned on under the **General (1D Options)** tab from the Unsteady Flow window, **Options** menu, **Calculation Options and Tolerances.** The user turns this option on by entering a value into the field labeled **Number of warm up time steps (0 – 100,000)**. This is the number of time steps the user wants the model to run for the warm up period. There is also an option to put in a computation interval to use during the model warm up period (**Time step during warm up period (hrs)**). If this field is not set, then the program uses the default computation interval set by the user on the Unsteady Flow Analysis window. However, sometimes it can be very useful to use a smaller time step during the model warm up period in order to get the initial conditions established without going unstable

CHAPTER 5

Running a Model with 2D Flow Areas

Running a 2D or combined 1D/2D unsteady flow model in HEC-RAS is no different than running a standalone 1D unsteady flow model. The 2D unsteady computational module is built directly into the HEC-RAS unsteady flow computational engine - it is not a separate program. The 1D and the 2D computations are directly coupled on a time step by time step basis (there is also an iteration option for connections between 1D and 2D elements). This allows direct feedback from 1D to 2D elements and from 2D to 1D elements for each time step. This makes the linking of the 1D and 2D very accurate when it comes to sending flow through a breach (using a lateral structure), or any other type of hydraulic link between 1D and 2D elements. This direct feedback allows the software to more accurately calculate headwater, tailwater, flow, and any submergence that is occurring at a hydraulic structure on a time step by time step basis.

Shallow Water or Diffusion Wave Equations

As mentioned previously, HEC-RAS has the ability to perform two-dimensional unsteady flow routing with either the Shallow Water Equations (SWE) or the Diffusion Wave equations (DWE). HEC-RAS has three equation sets that can be used to solve for the flow moving over the computational mesh, the Diffusion Wave equations; the original Shallow Water equations (SWE-ELM, which stands for Shallow Water Equations, Eulerian-Lagrangian Method); and a new Shallow Water equations solution that is more momentum conservative (SWE-EM, which stands for Shallow Water Equations, Eulerian Method). The SWE also have options for modeling turbulence and Coriolis effects. See Chapter 2 of the Hydraulic reference manual for the theory on the development of these equations for use in HEC-RAS.

Within HEC-RAS the Diffusion Wave equations are set as the default, however, the user should always test if the Shallow Water equations are need for their specific application. A general approach is to use the Diffusion wave equations while developing the model and getting all the problems worked out (unless it is already known that the Full Saint Venant equations are required for the data set being modeled). Once the model is in good working order, then make a second HEC-RAS Plan and switch the equation set to the **SWE** option (SWE will generally require a smaller computation interval than the Diffusion wave method to run in a stable manner). Run this second plan and compare the two answers throughout the system. If there are significant differences between the two runs, the user should assume the SWE (Saint Venant equations) answer is more accurate, and proceed with that equation set for model calibration and other event simulations.

There are some obvious situations that the SWE should always be used. The following is a list of examples of situations in which the user should generally use the SWE based equations:

1. **Highly Dynamic Flood Waves**: If the modeler is performing a Dam breaching or flash flood analysis, the flood wave will rise and fall extremely quickly. The change in velocity (acceleration) both spatially and over time will be dramatic. The Diffusion wave equations do not include the local acceleration (changes in velocity with respect to time) and convective acceleration (changes in velocity with respect to distance) terms. These two terms are extremely important in order to model rapidly rising flood waves accurately.

2. **Abrupt Contractions and Expansions:** In areas where there are very abrupt contractions and expansions, the Full Momentum based equation set will more accurately capture the associated forces through the contraction and expansion of the fluid. This is also due to the inclusion of the convective acceleration term (Which is not included in the Diffusion Wave equations). In general the Full Momentum equation set will compute a higher water surface upstream of the contraction zone.

3. **Flat Sloping River Systems:** For rivers with very flat slopes (less than 1 ft/mi), gravity and friction may not be the dominant forces acting on a body of water as it travels from one point to another. The forces associated with changes in velocity with respect to time and distance, will play a much more significant role in how a floodwave moves and changes shape in a flat system. Because of this, users should use the Full Momentum equation set instead of then Diffusion Wave equations. As Diffusion wave does not include the acceleration terms with respect to time and space.

4. **Tidally Influenced Conditions:** If you are modeling a bay, estuary, or a river that is tidally influenced, then you have to use the full momentum based equation set. The propagation of waves (tide cycle) cannot be modeled with the diffusion wave equations. The ocean tides are dynamic waves that can propagate way up into a river system. There are many examples of where you can see this in gaged river stage data that is very far from the ocean, but still showing a tidal cycle in the stage hydrographs during low flow. Shown below in Figure 4-1 is the Stage data at Vancouver Washington and at the mouth of the Columbia River. As you can see, the tide cycle is evident in the stage data at Vancouver, even though this gage is 106 miles upstream from the mouth of Columbia bay.

5. **General Wave Propagation Modeling:** If the user needs to model wave propagation due to rapidly opening or closing of gated structures, or wave run-up on a wall or around an object (e.g. bridge piers, buildings, etc…), then the Full Momentum equation set is necessary for this type of modeling.

6. **Super Elevation around Bends:** If you have a tight bend in either a natural or designed channel, and you want to see if there is any super elevation of the water surface on the outside of the bend, this required the Full Momentum based equation set.

7. **Detailed Velocities and Water Surface Elevations at Structures:** If you are trying to compute a detailed velocity distribution at or near a hydraulic structure, The full momentum based equation set is more accurate. Some examples are, detailed water surface and velocity through a bridges and around the abutments and piers; Open channel flow through a gate or culvert; detailed velocities though and around a levee breach; detailed water surface and velocities around a building. Note: this type of modeling also requires very small grid cells and small computational time steps.

8. **Mixed Flow Regime:** In order to accurately model transitions from subcritical to supercritical flow, and from supercritical to subcritical flow (hydraulic jumps), the Full Momentum equation is more accurate.

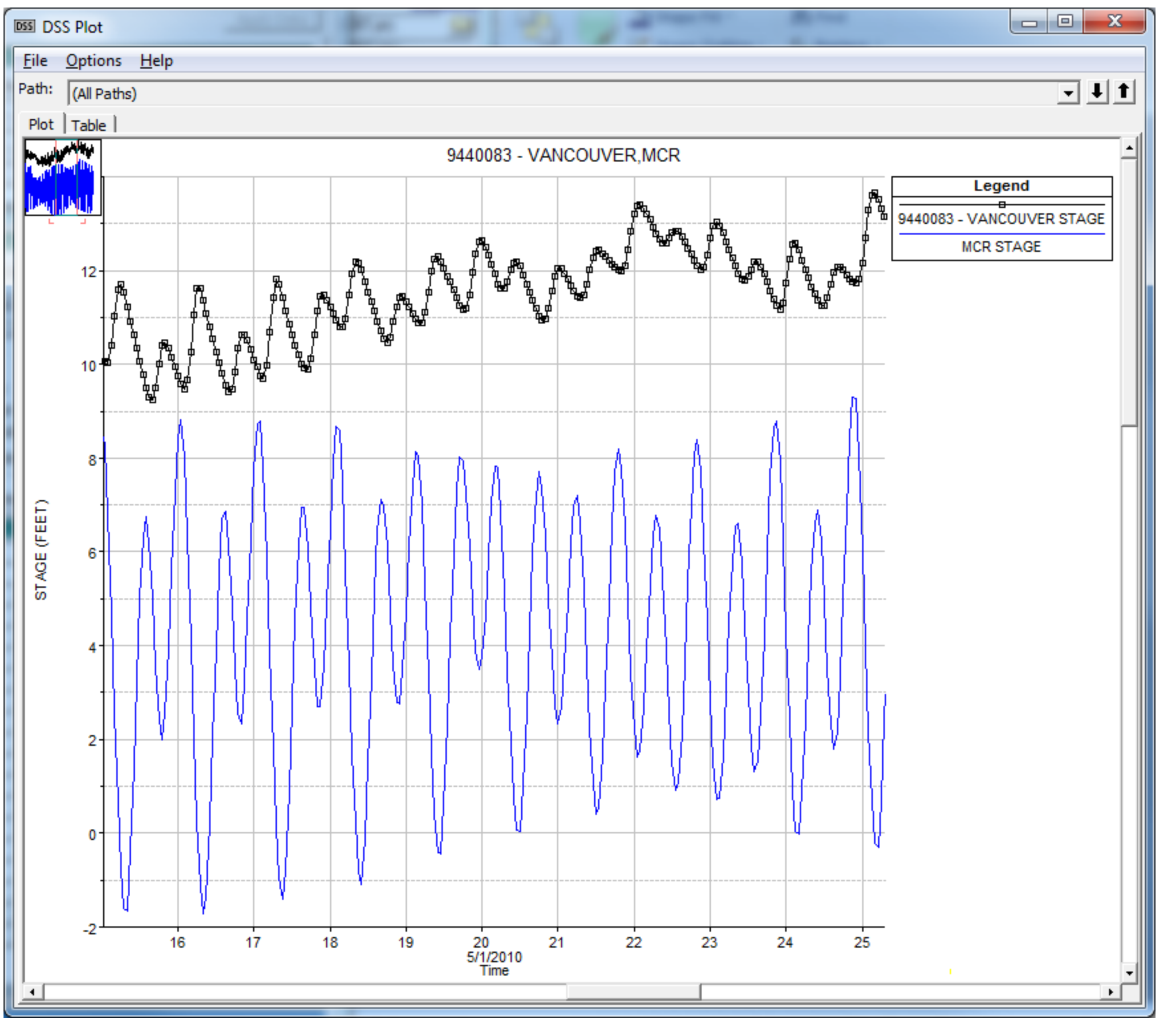


Figure 5-1. Stage Hydrograph at Vancouver Washington and Mouth of Columbia River.

Selecting an Appropriate Grid Size and Time Step

Assigning an appropriate mesh cell size (or sizes) and computational time step (ΔT) is very important to getting accurate answers with 2D flow areas. The first step is to develop a computational mesh that has cell sizes that are appropriate for modeling both the terrain as well as the water flowing over the terrain. Many 2D flow models use a single elevation for each cell and cell face (grid-based models). Finite Element models commonly (not always) use triangles (three elevations and a planar surface to represent each triangle) to represent the land surface, while each face has two elevations and a straight line between them. It is very important to understand the way the computational mesh is representing the underlying terrain in order to make a good decision on how many cells, and of what size, will be necessary to model the terrain and the event accurately.

HEC-RAS takes a very different approach than the two previously mentioned modeling techniques. Cells in HEC-RAS can have three, four, five… up to eight sides. Each cell is not a single elevation, but a detailed elevation volume/area relationship that represents the details of the underlying terrain. The HEC-RAS cell faces are detailed cross sections, which get processed into detailed elevation versus area, wetted perimeter, and roughness relationships. This approach allows the modeler to use larger cell sizes with HEC-RAS, and still accurately represent the underlying terrain. The key to making a good computational mesh in HEC-RAS is ensuring that the faces of the cells capture the high point of barriers to the flow. Additionally, one must consider changes in the water surface slope and changes in velocity. A single water surface elevation is computed in the center of each cell. So, the larger the cell size, the further apart are the computed values of the water surface, and thus the slope of the water surface is averaged over longer distances (in two dimensions). This is acceptable for some areas, but not appropriate for others. If the water surface slope and velocity vary rapidly, smaller cell sizes must be used in that area to capture the changing water surface and velocity. HEC-RAS allows the user to vary the cell size and shape at all locations in the model. So computational meshes can be developed with smaller cells where they need to be and larger cells where the terrain and water surface and velocity are not changing rapidly.

Some key factors for developing a good computational mesh with HEC-RAS are:

1. Make sure the cell sizes, shapes, and orientations adequately describe the terrain. Specifically, since the cell faces control the movement of water, there must be enough of them, oriented correctly to describe the key features of the terrain that will control water movement. This includes barriers to flow, such as roads, levees, floodwalls, and natural high ground areas, that will prevent flow from going from one area to another, until the water surface elevation is higher than the barrier. Additionally, the user may want to use a refinement region along the banks of the main channel. By using a refinement region along the bank of the main channel, the computational faces will be aligned along the high ground that separates the main channel flow from the overbank flow. This will ensure that no water leaves the main channel until it has

reached an elevation higher than the cell faces that are aligned with the channel banks. Additionally, the use of a refinement region allows the modeler to easily control the cell size within the main channel. Adding a breakline down the middle of the channel, on top of the refinement region, and using the near repeats option, will allow the user to align the cells with the direction of the flow.

2. The cell size must be adequate to describe changes in the water surface slope and changes in velocity. If the water surface slope and velocity does not change rapidly, larger cell sizes can be used to accurately compute the water surface elevation and slope. If the water surface slope and/or velocity changes rapidly, then smaller cell sizes need to be used to have enough computation points to describe the changing water surface and velocity, as well as compute the force/energy losses that are occurring in that area. While cell sizes (and shapes) can vary, transitioning from larger to smaller cell sizes should be done gradually to improve computational accuracy.

Once a good computational mesh is developed, then the user must pick an appropriate computational time step that works well with the mesh and the event being modeled. Picking an adequate time step is a function of the cell size and the velocity of the flow moving though those cells. HEC-RAS has three equation sets that can be used to solve for the flow moving over the computational mesh, the Diffusion Wave equations; the original Shallow Water equations (SWE-ELM, which stands for Shallow Water Equations, Eulerian-Lagrangian Method); and a new Shallow Water equations solution that is more momentum conservative (SWE-EM, which stands for Shallow Water Equations, Eulerian Method). The new Shallow Water equation solution methodology (SWE-EM) uses an explicit solution scheme for solving the equations. While this method is more momentum conservative, it requires smaller computational time steps (i.e. in general, the time step must be selected to ensure the Courant number is less than 1.0)

In general, the Diffusion Wave equations are more forgiving numerically than the SWE. This means that larger time steps can be used with the Diffusion Wave equations (than can be with the SWE), and still get numerically stable and accurate solutions. The following are guidelines for picking a computation interval for the Shallow Water equations and the Diffusion Wave equations:

<u>Shallow Water (SWE-ELM):</u>

$$C = \frac{V\Delta T}{\Delta X} \leq 1.0 \quad (with\, a \max\ C = 3.0)$$

Or

$$\Delta T \ \leq \ \frac{\Delta X}{V} \text{ (With C = 1.0)}$$

Where: C = Courant Number

V = Flood wave velocity (wave celerity) (ft/s)

ΔT = Computational time step (s)

ΔX = Average cell size (ft)

Shallow Water (SWE-EM):

$$C = \frac{V\Delta T}{\Delta X} \leq 1.0 \quad (with\ a \max\ C = 1.0)$$

Or

$$\Delta T \leq \frac{\Delta X}{V} \text{ (With C = 1.0)}$$

Diffusion Wave Equations:

$$C = \frac{V\Delta T}{\Delta X} \leq 2.0 \quad (with\ a \max\ C = 5.0)$$

Or

$$\Delta T \leq \frac{2\Delta X}{V} \text{ (With C = 1.0)}$$

Note: There are times when the diffusion wave method will need to be run with a time step that would produce a Courant number of 1.0 or less. Some examples are: very rapidly rising hydrographs and routing rapidly changing hydrographs over a completely dry channel.

Practical Time Step Selection: The way to use these equations is to find the area(s) with high velocities and rapid changes in water surface and velocity (with respect to space and time). Take the average cell size in that area for ΔX. Put in the maximum velocity in that area for V. User's will need to estimate a max velocity as a first guess for this calculation before running the model, then plot the max velocities from the run to make a better estimate. Users can also plot Courant Number directly from within HEC-RAS Mapper. Select a ΔT, such that the Courant Number (C) is equal to the suggested value (i.e. 1.0 for SWE). However, you may be able to get away with a Courant number as high as 3.0 for the SWE's and 5.0 for the Diffusion Wave equations, and still get stable and accurate results. If the event being modeled changes gradually with time and space, larger time steps can be used (i.e. Courant numbers approaching the maximum listed values). If the flood wave being modeled changes rapidly with respect to time and space, then you will need to use a time step closer to a Courant number of 1.0 (i.e. C = 1.0) for the high velocity zones. Also, if you have started the 2D area completely dry, then you will need to use a time step based on a Courant number of 1.0, in order to get a more accurate and stable wetting front.

Note: Users should always test the consistency of their computational mesh and selected time step. The consistency principle requires a reduction of both the space (grid) and time steps in order to guarantee convergence of a solution. If the grid is refined and the time-step is reduced simultaneously, the method will achieve convergence. The user should always test different cell sizes (ΔX) for the computational mesh, and also different computational time steps (ΔT) for each computational mesh. This will allow the user to see and understand how the cell size and computational time step will affect the results of your model. The selection of ΔX and ΔT is a balance between achieving good numerical accuracy while minimizing computational time.

Variable Time Step Capabilities

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).

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 5-2 below).

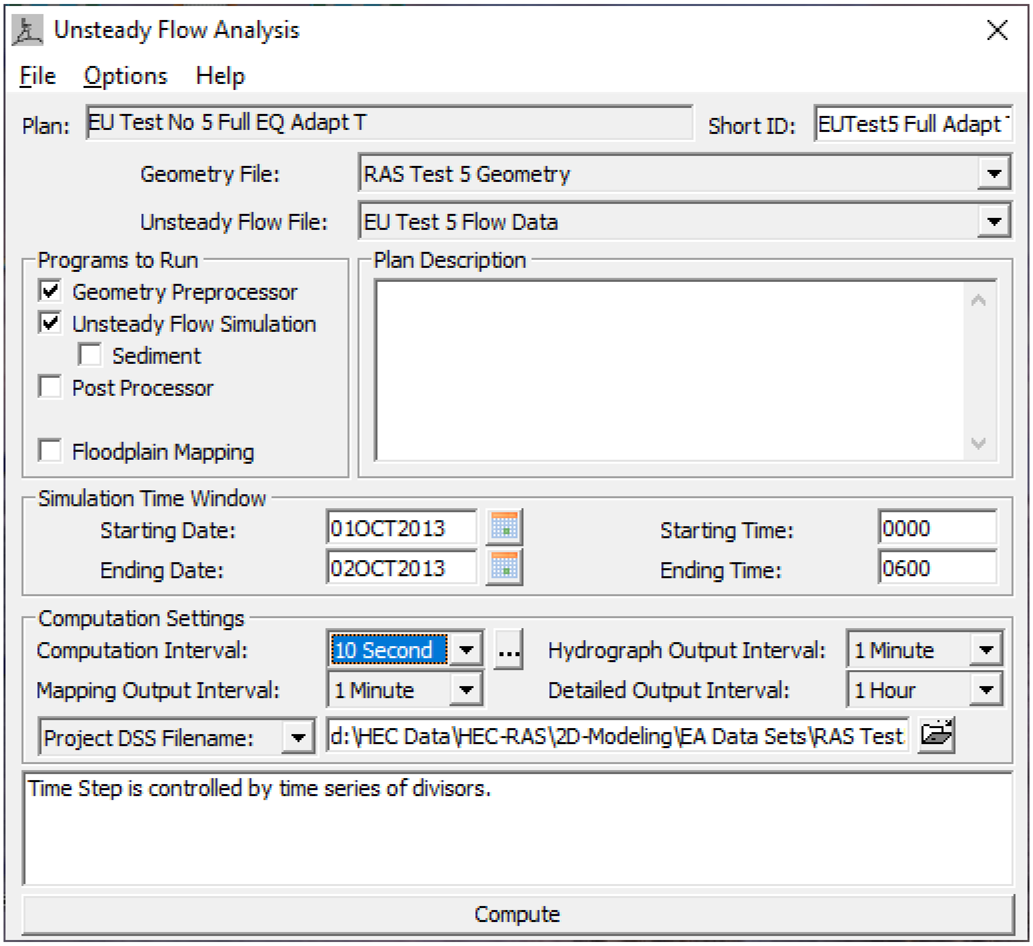


Figure 5-2. Button for Variable Time Step Option Shown on Unsteady Flow Analysis Window.

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 5-3). 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 5-3, 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.

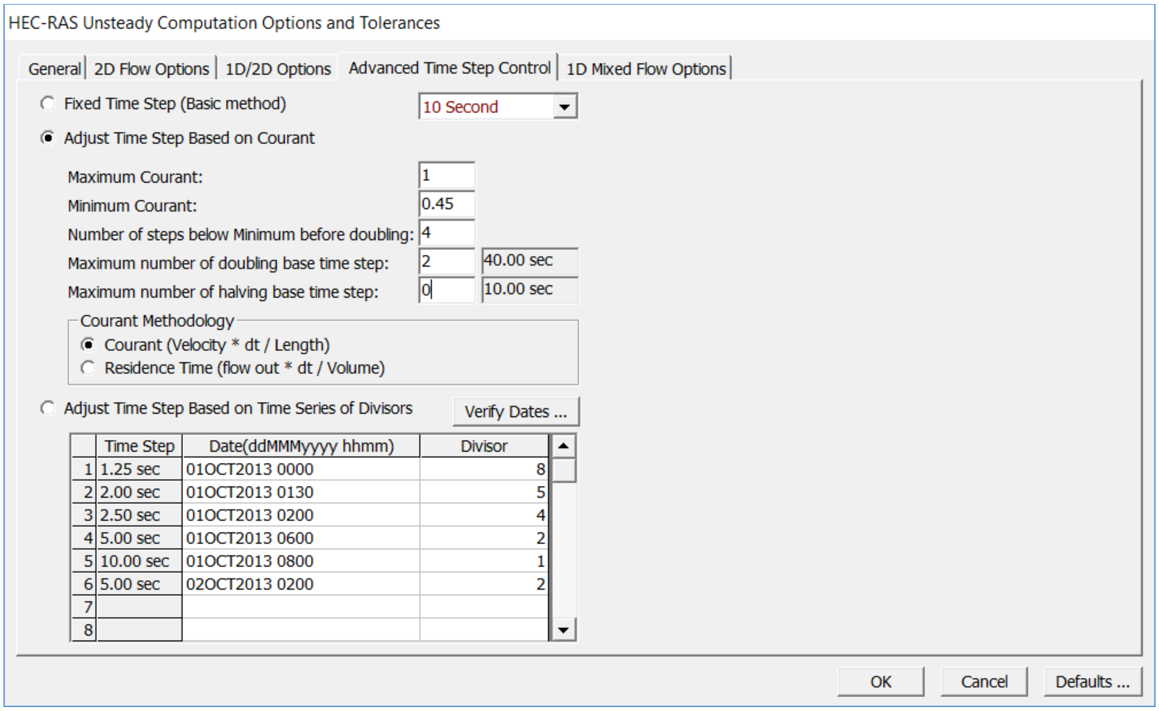


Figure 5-3. Variable Time Step Editor within the Computational Options and Tolerances.

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 5-2 and Figure 5-3, 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 5-3 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 software will just flip back and forth between halving and doubling the time steps. In the example shown in Figure 5-3, 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 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 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.

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 5-4).

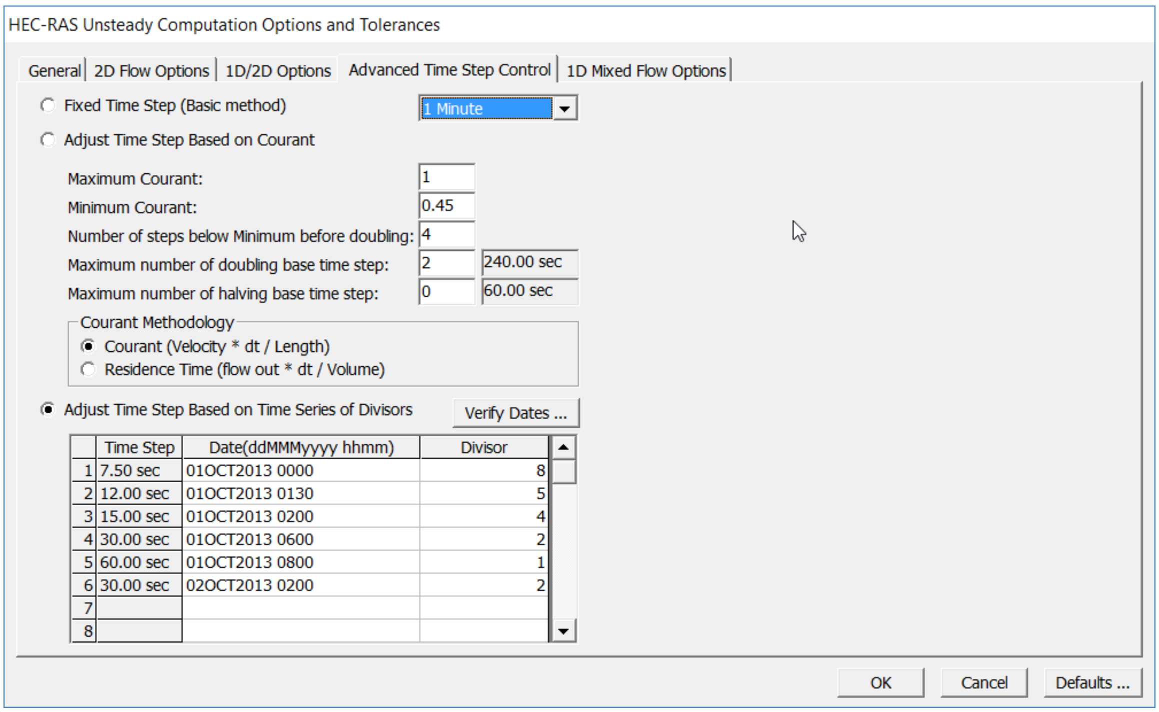


Figure 5-4. User Defined Variable Time Step Table.

As shown in Figure 5-4, 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 5-4, 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.

Performing the Computations

To run the model, open the **Unsteady Flow Analysis** window. Make a Plan by selecting the geometry that contains the combined 1D and 2D data, select an unsteady-flow file for the event to run, and give the Plan a Title and a Short ID. Set the following items: which Programs to Run; the Simulation Time Window; and all of the Computational Settings. Then press the **Compute** button to begin the run. (The window should look similar to Figure 5-5). If you have not previously run the 2D flow area pre-processor (from RAS Mapper) it will automatically be done first at the beginning of the unsteady flow process.

Under the **Programs to Run** area, there is a check box for **Floodplain Mapping.** If RAS Mapper has been set up correctly, by bringing in a terrain data set and associating that terrain with geometry files, then this option will work. If this option is turned on, after the program has completed the unsteady flow computations and the post processing, the last thing it will do is run a separate process called “ComputeFloodMaps.exe” in order to generate a depth grid (stored to disk) of the maximum inundation that occurred at all locations in the model. **This option is not required for flood mapping**. It is really only necessary if the user wants the process of computing a Stored Flood map to be done automatically at the end of the unsteady flow computations. The **Floodplain Mapping** option is off by default because the user can perform Dynamic Flood Mapping and create Static Flood Maps (Stored Depth Grids) from RAS Mapper, after the computations have been completed (this is the suggested work flow). The main purpose of this option, is to automate the process of computing an inundation map (Depth grid), for use in CWMS or HEC-WAT. In general, this option will most likely not be used when running HEC-RAS in standalone mode.

The post processor option provides additional (and detailed) output for 1D areas and it is only applicable for 1D data sets and mixed 1D/2D data sets. The 2D output is generated during the unsteady flow run.

Under the **Computational Settings** area, there is a new feature called **Mapping Output Interval**. This feature allows you to set a mapping interval that will be used in RAS Mapper for creating Dynamic Maps (computed on-the-fly in memory, and not stored to disk), as well as for performing animations of the flood maps. When the user selects a specific interval (e.g. 1-hr), a limited set of hydraulic output variables are written to a binary (HDF5) output file for all Cross Sections, Storage Areas, and 2D Flow areas. Additionally, the maximum and minimum values that occurred at all locations during the run are also written to this file. This file is then used by RAS Mapper to perform dynamic mapping of the results. There are some additional Mapping Output/HDF5 options under the **Options** menu. Select **Output Options…**, then select **HDF5 Write Parameters**.

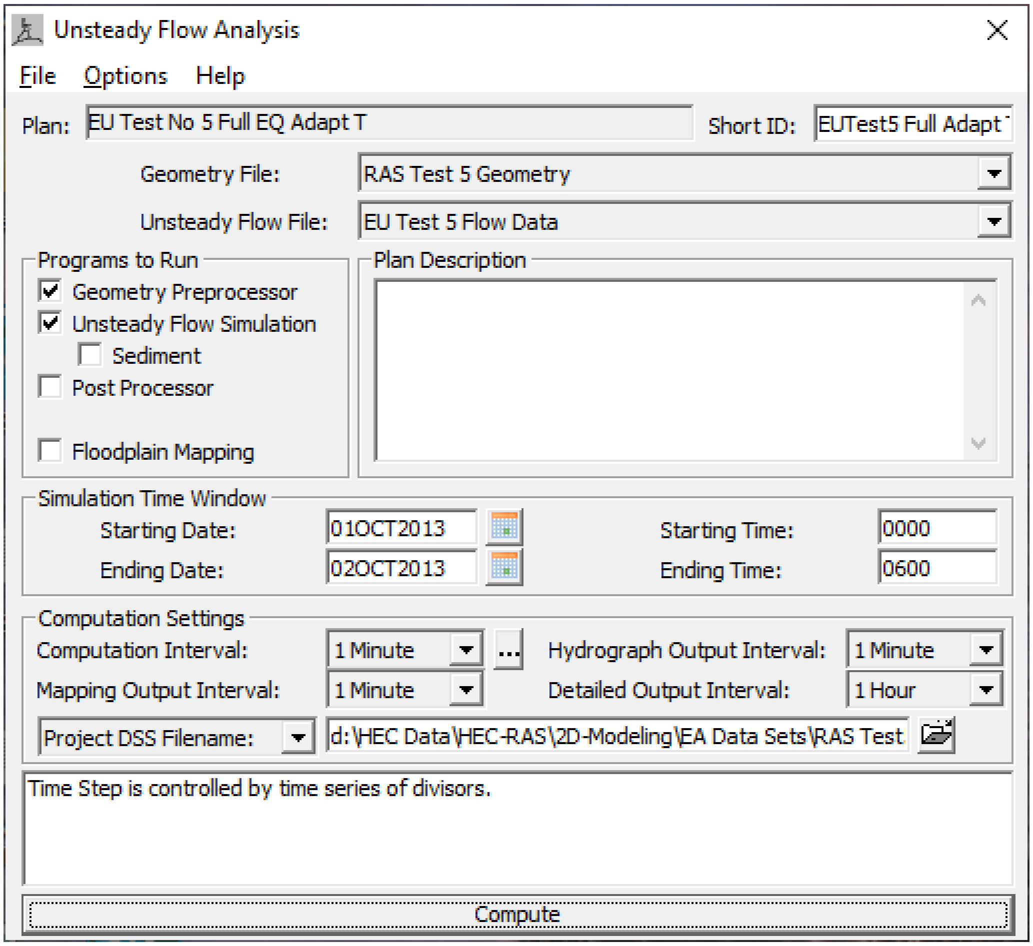


Figure 5-5. Unsteady flow Analysis Window with the new Floodplain Mapping feature.

Once the user presses the **Compute** button, the unsteady-flow computational engines will begin to run. This process consists of running the: 2D Geometry Pre-Processor (only if necessary); 1D Geometry Pre-Processor; Unsteady-Flow computations (combined 1D/2D); the 1D Post-Processor; and finally the Floodplain Mapping process (if it was turned on).

In general, the user does not need to run the Floodplain Mapping process, unless they are trying to automate the process of running an unsteady flow plan, mapping, and providing the results to another process (like HEC-FIA). The Dynamic Mapping within RAS Mapper will quickly become the main way to look at results. Once a good result is achieved, a static depth grid (stored to disk) can be created from within RAS Mapper in order to send to HEC-FIA (Flood Impact Analysis) or a GIS program for display and analysis.

Computation Progress, Numerical Stability, and Volume Accounting

As the Unsteady flow simulation runs, information is provided as to the progress of the run, how many iterations the 1D and 2D components are using to solve a particular time step, and numerical stability messages are written to the **Computational Messages** window. If any 1D or 2D element (cross section, storage area, or 2D cell) is not solved to within the pre-defined numerical tolerance during a time step, a message will be written to the message window. This message will provide information as to which cross section, storage area, or 2D cell had the greatest amount of numerical error for that time step. This information can be very useful for detecting numerical problems in the model. If the model goes to the maximum number of iterations for several time steps in a row, and the numerical errors are significant, then the user should investigate that area of the model, during the time that the model was having problems.

In addition to messages that come up in the Computation Messages window, a Computational Log file is written to the disk during the run. At a minimum, this file will always contain a volume accounting check for the simulation. The volume accounting is done for the entire 1D/2D model and all of its elements. There is also a separate volume accounting done for each 2D flow area. To view the computational log file, go to the **Options** menu on the Unsteady Flow Analysis window and select **View Computation Log File.** When this option is selected, the text file will appear in a Notepad window as shown in Figure 5-6.

BaldEagleDamBrk.bco08 - Notepad

File Edit Format View Help

```
################################################
#                                              #
#                                              #
#       1D and 2D Unsteady Flow Module         #
#                                              #
#                                              #
#        HEC-RAS 5.1 Alpha November  2019      #
#                                              #
#            31DEC19 at 14:37:12               #
#                                              #
################################################


Volume Accounting in Acre Feet

          External Boundary Flux of Water

US Inflow    Lat Hydro    DS Outflow    SA Hydro    Groundwater    2D Inflow    2D Outflow    Diversions
*********    *********    **********    ********    ***********    *********    **********    **********
278418.      113501.      435082.       45180.                     142274.

          River Reaches, Storage Areas, and 2D Areas

Start 1D Reach    Starting SA's    Starting 2D    Final 1D Reach    Final SA's    Final 2D Areas
**************    *************    ***********    **************    **********    **************
33017.                             56.72          161152.           2565.         13770.

          Error    Percent Error

          *****    *************
          123.5    0.02017




Volume Accounting for 2D Flow Area in Acre Feet

2D Area     Starting Vol    Ending Vol    Cum Inflow    Cum Outflow    Error     Percent Error
*******     ************    **********    **********    ***********    *****     ************


193         28.21           2496.         70323.        67856.         0.2861    0.000407
194         28.50           2150.         71951.        69830.         0.7645    0.001063
LockHaven                   9124.         109578.       100454.        0.03322   0.000030
```

Figure 5-6. Example Computational Log File with Volume Accounting Output.

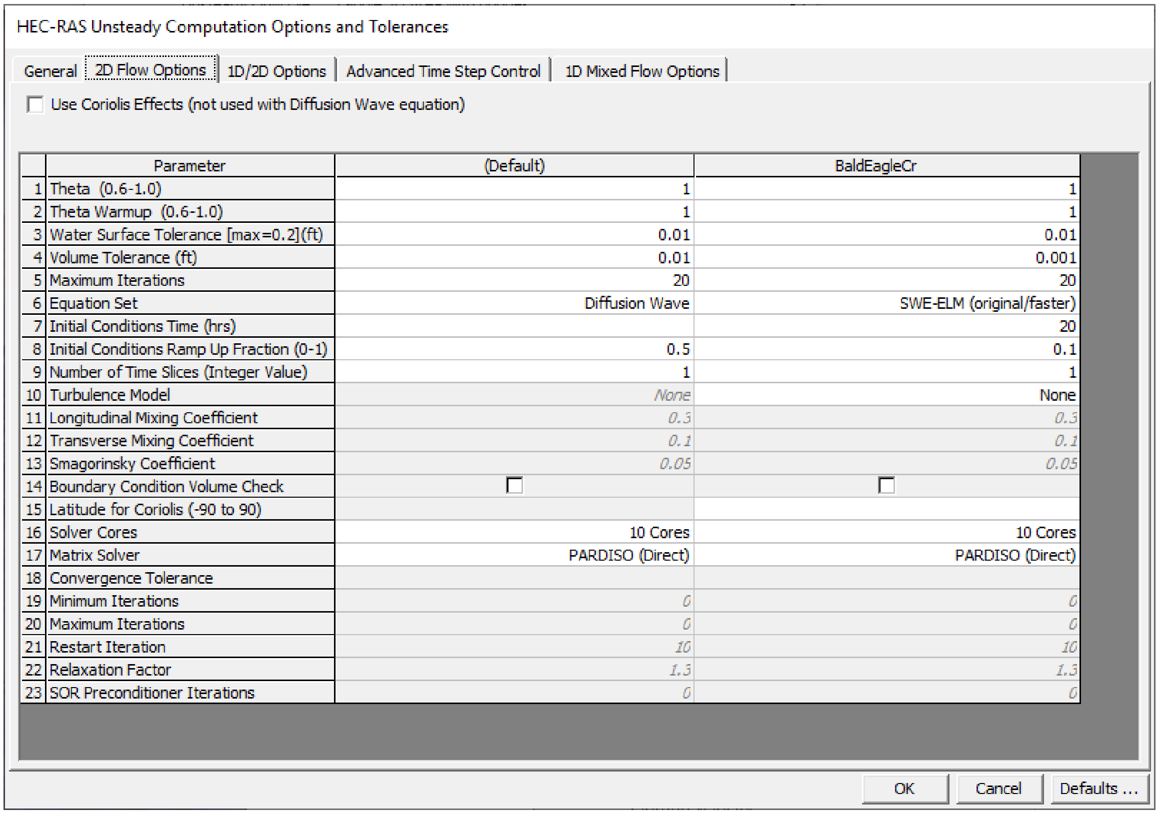
As shown in Figure 5-6, the volume accounting is shown for the entire model first, then for each individual 2D Flow Area. Volumes are shown in Acre-Feet for all elements. The example in Figure 5-6 shows that the overall model gained 123.5 acre-feet of water, which equated to a 0.02017 % volume error (very low). The separate 2D Flow Areas had flow gains ranging from 0.033 to 0.7646 acre-feet, which equated to volume errors ranging from 0.00003 to 0.001063 % volume error (extremely low).

2D Computation Options and Tolerances

2D Flow Options

Options for controlling the 2D computations during the run are available from the same editor that contains the 1D Computational Options and Settings. Select the **Options | Calculation Options and Tolerances** the menu item to invoke the window shown in Figure 5-7. This editor now has five tabs. The first Tab, labeled "**General,** is the original 1D Unsteady-flow calculation options. The second Tab, labeled **"2D Flow**

Options", contains the calculation options and tolerances for the 2D computational module. The third Tab, labeled "**1D/2D Options**" contains options for controlling iterations between 1D and 2D hydraulic connections. The fourth Tab "**Advanced Time Step Control**" is for using the variable time step capability. The fifth tab "**1D Mixed Flow Options**" is for controlling the mixed flow regime capabilities used with the original 1D Finite Difference solution scheme. This Option is not needed for the new 1D Finite Volume solution scheme.



	Parameter	(Default)	BaldEagleCr
1	Theta (0.6-1.0)	1	1
2	Theta Warmup (0.6-1.0)	1	1
3	Water Surface Tolerance [max=0.2](ft)	0.01	0.01
4	Volume Tolerance (ft)	0.01	0.001
5	Maximum Iterations	20	20
6	Equation Set	Diffusion Wave	SWE-ELM (original/faster)
7	Initial Conditions Time (hrs)		20
8	Initial Conditions Ramp Up Fraction (0-1)	0.5	0.1
9	Number of Time Slices (Integer Value)	1	1
10	Turbulence Model	None	None
11	Longitudinal Mixing Coefficient	0.3	0.3
12	Transverse Mixing Coefficient	0.1	0.1
13	Smagorinsky Coefficient	0.05	0.05
14	Boundary Condition Volume Check		
15	Latitude for Coriolis (-90 to 90)		
16	Solver Cores	10 Cores	10 Cores
17	Matrix Solver	PARDISO (Direct)	PARDISO (Direct)
18	Convergence Tolerance		
19	Minimum Iterations	0	0
20	Maximum Iterations	0	0
21	Restart Iteration	10	10
22	Relaxation Factor	1.3	1.3
23	SOR Preconditioner Iterations	0	0

Figure 5-7. 2D Flow Area Calculation Options and Tolerances.

As shown in Figure 5-7, there are several computational options and tolerances that can be set for the 2D module. These Options are discussed below.

Use Coriolis Effects: Only used in the SWE are turned on (Full Momentum)

This option allows the user to turn on the effects of the Earth's rotation on the solution (Coriolis Effect). When this option is turned on, the user must enter the latitude of the center of the 2D flow area in degrees (this is the field labeled **Latitude for Coriolis** in the table). A latitude with a value greater than zero is considered to be in the northern hemisphere, and a value less than zero is considered in the southern hemisphere.

Theta (0.6 – 1.0): 1.0 (default)

This is the implicit weighting factor that is used to weight spatial derivatives between the current solution time line and the previously computed time line. Theta of 1.0 (Default), uses only the currently solved time line for the spatial derivatives. This provides for the most stable solution, but possibly at the loss of some accuracy. Theta of 0.6, provides for the most accurate solution of the equations, but tends to be less stable. In general it has been found that in most applications of real world flood runoff events in rivers, Theta of 1.0, will give about the same answers as Theta of 0.6. However, this should be tested for each model due to site specific geometry and flood propagation, in which it may make a difference in the results.

Theta Warm-up (0.6 – 1.0): 1.0 (default)

This is the value of Theta (see description above) that is used during the model warmup and ramp up periods. This value of Theta is only used if the user has turned on the unsteady flow warm-up option, or the Boundary Condition Ramp up Option for 2D areas.

W**ater Surface Tolerance (ft):** 0.01 (default)

This is the 2D water surface solution tolerance for the iteration scheme. If the solution of the equations gives a numerical answer that has less numerical error than the set tolerance, then the solver is done with that time step. If the maximum error is greater than the set tolerance, then the program will iterate to get a better answer. The program will only iterate up to the maximum number of iterations set by the user. The default is set to 0.01 ft based on experience in using the model for a range of applications.

Volume Tolerance (ft): 0.01 (default)

This is the 2D water volume solution tolerance for the iteration scheme. The volume error is converted to feet of error, by taking the currently solved for water surface elevation into the elevation-volume curve for the cell, then calculating the change in water surface elevation based on the current volume error at that point on the curve. If the solution of the equations gives a numerical answer that has less volume error than the set tolerance (in terms of ft), then the solver is done with that time step. If the maximum error is greater than the set tolerance, then the program will iterate to get a better answer. The program will only iterate up to the maximum number of iterations set by the user. The default is set to 0.01 ft based on experience in using the model for a range of applications.

Maximum Number of Iterations (0 – 40): 20 (Default)

This is the maximum number of iterations that the solver will use while attempting to solve the equations (in order to get an answer that has a numerical error less than the user specified tolerance at all locations in the 2D computational mesh domain). The default is set to 20. However, the user can change it from 0 to 40. It is not recommended to change this unless you are sure that changing the value will either improve the chances that the model will converge (I.e. increasing the value) or speed up the computations without causing any significant errors.

Equation Set: Diffusion Wave (Default); SWE-ELM (original/faster); and SWE-EM (stricter momentum).

The HEC-RAS two-dimensional computational module has the option of running the following equation sets: **2D Diffusion Wave** equations; Shallow Water Equations (SWE-ELM) with a Eulerian-Langrangian approach to solving for advection; or a new Shallow Water Equation solver (SWE-EM), that uses an Eulerian approach for advection. The new SWE equation solution method is more momentum conservative but may require smaller time steps and produce longer run times. The default is the 2D Diffusion Wave equation set. In general, many flood applications will work fine with the 2D Diffusion Wave equations. The Diffusion Wave equation set will run faster and is inherently more stable. However, there are definitely applications where the 2D SWE should be used for greater accuracy. The good news is that it easy to try it both ways and compare the answers. It is simply a matter of selecting the equation set you want, and then running it. Create a second Plan file, use the other equation set, run it, and compare it to the first Plan for your application.

The new SWE solver (SWE-EM) is an explicit solution scheme that is based on a more conservative form of the momentum equation. This solver requires time steps to be selected to ensure the Courant number will be less than 1.0, in general (not always). This solver produces les numerical diffusion than the original SWE solver. However, in general, this new solver is only needed when user's are interested in taking a very close look at changes in water surfaces and velocities at and around hydraulic structures, piers/abutments, and tight contractions and expansions. The original SWE solver is more than adequate for most problems requiring the full momentum equation based solution scheme.

Initial Conditions Ramp up Time (hrs): Default is Blank (not used)

This option can be used to **Ramp up** the water surface from a dry condition to a wet condition within a 2D area (or from a flat water surface if an initial water surface elevation was entered). When external boundary conditions, such as flow and stage hydrographs (or 1D reaches), are connected to a 2D area, the first value of the connected flow or stage may be too high (i.e. a very large flow or a stage much higher than the cell elevation it is attached to). If the model were to start this way, such a high discontinuity could cause a model instability. This option allows the user to specify a time (in hours) to run the computations for the 2D flow area, while slowly transitioning the flow boundaries from zero to their initial value, and the stage boundaries from a dry elevation up to their initial wet elevation. The user specifies the total **Initial Conditions Ramp up Time** in this field (10 hours, for example). The user must also specify a fraction of this time for Ramping up the boundary conditions. A value of 0.1 means that 10% of the Initial Conditions time will be used to Ramp Up the boundary conditions to their initial values, the remaining time will be used to hold the boundary conditions constant, but allow the flow to propagate through the 2D flow area, thus giving it enough time to stabilize to a good initial condition throughout the entire 2D flow area. The Ramp up time for the boundary conditions is entered in the next row, which is labeled **Boundary Condition Ramp up Fraction.**

Boundary Condition Ramp up Fraction (0 to 1.0): 0.1 (10%) Default value

This field goes along with the previous field **Initial Conditions Ramp up Time**. This field is used to enter the fraction of the Initial Conditions Ramp up Time that will be used to ramp up the 2D flow area boundary conditions from zero or dry, to their initial flow or stage. Users can enter a value between 0.0 and 1.0, representing the decimal fraction of the Initial Conditions Ramp up Time.

Number of Time Slices (Integer Value): 1 (Default)

This option allows the user to set a computational time step for a 2D flow area that is a fraction of the overall unsteady flow computation interval. For example, if the user has set the unsteady flow overall computation interval to 10 minutes, then setting a value of 5 in this field (for a specific 2D area) means that the computation interval for that 2D area will be 1/5 of the overall computation interval, which for this example would be 2 minutes (e.g. 10/5 = 2). Different values can be set for each 2D flow area. The default is 1, which means that 2D flow area is using the same computational time step as the overall unsteady flow solution (computation Interval is entered by the user on the unsteady flow analysis window).

Turbulent Mixing Coefficients

The modeler has the option to include the effects of turbulent mixing and dispersion in the 2D flow field. HEC-RAS has two different formulations for turbulence modeling. Our original formulation was a non-conservative formulation. We now have a fully conservative formulation users can choose from also.

Turbulent mixing is the transfer of momentum due to the chaotic motion of the fluid. Dispersive mixing is the transfer of momentum due to subgrid variations in the time-averaged velocity field, both vertically and horizontally. The sum of horizontal molecular and turbulent mixing and dispersion are referred to simply as mixing here and are simulated as a Fickian diffusive process with a diffusion coefficient represented by the eddy viscosity. The default in HEC-RAS is to have all of the mixing coefficients empty, meaning turbulence is turned off. The numerical scheme in HEC-RAS provides some numerical diffusion. The larger the computational cell size and time step the larger the numerical diffusion. Therefore, when utilizing relatively fine computational cells with small time steps, it may be necessary to turn on mixing in order to obtain accurate results. To turn on horizontal mixing, enter a value for at least one of the mixing coefficients. These are the **Longitudinal Mixing Coefficient**, the **Transverse Mixing Coefficient**, and the **Smagorinsky Coefficient**. In previous versions of HEC-RAS, the longitudinal and transverse mixing coefficients were assumed to the equal. However, in this version these are specified separately, and an additional parameter, the **Smagorinsky Coefficient** has been added. Simple guidance is provided for initial values for these parameters. Nonetheless, these coefficients should be calibrated using spatially distributed velocity measurements. For the details on the Turbulence modeling approaches in HEC-RAS, please review the 2D modeling section of Chapter 2 in the HEC-RAS Hydraulic Reference Manual.

Turbulence Model: None (Default), Conservative, and Non-Conservative (original).

The Turbulence model choices are None (no turbulence modeling will be performed), Conservative (a conservative formulation that ensures little to no momentum loss when turbulence is turned on), and a Non-Conservative formulation (this formulation has some numerical diffusion, however it is very small when the cell size is small and the time step is small).

Longitudinal Mixing Coefficient: Default is Blank (not used)

In general, the turbulence intensity and dispersion are not the same in all. The **Longitudinal Mixing Coefficient** is utilized to compute the contribution of the eddy viscosity from the turbulence and dispersion in the longitudinal direction (i.e. the direction of flow). In practice the coefficient orders of magnitude depending on the flow. Below are some values for the **Longitudinal Mixing Coefficient** (D_L) that have been found to be appropriate under certain conditions (Table 5-1).

Table 5-1. Longitudinal Coefficients.

$\boldsymbol{D_L}$	**Mixing Intensity**	**Geometry and surface**
$0.1\ to\ 0.3$	Little longitudinal mixing	Straight channel Smooth surface
$0.3\ to\ 1$	Moderate longitudinal mixing	Gentle meanders Moderate surface irregularities
$1\ to\ 3$	Strong longitudinal mixing	Strong meanders Rough surface

Transverse Mixing Coefficient: Default is Blank (not used)

The **Transverse Mixing Coefficient** is utilized to compute the contribution of the eddy viscosity from the turbulence and dispersion in the transverse direction (i.e. perpendicular to the flow direction). In practice the coefficient orders of magnitude depending on the flow. Below are some values for the **Transverse Mixing Coefficient** (D_T) that have been found to be appropriate under certain conditions (Table 5-2).

Table 5-2. Transverse Mixing Coefficients.

D_T	Mixing Intensity	Geometry and surface
$0.05\ to\ 0.1$	Little transversal mixing	Straight channel Smooth surface
$0.1\ to\ 0.3$	Moderate transversal mixing	Gentle meanders Moderate surface irregularities
$0.3\ to\ 1$	Strong transversal mixing	Strong meanders Rough surface

Smagorinsky Coefficient: Default is Blank (not used)

The Smagorinksy component of the eddy viscosity represents the turbulence intensity produced by flow shear and assumes that the energy production and dissipation are equal at subgrid scales. The component is expressed as a function of length scale defined by the local grid resolution, the flow strain (magnitude of the depth-averaged current velocity gradients), and the dimensionless **Smagorinsky Coefficient**. The Smagorinksy eddy viscosity component requires computing four current velocity gradients for the flow strain and therefore it is relatively computationally expensive and should only be used (and is only necessary) when simulating high-resolution flows. This Smagorinksy eddy viscosity is important in regions close to solid boundaries or obstacles in the flow. The **Smagorinsky Coefficient** value is typically in the range of **0.05 to 0.2**. However, it is best to calibrate this and other mixing parameters using spatially distributed current velocity measurements. Since the eddy viscosity is computed as the sum of the various components, their values must be calibrated together.

Boundary Condition Volume Check: When this option is turned on, the software will check 2D to 1D boundary connections to ensure that there is enough water in a 2D cell over the time step to actually satisfy the computed flow exchange. If there is not, the computed flow exchange will be reduced for the first guess of the flow rate for that time step.

Latitude for Coriolis (-90 to 90): If the user turns on the "Use Coriolis Effect" option above, then a latitude for the centroid of the project should be entered in this field.

Solver Cores: The number of CPU cores to use in solving the 2D Flow Area solution. All Available (Default)

The HEC-RAS 2D computational module was developed from the ground up with parallel processing in mind. The HEC-RAS 2D computations will use as many CPU cores as there are available on your machine (which is the default mode for running). However, HEC-RAS provides the option to set the number of cores to use for the 2D computations. In general, it is recommended to use the default of "All Available". However, the user may want to experiment with this for a specific data set to see if it will either speed up or slow down computations based on a specific number of cores. The ideal number of cores for a given problem is size (number of cells) and shape dependent (shape of the 2D Flow Area). As more cores are used, the problem is split into smaller pieces, but there is overhead in the communications between the pieces. So, it is not necessarily true that a given problem will always run faster with more cores. Smaller data sets (2D areas with fewer cells) may actually run faster with fewer cores. Large data sets (2D Areas with lots of cells, i.e. > 100,000 cells) will almost always run faster with more cores, so use all that is available.

Shown below in Table 5-3 and Figure 5-8 are the results of testing a few data sets by running them with different numbers of Cores. Each model was run several times with the number of cores set to: 1, 2, 4, 8, and 16 (Figure 4-5). As you can see four of the data sets had speed improvements up to 8 cores, but actually ran slower with 16 cores. These are smaller data sets ranging from 10,000 to 80,000 cells. However, one data set had speed improvements all the way up to 16 cores. This was the largest data set, with 250,000 cells. Further runs were done to find the optimal number of cores for each data set, these results are shown in Table 5-3.

Table 5-3. Optimal No. of CPU Cores vs Number of 2D Cells

Data Set Name	**Time Window**	**Time Step**	**Number of Cells**	**Optimal No. of Cores**	**Computation Time**
Saint Paul 1D/2D	16 days	30 s	2,251	6	48 s
EU Test No 5 2D	1 day 6 hrs	10 s	7,460	8	36 s
Ohio/Miss 1D/2D	49 days	5 min	23,087	8	3 min 37 s
EU Test No 4 2D	5 hrs	20 s	80,000	10	41s
400 sq mi Watershed 2D	2 days	2 min	250,000	16	23 min 23 s

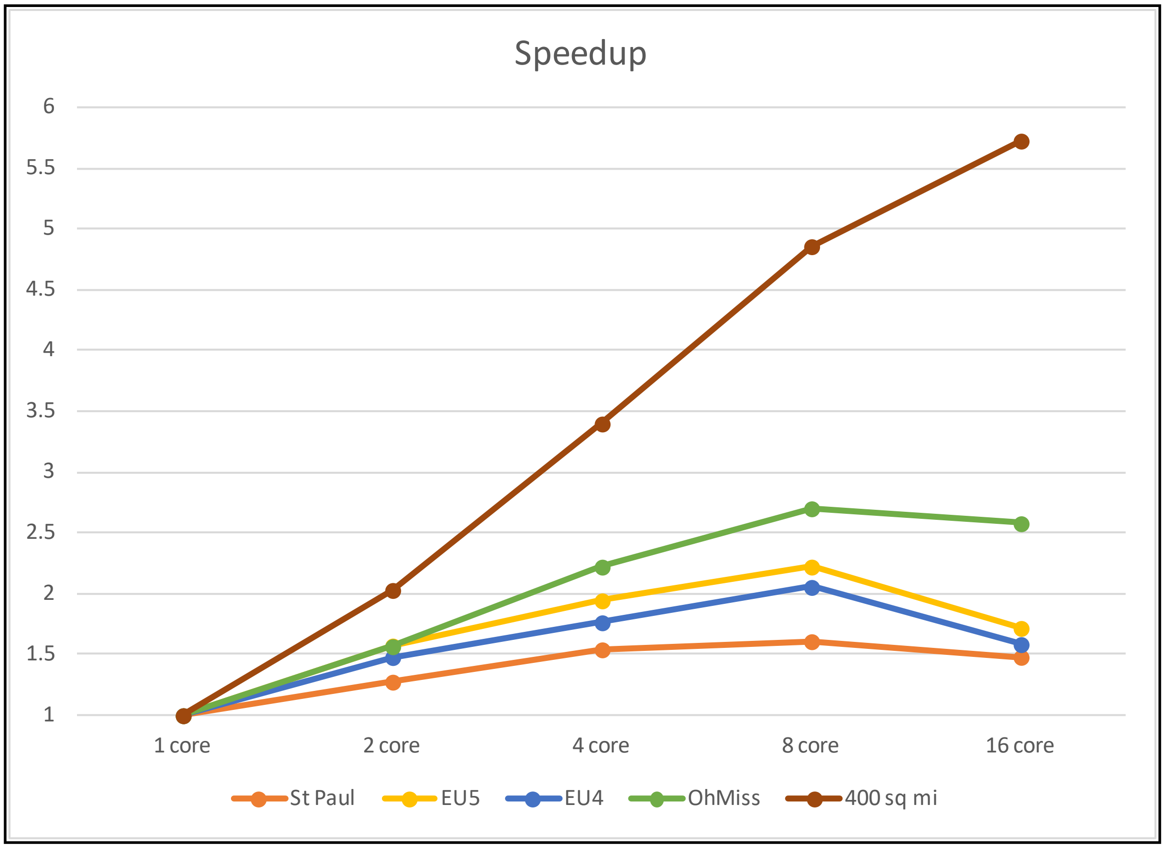


Figure 5-8. Number of processor cores vs. computational speed.

Matrix Solver: PARDISO (Default)

This option allows the user to select one of several sparse matrix solvers. There are two types of solvers available: direct and iterative. PARDISO is the only direct solver available. Direct solvers compute the final solution within a finite number of steps and do not require an initial guess. Since direct solvers produce very accurate solutions, they are very robust. For this reason, PARDISO is default matrix solver in HEC-RAS. One other advantage of the PARDISO solver is that it does not require any user-specified parameters. The second type of solvers in HEC-RAS are iterative solvers. Iterative solvers generally require less memory because unlike with direct solvers, the structure of the matrix does not change during the iteration process. In addition, iterative solvers utilize matrix-vector multiplications which can be efficiently parallelized. The main drawback of iterative solvers is that the rate of convergence depends greatly on the condition number of the coefficient matrix. For poorly conditioned matrices, the iterative solver may not converge at all. Therefore, the efficiency of iterative solvers greatly depends on the size and condition number of the coefficient matrix.

However, HEC-RAS 6.0 and newer also have the option to select one of two iterative solvers. The iterative solvers can be faster under certain conditions. The iterative solvers available are: SOR (Successive Over-Relaxation), and FGMRES-SOR (FGMRES Flexible Generalized Minimal Residual with SOR preconditioner). The SOR method is used both as an iterative solver and as a preconditioner for FGMRES. The SOR solver works well for small (~5000-10000 cells) to medium (~30000-50000 cells) size 2D areas. Generally, the larger the mesh the more iterations are required to for convergence.

Convergence Tolerance: None (Optional)

This option allows the user specify the convergence tolerance for the iterative solver. The tolerance is compared to the root-mean-squared of the normalized residuals. If not specified the tolerance is set to $1x10^{-4}$ times the water surface tolerance. Decreasing the tolerance will increase the number of iterative solver iterations and decrease the speed of the simulation and vice-versa. If the tolerance is too small, it is possible the iterative solver will take large amount of iterations to reach the tolerance or never reach it at all. Conversely, if the tolerance is set too large, the solution errors will produce water volume errors. Therefore, the user must be careful when setting this parameter.

Minimum Number of Iterations: None (Optional)

This option allows the user specify the minimum number of iterations. The iterative solver will continue to iterate even if the convergence criteria is satisfied until the minimum number of iterations. The minimum number of iterations avoids solution creep or divergence in cases where the solution is changing slowly or the convergence tolerance is set too high. The default minimum number of iterations is computed internally based on the number of cells and the matrix solver. The SOR solver generally requires for iterations than the FGMRES-SOR solver.

Maximum Number of Iterations: None (Optional)

This option allows the user specify the maximum number of iterations. The maximum number of iterations keeps the solver from iterating too many times past the point of diminishing returns. Generally iterative solvers will reach a point at which the improvement in the solution for each iteration is relatively small and the computational cost of each iteration does not outweigh the benefit of continuing to iterate. The default maximum number of iterations is computed internally based on the number of cells and the matrix solver. The SOR solver generally requires for iterations than the FGMRES-SOR solver.

Relaxation Coefficient: 1.3 (Default)

This option allows the user specify the relaxation coefficient for the SOR solver. The optimal relaxation value of the relaxation coefficient is generally between 1.1 and 1.5. However, if the value is too large the solver may diverge. Therefore, it is best to utilize a conservative value which will be relatively fast and not have divergence problems.

Restart Iterations: 10 (Default)

This option allows the user specify the restart iterations used in the FGMRES solver. Because the amount of storage and computational work required by FGMRES increases with each iteration, the method is typically restarted. The group of iterations between successive restarts is referred to as a cycle. When a cycle is completed the results approximate solution is used as an initial guess for a new cycle of iterations. Generally the restart iterations is between 8-20.

SOR Preconditioner Iterations: 10 (Default)

The SOR method is used both as a solver and as a preconditioner for the FGMRES solver in HEC-RAS. When SOR is used as a preconditioner, the SOR method is applied for a fixed number of iterations which is specified as the **SOR Preconditioner Iterations**. This eliminates the need specify separate convergence criteria for the auxiliary system of equations which the SOR method is applied to. It also eliminates the need to compute convergence parameters during the SOR iterations. The optimum number of iterations will be problem specific. However in general the larger the mesh the more iterations that will be necessary.

1D/2D Options Tab.

There are also some options for Controlling 1D/2D Iterations, which can be used to improve the computations of flow passing from a 1D element (reach or storage area) to a 2D Flow Area. By default this option is turned off, and most 1D to 2D connections will not need iterations. However, when the1D/2D hydraulic conditions become highly submerged, or there are flow reversals, or tidally influence stages/flows, then iterating between the 1D solution and 2D solution may be necessary to get an accurate and stable solution. To turn on the 1D/2D iterations option, select the "**1D/2D Options"** tab. Then set the **Maximum iterations between 1D and 2D**, as well as tolerances for controlling the convergence criteria. Iteration can be set from 0 to 20, with zero meaning that it does not do any extra iterations (this is the default). In general, only use this option if you are having a stability problem at a 1D/2D hydraulic connection. Set the number of 1D/2D iterations to as low as possible in order to get a stable answer between a 1D and 2D connection that is having stability problems. The number of 1D/2D iterations will cause the entire solution to be done multiple times for each time step in order to get the desired convergence. **This could dramatically lengthen run times.** If the user turns this option on, it is suggested to start with a low value, like 3 or 4. If the stability problem still exists with that number of iterations, then increase the value until a stable solution is achieved.

The convergence criteria for 1D/2D iterations consists of a **Water Surface Tolerance**, **Flow Tolerance (%)**, and a **Minimum Flow Tolerance**. The water surface tolerance is currently only used when an upstream 1D reach is connected to a downstream 2D Flow Area. In this situation, the 1D region is computed, then the 2D region. The assumed water surface elevation at the boundary is re-evaluated. If the water surface has changed more than the **Water Surface Tolerance**, then the program will iterate. When the water surface elevation at the boundary has change less than the tolerance, the solution stops iterating and moves on to the next time step.

The **Flow Tolerance (%)** is utilized for the following 1D/2D connections: Lateral Structure; SA/2D Hydraulic Connection (SA to 2D, or 2D to 2D); and 2D Flow Area to 1D Reach connection. The default value for the Flow Tolerance (%) is 0.1 %. If 1D/2D iterations are turned on, then the flow between these types of 1D/2D connections gets recomputed after each trial to see if it has changed more than the user defined **Flow Tolerance** (%). If it has changed more than the flow tolerance, then the program iterates. A companion tolerance to the **Flow Tolerance**, is the **Minimum Flow Tolerance (cfs)**. The purpose of this tolerance is to prevent the program from iterating when the flow passed between a 1D and 2D element is very small, and not significant to the solution. For example, there may be a connection from a 1D reach to a 2D Flow Area via a Lateral Structure, in which the flow under certain conditions is very low, so the actual change in the flow from one iteration to the next could be very small (but the percent error is very high). Such a small flow may have no significance to the solution, so iterating the entire solution to improve this small flow between the 1D and 2D elements makes no sense, and may be just unnecessarily increasing the computational time. In general it is a good idea to set a minimum flow when turning on 1D/2D iterations. The default value is 1 cfs, however, this is most likely model specific.

New 1D Computational Options

A few new 1D only computational options have been added, and one existing option has been removed. If the user selects the Tab labeled **General (1D Options)** on the **Computational Options and Tolerances** window, they will see the following new 1D options:

1. **1D Finite Volume Solver**. HEC has developed a new 1D Finite Volume solution algorithm for solving the 1D Shallow Water equations. The current 1D Finite Difference scheme has trouble in the following situations:

- Can’t handle starting or going dry in a XS
- Low flow model stability issues with irregular XS data.
- Extremely Rapidly rising hydrographs
- Mixed flow regime (i.e. flow transitions)
- Stream Junctions do not transfer momentum

The new 1D Finite Volume algorithm has the following positive attributes:

- Can start with channels completely dry, or they can go dry during a simulation (wetting/drying)
- Very stable for low flow modeling
- Can handle extremely rapidly rising hydrographs without going unstable
- Handles subcritical to supercritical flow, and hydraulic jumps better – No special option to turn on.

- Junction analysis is performed as a single 2D cell when connecting 1D reaches (continuity and momentum is conserved through the junction)

Additionally, the new 1D Finite Volume solution scheme breaks up the cross section into separate cells for the left overbank, main channel, and the right overbank. The current finite difference scheme combines the properties of the left and tight overbank into a single flow area called the floodplain. Additionally, the hydraulic properties of the left and right floodplain are combined, and the reach lengths are averaged together.

Stream junctions are also handled very differently in the new 1D Finite Volume scheme. The 1D finite difference scheme had two options. The default option was to assume all cross sections bounding the junction had the same water surface elevation. An option the user can turn on is to compute the water surface across the junction using a steady flow energy solution each time step. For the 1D Finite Volume solution, the junction is treated as a single 2D cell, and the 2D shallow water equations are solved through the junction. This allows for more complex junctions within a 1D framework. This approach also allows for the transfer of momentum through the junctions, as well as a more detailed calculation of the water surface elevations around the junction. The 1D model must be georeferenced in order for this to work correctly, as that is how the software forms the 2D cell correctly. See the example in Figure 5-9 below.

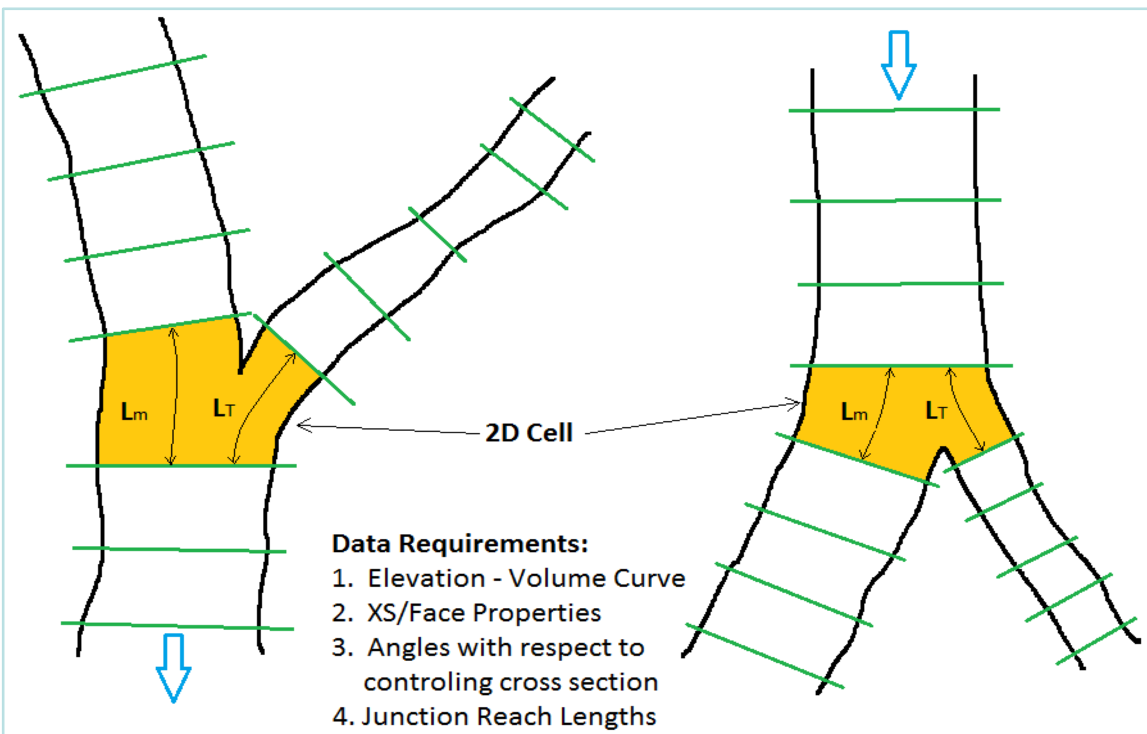


Figure 5-9. 2D Cell used for Modeling Junctions in new 1D Finite Volume Scheme.

To use the new 1D finite volume solution scheme, select **Computational Options and Tolerances** from the Unsteady Flow Analysis window. Then select the **General** tab, and you will see the window as shown in Figure 5-10 below:

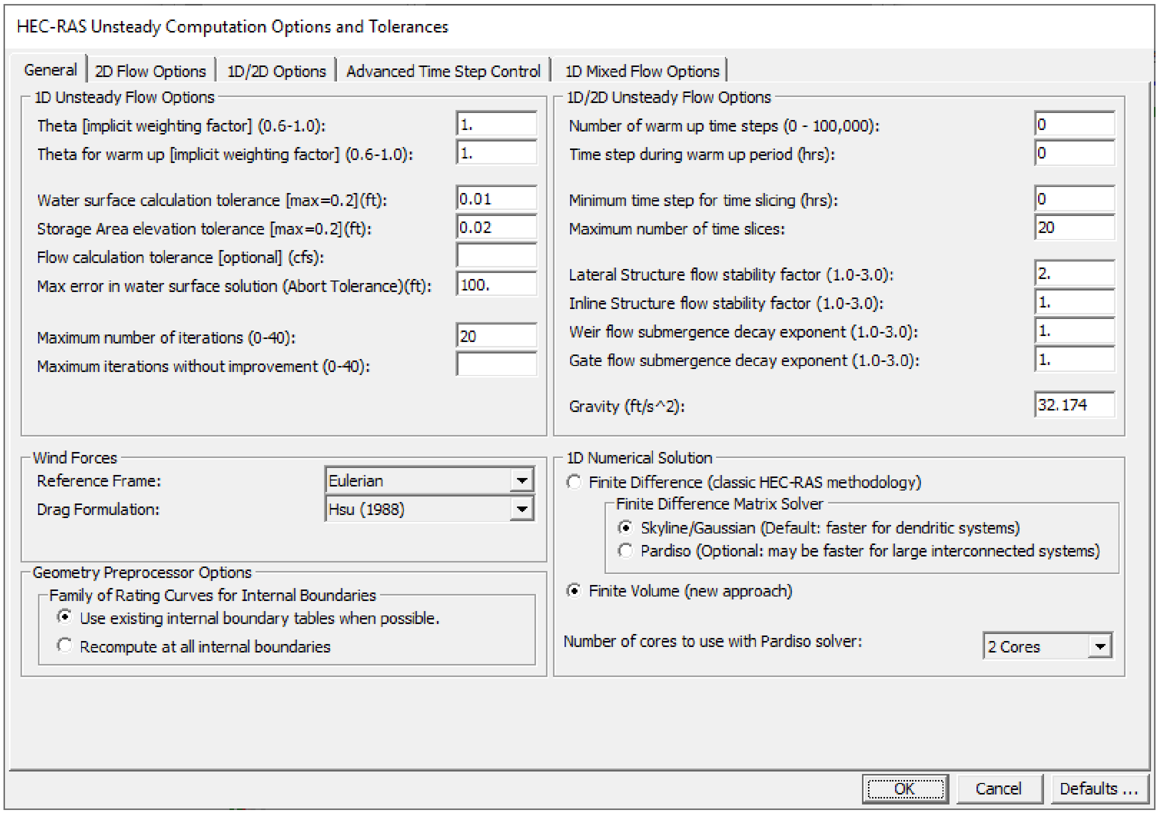


Figure 5-10. General Tab from the Unsteady Flow Analysis Computational Options and Tolerances.

As shown in Figure 5-10, to turn on the new 1D Finite Volume solution scheme, simply select **Finite Volume (new approach)** from the **1D Numerical Solution** area in the lower right hand side of the window. The user can also control the number of cores used to solve the problem. However, 1D models are generally small in comparison to 2D models. Most of the time, the solution will be solved the fastest with a single core.

An example of a model that was started completely dry, and containing bridges, is shown in Figure 5-11.

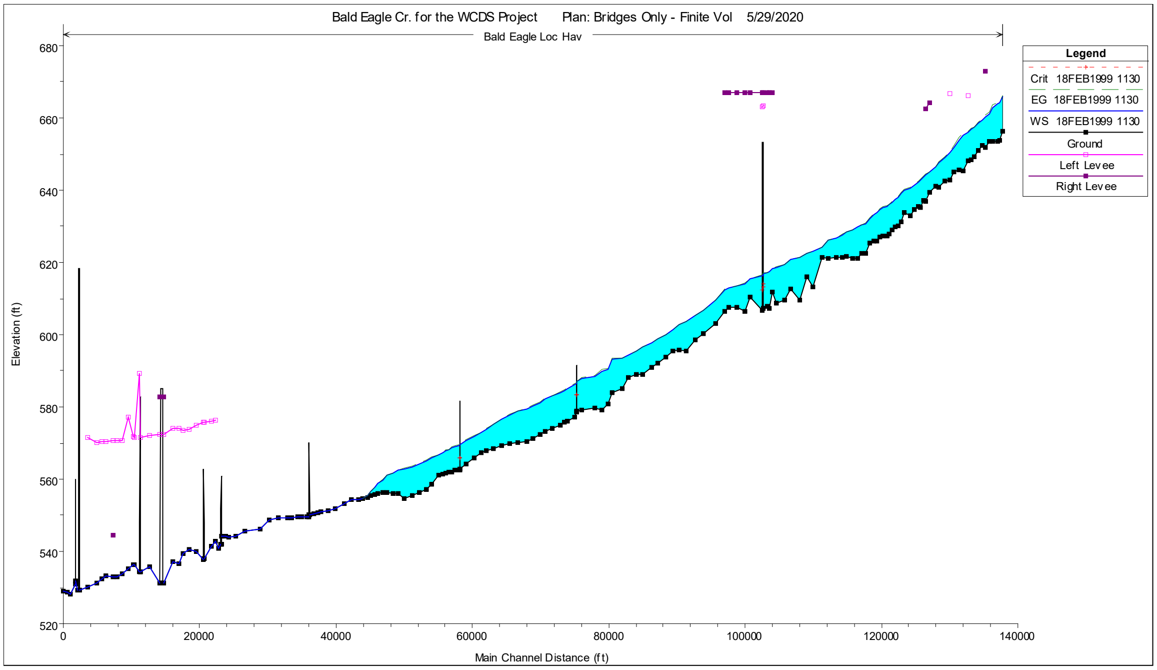


Figure 5-11. Example Model using 1D Finite Volume and Starting Dry.

In general, the 1D Finite Volume solution algorithm is more robust (greater model stability) that the existing 1D finite Difference solution scheme. However, there are some draw backs to the new 1D Finite Volume solution scheme. These deficiencies are:

1. **You cannot use lidded cross sections with the new 1D Finite Volume solution scheme**. Which also means that it cannot handle pressurize flow, even with the Priessman slot option turned on. If you have lidded cross sections, as soon as the water hits the high point of the lids low chord it will go unstable.

2. **The 1D Finite Volume solution scheme is sensitive to the volume of water between any two cross sections.** The equations are written from a "volume" perspective. If two cross sections are very close together, then there is very little volume between those two cross sections. The 1D Finite Volume scheme will require smaller time steps in order to handle the change in volume over the time step for this type of situation. The 1D Finite Difference scheme handles this better, because the equations are written in terms of change in water surface and velocity, which may be a small change for cross sections that are closer together. Therefore, for the 1D Finite Volume scheme to work well with larger time steps, users may need to remove cross sections that are very close together.

3. **The 1D Finite Volume scheme is computationally slower, than the 1D Finite Difference solution scheme.** This is because the 1D Finite difference scheme combined the properties of the left and right overbank together (area, wetted perimeter, average length between cross sections, etc.) and solved the equations for a main channel and a single floodplain. The new 1D Finite Volume solution scheme keeps the left and right overbank properties completely separate. Thus the equations are written for a separate left overbank, main channel, and right

over bank, and then solved. This is more computational work for every time step, but it is computationally more accurate.

2. **Maximum number of Iterations without improvement.** This option is off by default, but if the user turns it on, it will monitor the maximum numerical error computed during the 1D Iterations, and if the error does not improve within the specified number of iterations, then the 1D solver stops iterating and goes on to the next time step. For example, let's say the default maximum number of iterations set to 20. If the "Maximum number of iterations without improvement" is set to 5, then during any time step, if the iteration scheme does not continue to improve the numerical solution for 5 iterations in a row it will stop and go to the next time step, using whichever previous iteration was the best solution. In general, 5 is a good number to start with for this option, but the user may want to try lowering it. This option will improve computational speed for data sets that iterate a lot. However, if you turn it on and set the value too low, you may increase the model instability.

3. **1D Equation Matrix Solver.** We use a matrix solution solver called "Skyline" which uses Gaussian elimination for reducing the size of the matrix. It has been streamline towards dendritic river systems, and is very fast. However, sometimes HEC-RAS models can be very large and have many interconnections (loops in the stream network, or many interconnected storage areas). We have added an option to solve the 1D matrix with the "PARDISO" solver that we use in 2D. This solver has the benefit of being able to use multiple cores. In experiments at HEC, we have found that the Skyline Matrix solver is still faster for dendritic systems. However, large models with lots of Lateral structures, storage areas, and loops in the reaches, **may** be solved faster using the PARDISO solver. Try it out to see which one works better on your specific data set. We do not have a lot of experience in using this solver on the 1D side. So use it at your own risk. That is, don't just compare the computational times, also compare the results to make sure they are the same.

Note: We got rid of the option to **"Convert 1D Energy Bridges to Cross Sections with Lids"**. This option was not used often, and in some cases caused model stability issues. So now all bridges are pre-processed into a family of curves. If this option was turned on in your model, this change may produce different computed results in the vicinity of that bridge.

Wind Force Options

Specifying the wind data in the Unsteady Flow Data editor automatically turns on wind forcing in the computational engine. However, there are some additional computational options available to the user from the **HEC-RAS Unsteady Computation Options and Tolerances** window. This is done by opening the **Unsteady Flow Analysis** window and select the men **Options** | **Calculation Options and Tolerances…** as shown in the figure below.

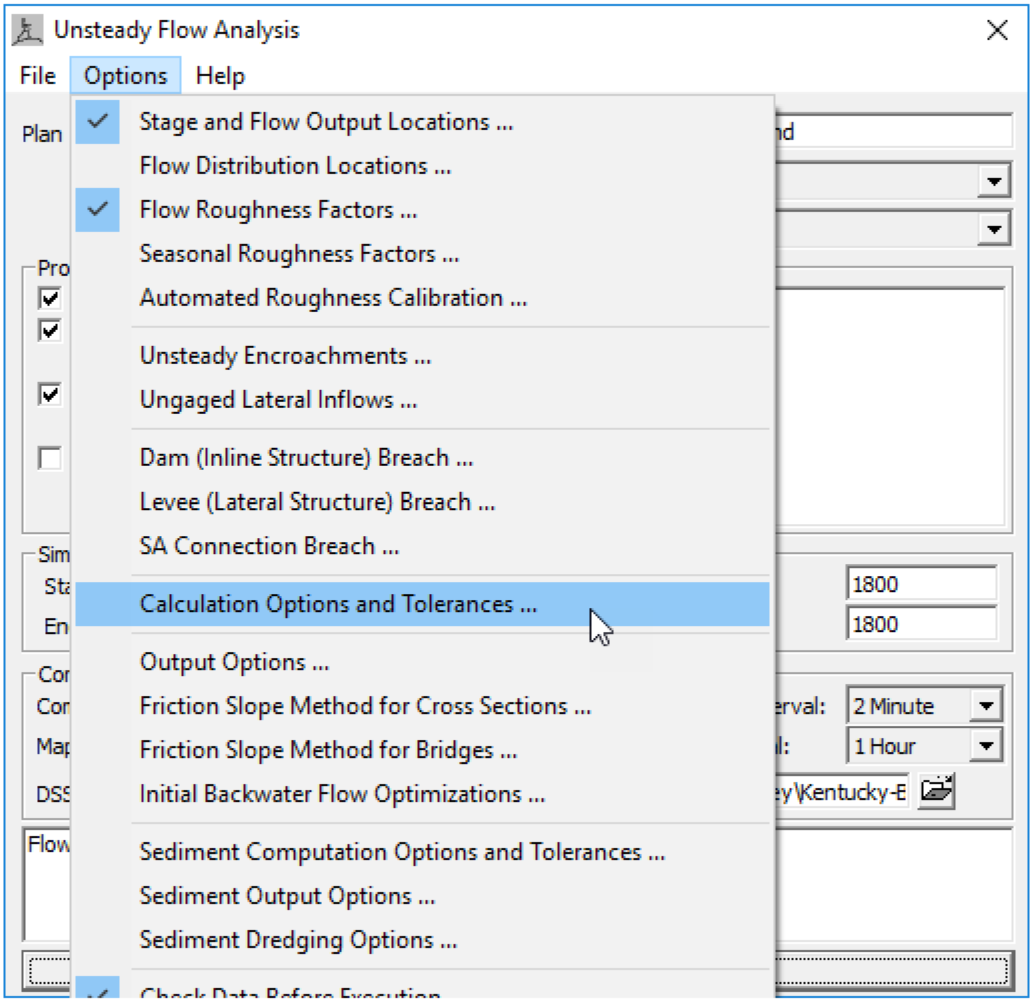


Figure 5-12. Selecting the menu Options | Calculations Options and Tolerances within the Unsteady Flow Analysis window.

Within the **HEC-RAS Unsteady Computation Options and Tolerances** window (shown below), the wind **Reference Frame** may be specified as **Eulerian** or **Lagrangian**. An Eulerian reference frame only takes into account the speed of the wind when computing the wind drag forces. The Lagrangian reference frame also takes into account the speed and direction of the water.

The wind **Drag Formulation** may be specified using the drop-down menu. Users can choose from four different equations to compute the drag coefficient (i.e. Hsu (1988), Garratt (1977), Large and Pond (1981), Andreas et al. (2012)), or they can enter the drag coefficient as a constant value. Please see Chapter 2 of the Hydraulic Reference Manual for the details of the different wind drag equations and their applicability.

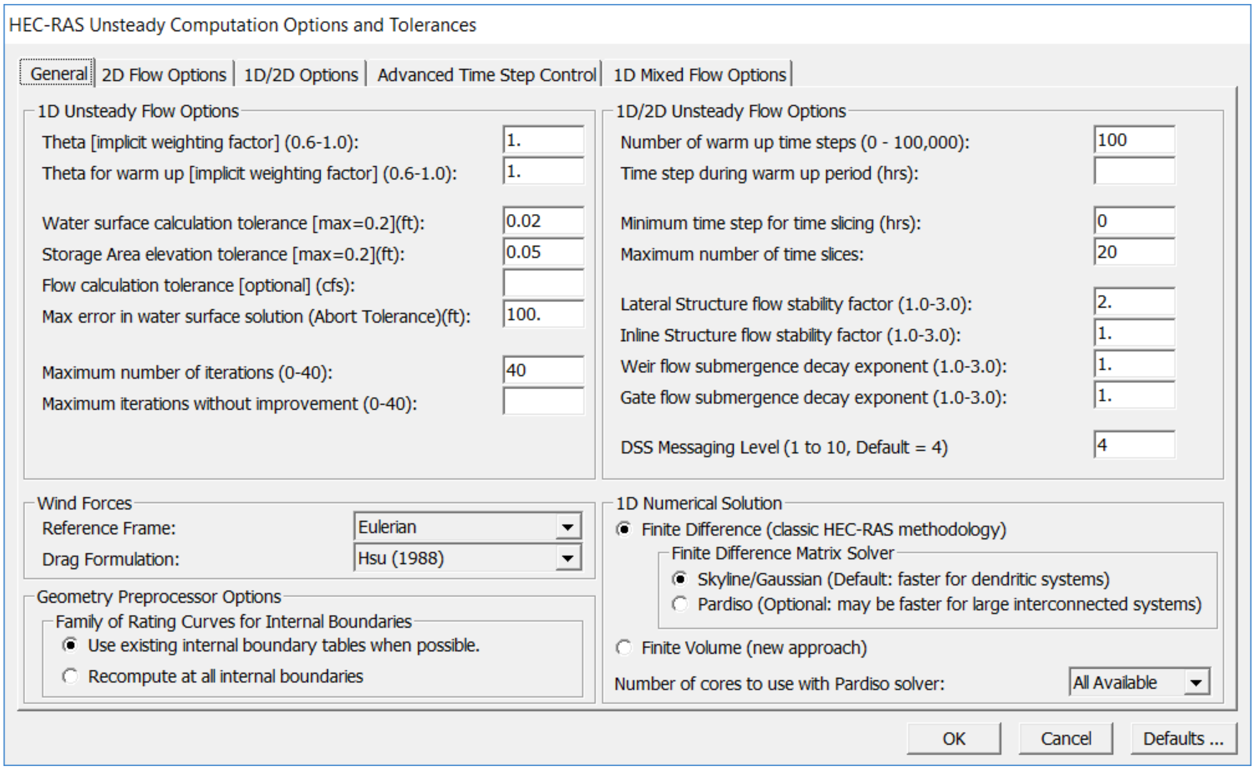


Figure 5-13. HEC-RAS Unsteady Computation Options and Tolerances dialog shown the Wind Forcing options.

64-bit Computational Engines

HEC-RAS now only supports 64-bit computational engines. The 64-bit computational engines can handle larger model data sets, and will also run faster than the 32-bit engines. Because of this, users are now required to have a 64 bit operating system installed on their machines. This and future versions of HEC-RAS will no longer run on 32-bit operating systems.

CHAPTER 6

Viewing 2D or 1D/2D Output using HEC-RAS Mapper

Once the user has completed an unsteady flow run of the model, the user can look at all of the 1D output (plots and tables) in the same manner as before, using the traditional plots and tables. However, the 2D output results can only be viewed within RAS Mapper. Currently, the user can visualize inundation areas (and other types of output) within RAS Mapper for River Reaches, Storage Areas, and 2D flow areas at the same time. To visualize the output, select the **GIS Tools | RAS Mapper** menu on the main HEC-RAS window (or just select the RAS Mapper button  on the main HEC-RAS window). The RAS Mapper window shown in Figure 6-1 will appear.

Note: This chapter is an overview of RAS Mapper to assist you with visualizing 1D/2D output. For more details on RAS Mapper, see the separate RAS Mapper User's manual.

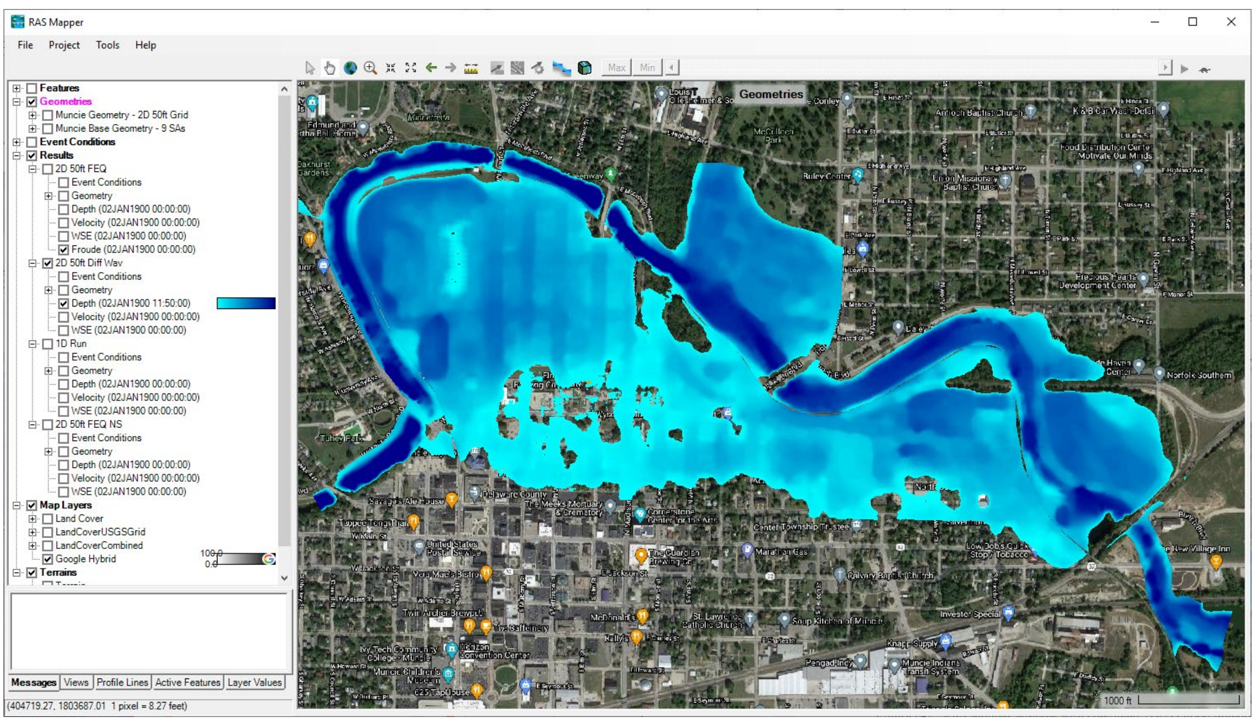


Figure 6-1. RAS Mapper window with combined 1D/2D flood inundation output displayed.

Overview of RAS Mapper Output Capabilities

HEC-RAS Mapper can be used to develop terrain models and visualize HEC-RAS results in a map-based format. For a more detailed description of RAS Mapper, see the RAS Mapper User's Manual. RAS Mapper has the following capabilities:

1. Develop terrain models for use in 2D modeling and visualizing 1D/2D model results. Terrain models can be developed from one or more terrain tiles, and these tiles can have different grid resolutions.
2. Develop Land Cover Layers (for use in defining Manning's n values and percent impervious); soils layers, and infiltration layers.
3. Various types of map layer results can be generated, such as: depth of water; water surface elevations; velocity; inundation boundary (shapefile); flow (1D only right now); depth times velocity; depth times velocity^2; Courant number; Froude number; Residence Time; Shear stress; Energy Depth; Energy Elevation; arrival time; Arrival time (max); Recession; flood duration; percent time inundated; Stream power; and wet cells (completely color any cell that gets wet).
4. Computed model results can be displayed dynamically on the fly, or they can be written to a static (stored to disk) map layer/depth grid file.
5. Computed model results can be animated (dynamic mapping) or shown for a specific instance in time. There are also options for plotting maximum (Max) and minimum (Min) values at all locations.
6. Time series plots and tables can be displayed for 1D and 2D output directly from RAS Mapper, at any location where there is a map layer result. Time series plots and tables include: water surface elevation; depth; and velocity (2D node velocities, 2D average face velocities, and 1D velocities), etc….
7. Users can query any active map layers value by simply moving the mouse pointer over the map.
8. Web imagery, shape files, and point layers can be displayed as background layers behind the computed results. Shapefiles and Point layers can also be added to the "Features" layer in order to be plotted on top.
9. The user can make a terrain model from the cross sections (channel only or entire sections), the river and bank lines, and the cross-section interpolation surface. Elevations between the cross sections are interpolated using the interpolation surface and the cross-section elevations. This terrain model can then be combined with other terrain data (overbank/floodplain) to make a new terrain model in which the channel/cross section data is now burned into the overall terrain model.
10. Users can create User Defined Profile Lines, then request various types of output along those profile lines (i.e. WSE and terrain, velocity, depth, flow, etc…)

11. User can zoom into an area of their model then store that location as a User Define View. Then later you can click on any User define view and it will jump to that location and view.

Adding Results Map Layers for Visualization

Once an HEC-RAS model run is completed, and RAS Mapper is opened, there will be a **Results** Layer that has the same name as the HEC-RAS Plan Short ID for that run (see Figure 6-2 below). Beneath the **Results | Plan Short ID** Layer, will be a tree of related results. By default there will be an **Event Conditions** layer, a **Geometry** layer, **Depth** layer, **Velocity** layer, and **WSE (water surface elevation)** layer. The Geometry layer will contain the HEC-RAS Input Geometry layers that were used in that specific run. The Geometry layer includes sub layers of: River; XS (cross sections); Storage Areas; 2D flow areas; Bridges/Culverts, etc… Any or all of these Geometry layers can be turned on for visualization of model elements.

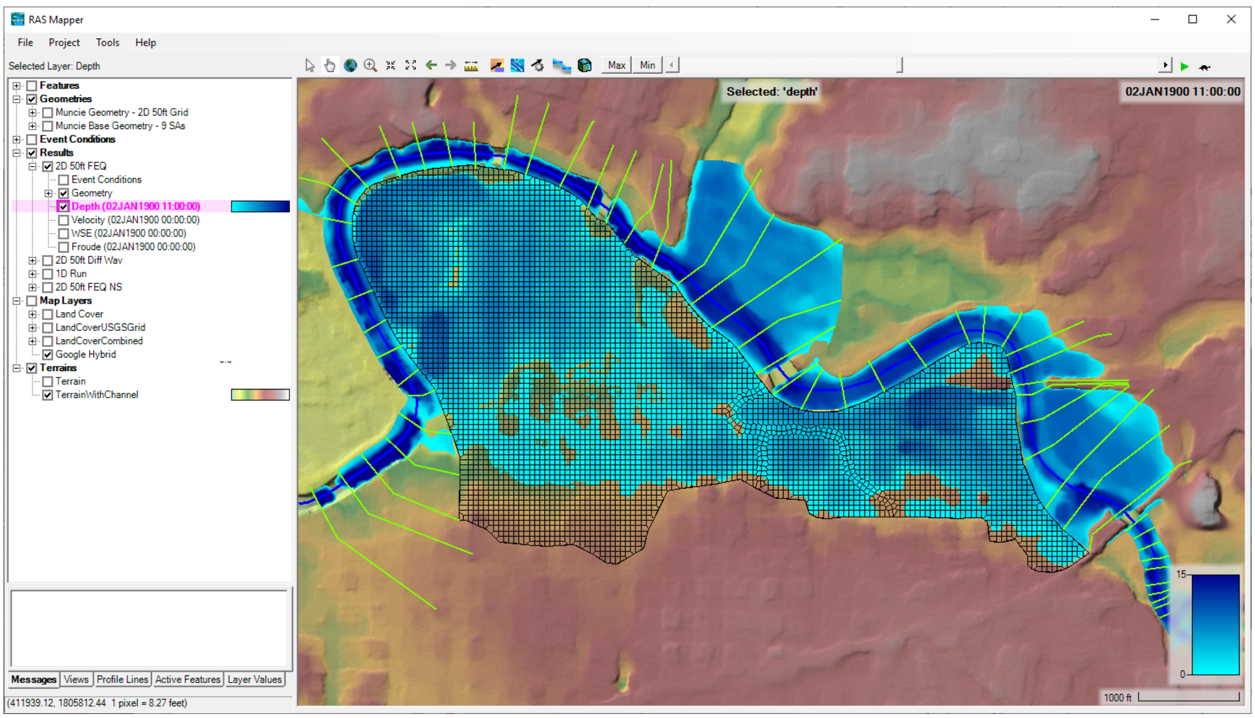


Figure 6-2. RAS Mapper with Default Results Layers shown.

By default, after a successful HEC-RAS model run, there will be three dynamic results layers called **Depth, Velocity, and WSE** (Water Surface Elevation). These layers can be used to visualize the model results in an inundation mapping form (e.g. two-dimensional map of the geometry, with water and other layers on top of it). The layers will be computed and displayed on-the-fly, meaning RAS Mapper reads the computed model result from a file, and then it computes the map in memory and displays it as needed. The underlying terrain used for computing the map layers is based on the view scale of the map. If the user is zoomed in, the base (raw) data will be used for computing the inundation map layers; however, if the user is zoomed out, a re-sampled version of the terrain is used. Therefore, the displayed map layers may change slightly based on the scale at which the user is zoomed. By default, the map layers are not pre-computed grids stored on the hard drive. By computing the map layer on the fly, the mapping is actually faster and takes much less disk space. The user has the option to create a "Stored" map layer (a grid stored to the hard disk) if desired. The "Stored" grids are based on the raw (most detailed) terrain layer for computing the grid.

Other results layers are available for visualization, but the user has to request/create a results layer to display. To create a new results layer, right click on the desired Plan Name (listed in the Results Layer) and select the option called **Add New Results Map Layer**. This option will bring up a window that will allow you to select a new Results Map Type (see Figure 6-3). This window can also be displayed by selecting **Tools | Manage Results Maps**. Then the **Results Map Manager** will appear, and the user can then select **Add New Map** button from any of the Plan names listed in that window to create a new results map layer.

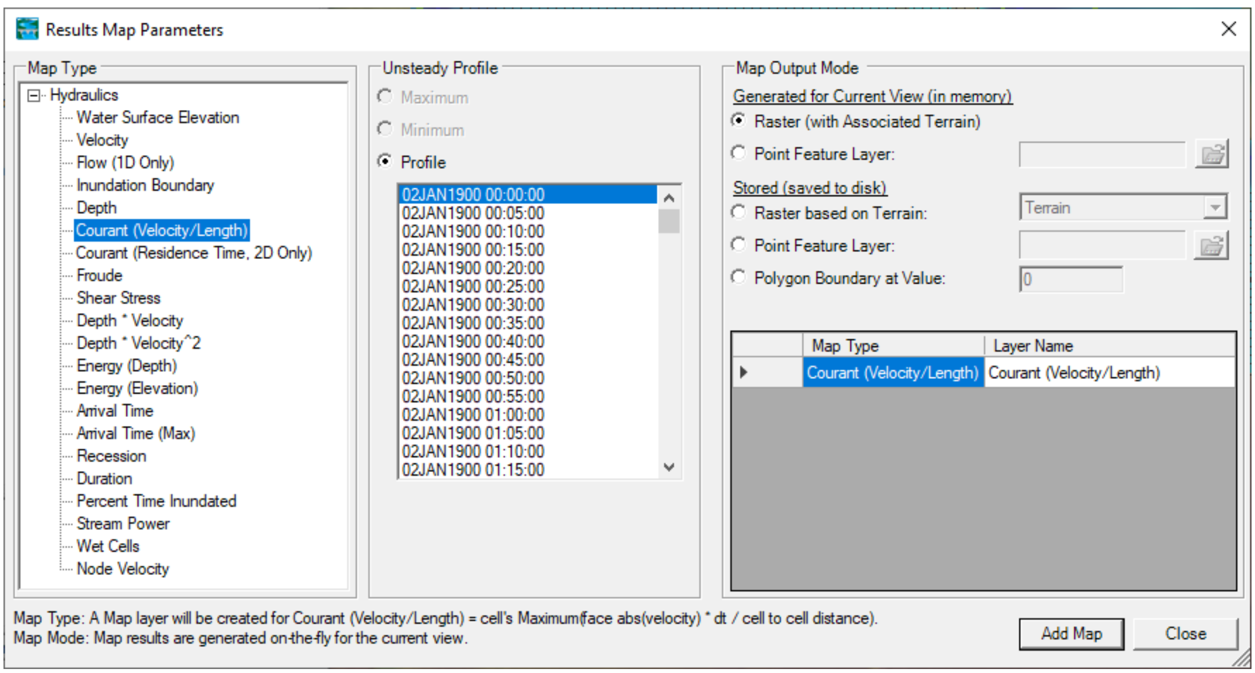


Figure 6-3. Example of the Results Map window used to create new results map layers.

As shown in Figure 6-3, the new **Results Map Parameter** window has three sections to select from. On the left is the **Map Type**, where the user selects the parameter to map (create a layer for). Currently RAS Mapper allows the user to create 20 different Map Types (Table 6-1).

After a Map Type is selected, the middle section of the window (**Unsteady Profile**) is used to pick the profile type: Maximum (Max stage everywhere regardless of time); Minimum (Min stage everywhere regardless of time); or a specific date and time (results at that specific instance in time). If a map is going to be displayed dynamically (computed in memory and displayed on-the-fly), it does not matter what is picked for the profile, the user will be able to dynamically visualize all the profiles. If a map needs to be created as a static map (a results or depth grid written to a file) then the specific profile picked will be used for that static map.

Table 6-1. Current RAS Mapper Map Types.

Map Type	Description
Depth	Water depths computed from the difference in water surface elevation.
Water Surface Elevation	Water surface elevations at all computed locations and spatially interpolated water surface elevations between those locations.
Velocity	Velocity at all computed locations and spatially interpolated velocity between those locations.
Inundation Boundary	Inundation boundary computed from the zero-depth contour of flood depths for the selected water surface profile.
Flow (1D Only)	Computed flow values at the 1D cross sections and Interpolated flow values between cross sections. This option is only available for 1D simulations
Shear Stress	Shear stress computed as: (γ R Sf). For 2D cells it is the average shear stress across each face, then interpolated between faces. For 1D cross sections, the cross section is broken into user defined slices, then average values are computed for each slice. Values are interpolated between cross sections using the cross section interpolation surface.
Depth * Velocity	Computed as the hydraulic depth (average depth) multiplied by the average velocity at all computed locations and spatially interpolated between those locations. For 2D cells the hydraulic depth is computed for each Face, then multiplied by the average velocity across that face. For 1D cross sections, the cross section is broken into user defined slices, then average values are computed for each slice. For unsteady-flow runs, the Maximum value is the maximum of depth times velocity based on the user mapping interval, and not the computation interval.
Depth * Velocity^2	Computed as the hydraulic depth (average depth) multiplied by average velocity squared at all computed locations and spatially interpolated between those locations. For 2D cells the hydraulic depth is computed for each Face, then multiplied by the average velocity squared across that face. For 1D cross sections, the cross section is broken into user defined slices, then average values are computed for each slice. The Maximum value is the maximum of depth times velocity squared based on the user mapping interval, and not the computation interval.
Arrival Time	Computed time (in hours or days) from a specified time in the simulation when the water depth reaches a specified inundation depth (threshold). The user may specify the time units, start time, and depth threshold.

Map Type	Description
Arrival Time (Max)	Arrival time max is the time it takes, from a user specified time and date, to get to the maximum water surface at each location (XS, 2D cell, storage area)
Recession	Computed time (in hours or days) from a specified time in the simulation when the water depth recedes back below a specified inundation depth (threshold). The user may specify the time units, start time, and depth threshold.
Duration	Computed duration (in hours or days) for which water depth exceeds a specified flood depth (threshold). The user may specify the time units, a start time, and depth threshold. (Note: RAS ignores multiple peaked events. Once a depth threshold is reach the duration continues until the depth has completely receded for the event.)
Percent Time Inundated	The amount of time an area is inundated as a percentage of the total simulation time range.
Stream Power	Stream Power is computed as average velocity time's average shear stress. For 2D cells it is the average velocity times average shear stress across each face, then interpolated between faces. For 1D cross sections, the cross section is broken into user defined slices, then average values are computed for each slice. Values are interpolated between cross sections using the cross-section interpolation surface.
Courant Number	The Courant number is value used to assist understanding numerical stability. Review the section in Chapter 5 of this document on picking a computational time step for more details on Courant number.
Froude Number	The Froude number is used to define if the flow regime is subcritical, supercritical, or at critical depth. If Froude number is greater than 1.0, then the flow is supercritical. If the Froude number is less than 1.0, then the flow is subcritical. If the Froude number is 1.0, then it is at critical depth.
Residence Time (2D only)	Residence time is the time it would take for the volume of water in a cell to leave, given the current velocity of the flow leaving the cell.
Wet Cells	This map simply shows which 2D cells are wet. When using this map layer, if a 2D cell even gets a drop of water it shows the entire cell as wet.
Energy (Depth)	The energy depth is the depth of the water plus the velocity head, mapped as an Energy Depth.
Energy (Elevation)	The energy elevation is computed as the water surface elevation plus the velocity head, and mapped as an Energy Elevation.

Arrival time and duration map layers, require additional information from the user: whether to write the results out in hours or days; a depth threshold (default is zero, but the user may want to enter a higher value, like 0.5 or 1.0 feet); and finally a starting data and time to be used for the evaluation (this may be the start of a warning time, which would then make the arrival time calculation a warning time).

The last thing to select on the window is the **Map Output Mode** (Table 6-2). The **Map Output Mode** is where the user selects whether the map will be a Dynamic layer, **Generated for Current View (in memory)**, or a **Stored (Saved to disk)** map layer. **Dynamic** layers get computed on-the-fly as needed and can be animated through the time steps of the solution. **Dynamic** maps are the most useful for visualizing the results. **Stored** maps only need to be created when the user wants to create a depth grid, or other layer type, that needs to be written to the hard disk. A **Stored** layer can be used by another program (for example by HEC-FIA to compute damages or life loss), or it can displayed in a GIS and used for another purpose.

Note: The Dynamic mapping uses a pyramiding scheme to display the maps (Just like Google does). You have to be zoomed way in to see the full flooding in detail. As you zoom out, the software grabs a coarser terrain layer in which 4 of the original terrain cells are averaged into a single cell (average elevation) then an inundation map is made from that. If you zoom out further, then an even coarser terrain layer is grabbed (based on how far out you are zoomed), then it creates an inundation map from that. So the further you zoom out, the more coarse the terrain that gets used (i.e. more averaged). This is done for speed. There would be no way we could produce and display the maps on the fly so fast without doing this. Also, Dynamic mapping is done with a sloping water surface TIN, which is then intersected with a TIN of the Terrain surface, in order to find the zero depth intersection.

Stored maps are made from the most detailed level of terrain (Your base terrain). However, stored maps are grids in which a single water surface elevation (or other data layer value) is stored per cell (cell size is based on the terrain data being used to create the map). Ultimately the Dynamic maps and Stored maps can show some differences in inundated area, depending on your zoom level and the cell size used in creating your terrain model.

Table 6-2. RAS Mapper Map Output Modes.

Map Output Mode	Description
Dynamic	Map is generated for current view dynamically in memory (RAM). Results may also be animated if there are multiple profiles for the variable.
Raster	Gridded output is computed based on the associated Terrain.
Point Feature Layer	Values are computed a locations specified by a point shapefile. For the Depth map type, elevation values to compare against can come from the Associated Terrain, Z-Values from the points, or using a column for elevation data.
Stored	Computed map is stored to disk for a specific profile. Stored maps cannot be animated.
Raster	Gridded output is computed based on the user-specified Terrain and stored to disk. The Associated Terrain is the default.

Map Output Mode	Description
Point Feature Layer	Values are computed a locations specified by a point shapefile and stored to disk. For the Depth map type, elevation values to compare against can from the Associated Terrain, Z-Values from the points, or using a column for elevation data.
Polygon Boundary at Value	A boundary is polygon is created at the specified contour value stored to disk. This is the default option for the Inundation Boundary map type using the computed depth at the zero-depth contour.

Once a **Map Type**, **Unsteady Profile**, and a **Map Output Mode** have been selected, the user then selects the **Add Map** button and the map will be added to the Results Layer underneath the selected Plan.

Map Rendering Modes

There can be important differences in the computed values between the dynamic results and the stored raster information. Dynamic maps differ in that they are a surface created through the interpolation of values; therefore, as the user moves the mouse in the display the interpolated value for the corresponding location will be displayed. This is in contrast to viewing a raster in a typical GIS where the reported value will be that for the entire grid cell.

For dynamic depth results, the values reported to the user at a specific map location may change depending on the zoom level. This is because for dynamic mapping, the terrain pyramid level used for evaluating the ground surface elevation is dependent on how far the user has zoomed in or out on the map. Even while zoomed in so that the base data are used for analysis, the map results will not be identical, especially at the floodplain boundary. This is because for the stored depth grid the cell is considered either wet or dry. For a dynamic map, the flood boundary is determined by interpolating the elevations values and intersecting the interpolated water surface with the interpolated terrain elevations for the boundary. The dynamic map will, therefore, result in a "smooth" floodplain boundary. An example of the differences between the dynamic mapping and stored mapping are shown in Figure 6-4.

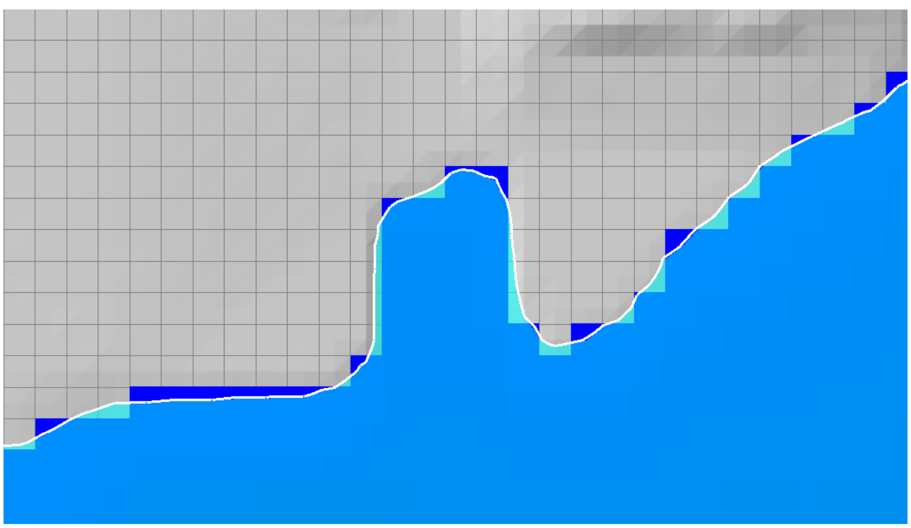


Figure 6-4. Differences in dynamic (smooth boundary) and stored (gridded) map results.

2D Mapping Options

RAS Mapper has three Rendering Mode options for how the water surface is interpolated and displayed for 2D model results. The **Render Mode** options are to plot the water surface as either **Hydrid (Sloping/Horizontal)** (default)**; Sloping;** or as **Horizontal** within a 2D cell. The render mode selection will affect both the dynamic map and the stored map results. To select the Render Mode, select **Tools/Options** to bring up the Options window, then select **Render Mode**, or select the render mode button  on the button layer. Either way the RAS Mapper Options window will appear with the Render Mode settings display in the window.

***Hybrid (Sloping/Horizontal* – Default) -** This render mode is a combination of both the Sloping and the Horizontal render modes described below. In general the water surface is plotted using the sloping rendering mode, except for locations where the water surface is not hydraulically connected. For example, two cells that are wet, but have a Face between them that is dry. These two cells are not hydraulically connected, and therefore the sloping rendering model should not be used to plot the water surface across these two cells. Instead, the horizontal rendering mode is used for both cells.

Sloping Water Surface – The Sloping water surface rendering mode plots the computed water surface by interpolating water surface elevations from each 2D cell face. This option of connecting each cell face provides a visualization for a more continuous inundation map. The more continuous inundation map, looks more realistic; however, under some circumstances it can also have the appearance of more water or less water volume in the 2D cells than what was computed in the simulation. This problem generally occurs in steep terrain with large 2D grid cells. This sloping water surface approach is most helpful when displaying shallow inundation depths in areas of steep terrain. This is the default rendering mode. However, users should plot with both rendering modes and closely inspect the differences.

Horizontal Water Surface – The Horizontal water surface rendering mode plots the computed water surface as horizontal in each 2D Area cell. This option fills each 2D cell to the water surface as computed in the 2D simulation. In areas where the terrain has significant relief between 2D cells this plotting option can produce a "patchwork" of isolated inundated areas when visualizing flood depths. These isolated inundation areas are more visible in areas of steep terrain, using large grids cells, with shallow flood depths.

An illustration of the sloping and horizontal rendering modes is shown in Figure 6-5 while Figure 6-6 shows the spatial differences due to the rendering mode.

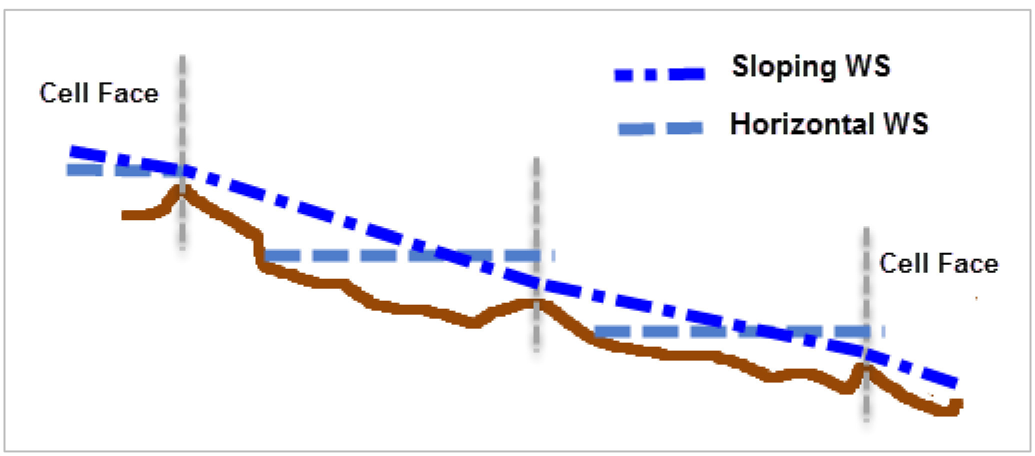


Figure 6-5. Comparison of Sloping and Horizontal water surface rendering for steep terrain.

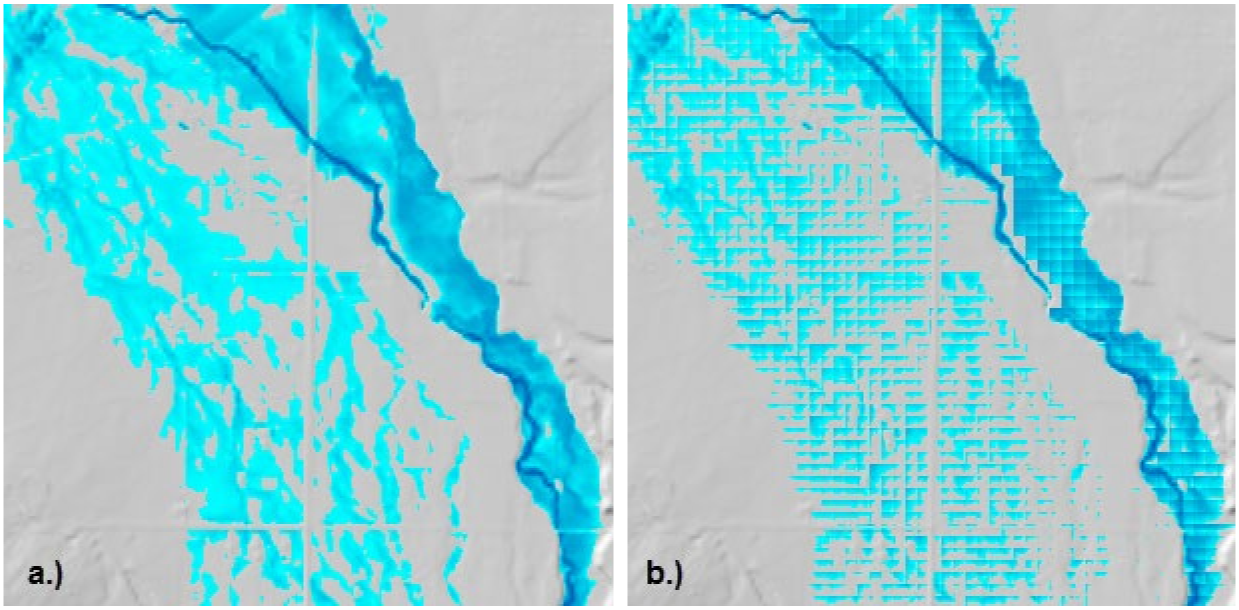


Figure 6-6. Comparison of (a) sloping and (b) horizontal water surface mapping option for inundation depth.

Dynamic Mapping

As shown in Figure 6-7, there will be results layers for each Plan that has been run. For this example, under the **Results** layer in the panel on the left side, there are results for a Plan called "2D 50 ft Grid". Under the "2D 50 ft Grid" layer there are five sub layers: **Event Conditions**, **Geometry**, **Depth**, **Velocity**, and **WSE (water surface elevation)**. The **Event Conditions layer** will contain any spatial meteorological data that has been set up for the boundary conditions file attached to that plan (ex. Precipitation data). The **Geometry Layer** represents the geometry data used in the plan and written to the output file. The other layers (**Depth**, **Velocity**, and **WSE**) are **Dynamic Mapping** output layers. Each layer name is followed by a set of parentheses. The information following in the parentheses describes what current grid is showing in the plot. Inside the parentheses could be a specific date and time, or max, or min.

Right clicking on any of the results layers (for example **Depth**) will show a context sensitive menu that has several options for this output layer. These options are: Layer Properties; Edit Map Parameters; View Map in 3D Viewer; Zoom to Layer; Add Watch to Layer Values; Remove Layer; Move Layer; Export RAS Tiles for Web Mapping (Figure 6-7) and Show Summary Statistics.

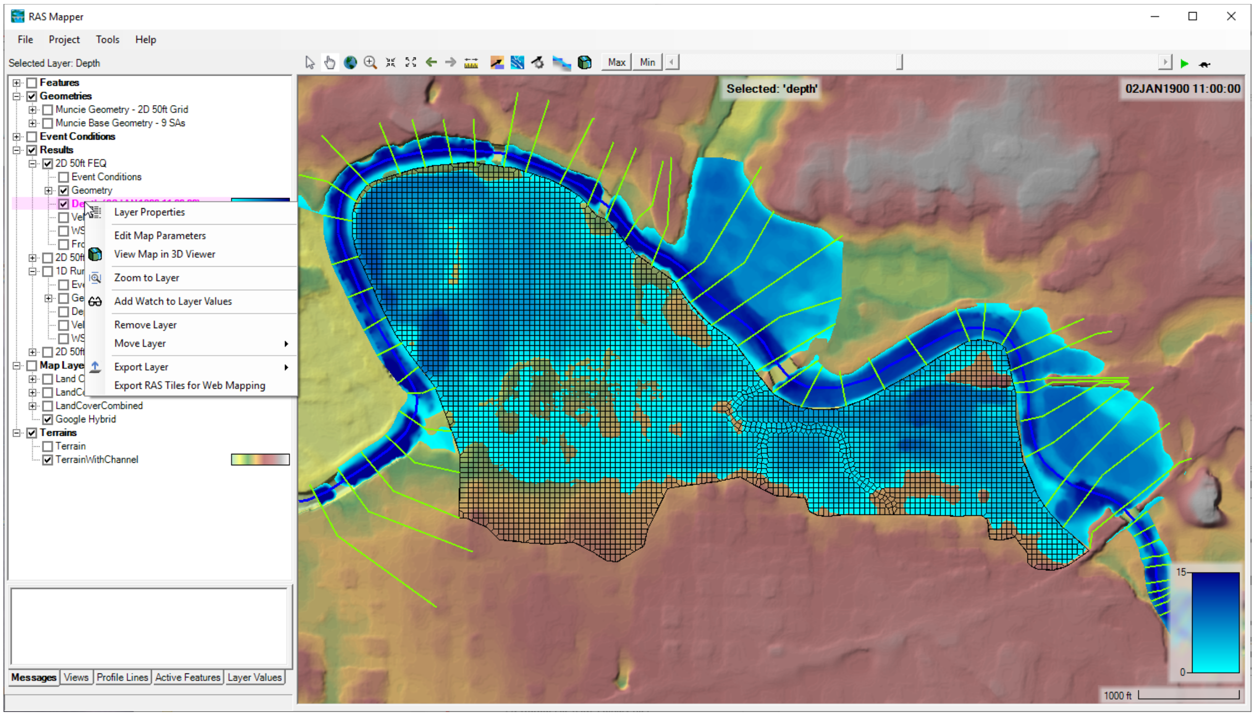


Figure 6-7. RAS Mapper with the Options menu displayed for a depth output layer.

Layer Properties – This option allows the user to control what each layer looks like, such as colors, color ramps, symbol and line colors and types, fill styles, etc… It also contains a tab called "Source" which provides information about the layer, as well as what the file name is, where it is stored, etc…

Edit Map Parameters – This option allows the user to edit or change the parameters of the map layer.

View Map in 3D Viewer – this options opens the currently selected result into the 3D Viewer.

Zoom To Layer – This option zooms in or out, such that the extents of the selected layer are visible.

Add Watch to Layer Values – This option allows the user to add variables to a "Watch" list. When turned on, the variables will show up next to the mouse pointer and also in a table at the bottom left, when the "**Layer Values**" tab is selected. As you move the mouse over top of the wet area on the map, the magnitude of the watch variables is displayed dynamically.

Remove Layer – This option allows the user to delete the layer from the RAS Mapper window.

Move Layer – This option allows the user to move the layer up or down within the RAS Mapper layer list. This is important, since something that is higher in the list (above) will be plotted on top of things that are lower in the list (below), when multiple layers are turned on together for display. When this option is selected, several sub menus will popup that allow the user to move a layer: Top; Up One; Down One; Bottom. The user can also left-click on a layer and drag it up and down within the layers list.

Export Layer – This option allows the user to export the layer as a Raster (*.tif) to a file.

Export RAS Tiles for Web Mapping– This option will export the Depth grid to what is termed a Tile Cache (before doing this the user must set the spatial projection using the **Set Projection for Project** option from the RAS Mapper **Tools** menu). A Tile Cache is a series of files (256 x 256 pixels) with different resolutions, written out using a pyramided and tiled scheme that meets Google's mapping specifications (i.e., there will be several directories, each one representing a different resolution). When this option is selected, the window shown in Figure 6-8 will appear.

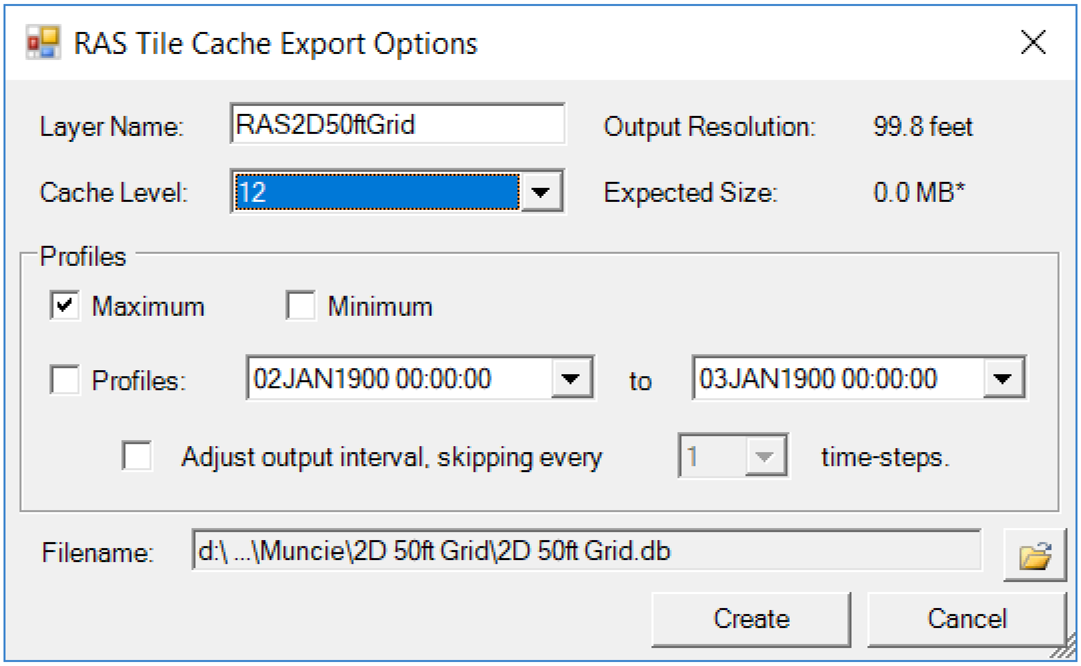


Figure 6-8. Tile Cache Options Editor.

The user is required to set the Filename and Folder in which these subfolders will be developed and the tiles will be stored. Additionally the user should select the **Cache Level** that will represent the most detailed tile level (when zoomed in). If the user plans on zooming way in for greater detail in the map, a higher Cache level (smaller cell (pixel) size) should be selected. However, the higher the cache level selected, the longer it will take to generate this Tile Cache, and the larger the file storage space required. Also, this cell (pixel) size will change depending on the latitude of the project.

Show Summary Statistics – When this option is selected for a specific layer (ex. Water Surface Elevation), the software will compute the following summary statistics for that layer based on the currently displayed time step: Mean; Standard Deviation; Minimum; and Maximum.

Animating Map Layers

Any Map Layer that is "Dynamic" can be animated in time. The animation control can be used to animate a single map layer; multiple map layers within the same Plan; or multiple map layers from different plans.

To animate a single map layer, turn that map layer on, then make it the active map layer (Layer will be highlighted in a Magenta color). Once a layer is turned on, and made the active layer, then the animation control at the top of the map window, can be used to animate that layer in time. The animation control has a play button, as well as Max, and Min options.

To animate multiple map layers within the same Plan, turn on all of the desired map layers within the Plan, then select the Plan identifier to make it the Active plan (It should be highlighted in Magenta). Then use the animation control to animate all of that plans layers in time together.

To animate multiple map layers from different plans, turn on all the desired map layers to be animated from separate plans. Next, select the results layer to activate all of the results for animation (The **Results** layer will be highlighted in Magenta). Then use the animation control to animate all of the turned on map layers in time. An example of animating two different Depth layers, from different plans, is shown in Figure 6-9. As long as both plans have output for the same date and time, they will be shown. The software uses the highest layer in the tree for deciding on the time window, and time steps available for animation. If any of the other map layers that are turned on do not have an output at that specific point in time, then it is simply not shown, and the layer will have a label called "No Data" for the date and time portion of the label.

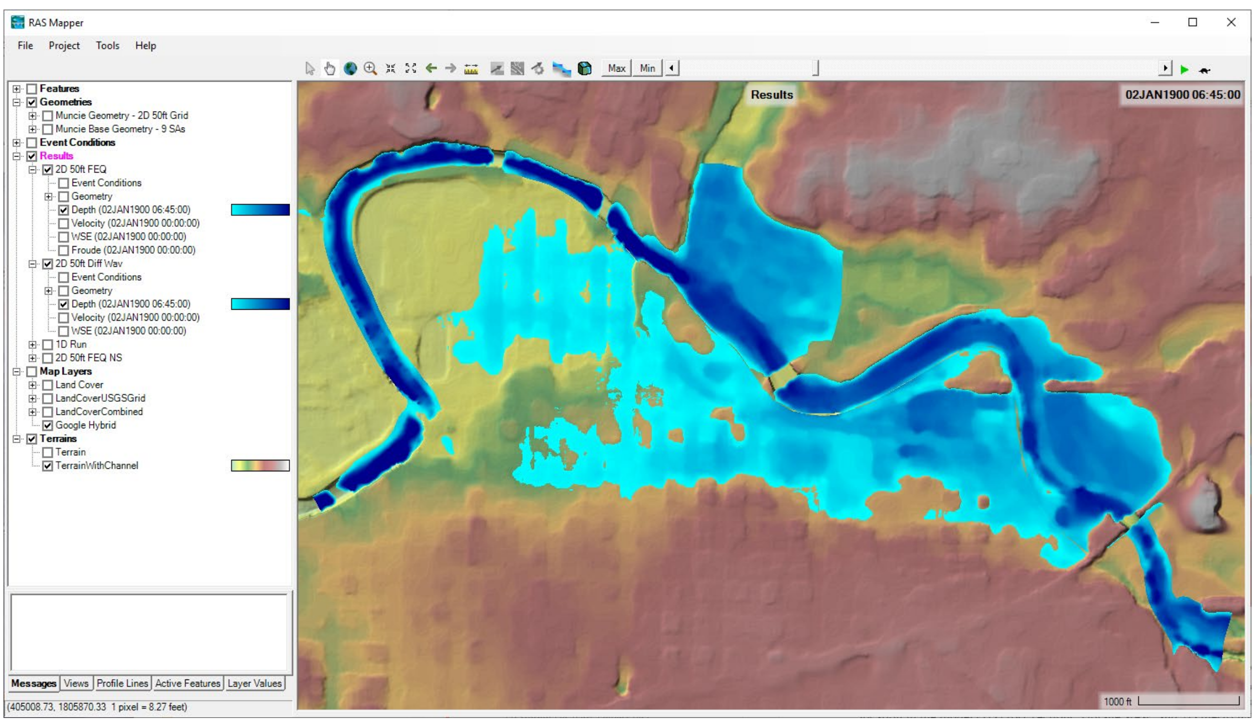


Figure 6-9. Dynamic Mapping/Animation Tool Bar.

As shown in Figure 6-9, when animating the **Depth** layer for a specific Plan(s), the user will have the option of selecting: **Max**, **Min**, or using the slider bar to move to a specific point in time during the event. If **Max** is selected, then the maximum inundation that occurred at every location in the model (1D cross sections, storage areas, and 2D mesh) will be plotted for the depth grid. If **Min** is selected, then the minimum depth that occurred at every location within the event will be displayed. If the user selects the Play button (far right green arrow), then the map(s) will automatically plot each mapping output interval in sequence (i.e. animate). The animation can be stopped by selecting the **Pause** button. If the user uses the slider bar, they can select a specific date and time, which will result in showing the inundation map for that specific instance in time. The times available are based on the Mapping Output Interval specified in the Unsteady-Flow Simulation window. The user can also hold down the right arrow or left arrow at the ends of the slider bar to have it animate the map, either forwards or backwards in time. Keep in mind these maps are being created on-the-fly in memory. They do not exist in a file on the disk. Because these maps are being created on-the-fly, this is referred to as Dynamic Mapping (Try it out, it's fun!!!). It is also the best way to visualize the results of the model from a mapping perspective.

To create a movie file, one option is to use commercial software, such as Snagit® software (or similar screen capture utility package) to capture the screen while animating the inundation results. Another Option for capturing video is to use the built in video capture tools that are part of Microsoft office. This is available from the PowerPoint software application.

Creating Static (Stored) Maps

The user can create a static map (map stored to the disk) at any time from RAS Mapper by right clicking on **Results | Manage Result Maps** menu item. When this option is selected the window shown in Figure 6-10 will appear.

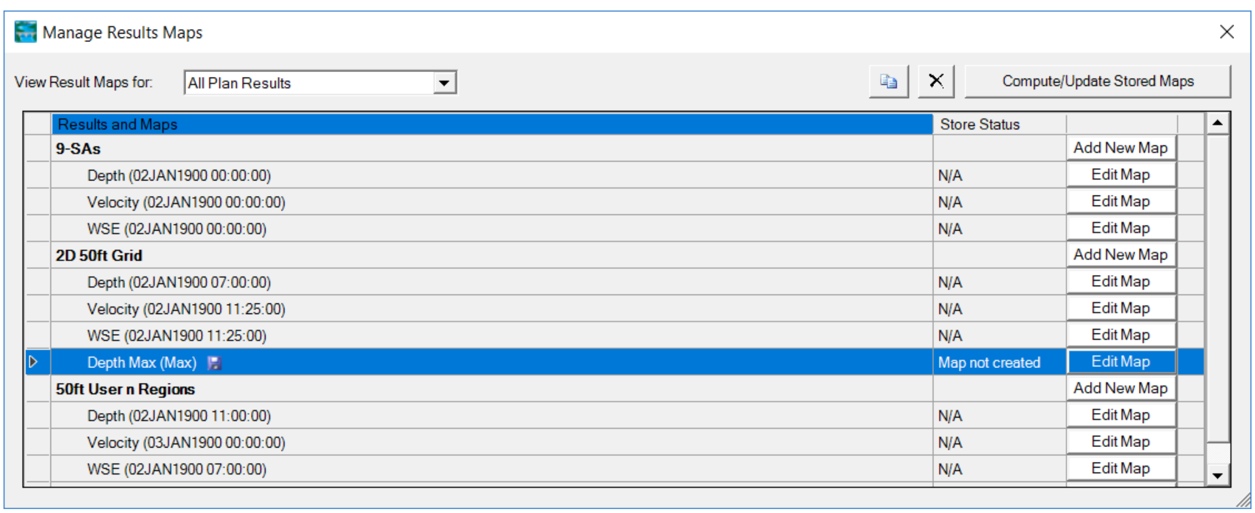


Figure 6-10. Results Mapping Window.

As shown in Figure 6-10, this editor will allow the user to create new map layers (**Add New Map**), as well as generate stored maps to a file (which can be used with HEC-FIA, or in a GIS, etc…). To add a new results map layer, press the button labeled **Add New Map…** for the desired Plan that the user wants to create a map from, this will bring up the window shown in Figure 6-11.

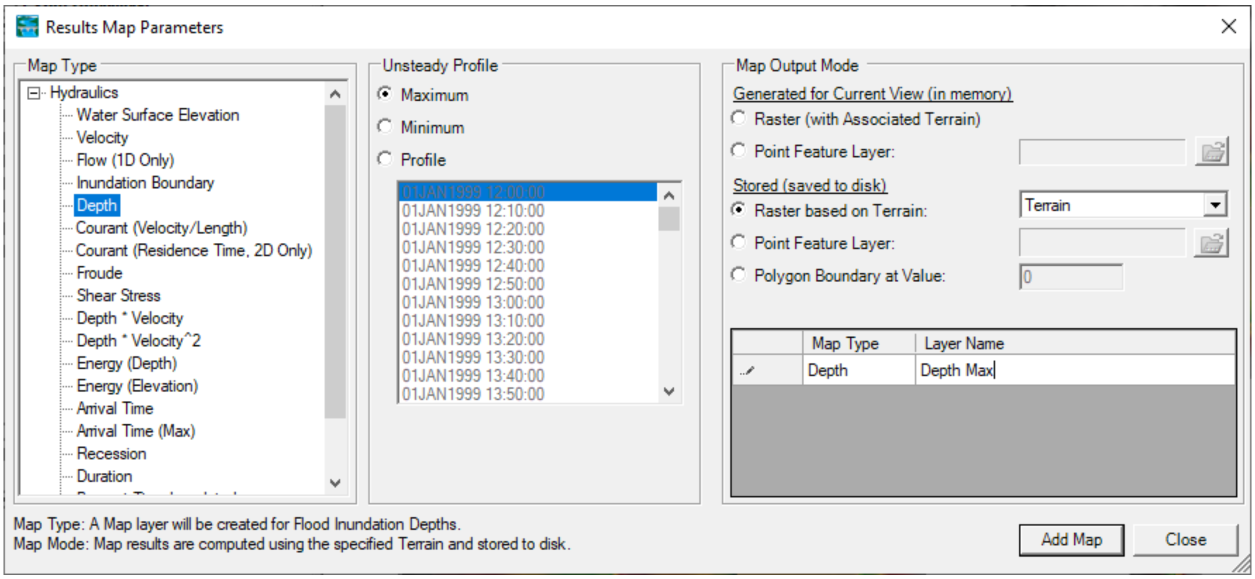


Figure 6-11. Add New Results Map Layer Window.

This window was described earlier under the section called **Adding Results Map Layers for Visualization**. This window can be used to create a Dynamic Map Layer, or create a Static Map by checking on one of the options under the **Stored (saved to disk)** section, then pressing the **Add Map** button at the bottom of the window (Figure 6-11). A new Layer will then show up on the **Manage Results Maps** window for the selected Plan, however, the status of that map will be labeled "**Map files are out of date**", which means the stored map has not been created yet.

To create the stored map, first highlight the layer(s) to be created, then press the button labeled **Compute/Update Stored Maps** in the upper right corner (Figure 6-10). This will start the process of creating/updating stored maps for all of the stored map layers that are out of date (only layers that are highlighted will be created/updated). When this process is complete, there will be a subdirectory within the project directory that is labeled the same name as the RAS Plan Short ID. This folder will contain the results in a gridded file format. RAS Mapper creates files in the GeoTIFF (Geospatial Tiff with .tif file extension) file format. The GeoTIFF is a file standard and can be used directly in ArcGIS 10 and higher and other software packages. The user can simply drag and drop the GeoTIFF files onto an ArcGIS project. The latest version of HEC-FIA (version 3.0 and above) also uses the GeoTIFF file format for incorporating HEC-RAS results for the computation of flood damages and potential life loss. Additionally RAS Mapper creates a file with the extension ".hdf" (HDF5 file). The .hdf file contains information for RAS Mapper about the GeoTIFF files. RAS Mapper also creates an XML file with the extension ".vrt". The .vrt file is supported by other GIS software for visualizing raster files. If the user has more than one terrain grid for the Terrain model, then RAS Mapper will also make more than one output depth grid. (i.e. it tiles them). The .vrt file is just a collection file that describes the other files and where they live spatially. If the user drags that file over to a GIS, or import it, then it brings in all the tiles as a single collection in one layer, and the user can have them all attributed the same.

Plotting Velocity

RAS Mapper has the ability to plot velocities spatially for both 1D river reaches and 2D flow areas. Velocity is plotted with a color palette reflecting the magnitude of the velocity. Users can change the color palette, as well as the magnitude range for plotting the colors. Velocity vectors, which reflect direction and magnitude of the velocity, can be added to the plot. Additionally, there is an option to turn on a particle tracing visualization, which allows for much greater understanding of the velocity flow field, in both magnitude and direction.

Turn on the velocity output layer by checking the box to the left of the layer. From there the user can make the **Velocity** layer the active layer (highlighted in pink) and select the **Animation** tool to animate the entire flood event. See an example 2D model Velocity plot in the Figure 6-12 below.

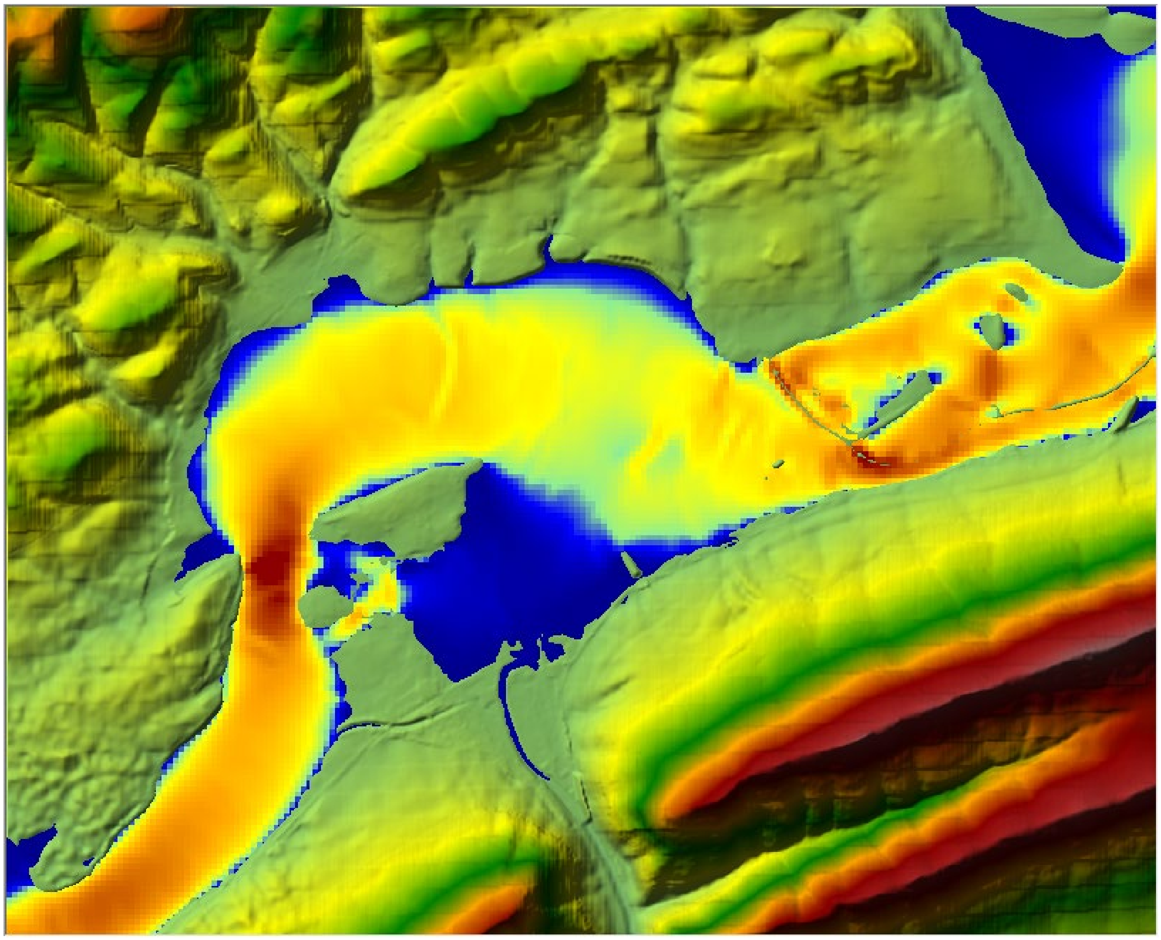


Figure 6-12. Example Color based Velocity Plot

In addition to color velocity plotting, RAS Mapper has the option to add velocity vectors and show particle traces on top of the map layers. To add velocity vectors, press the **Static Velocity Arrows** button  above the map window. This will turn on the velocity direction and magnitude arrows. To control the density of the arrows, select the **Velocity Setting** button  above the map window. This button will bring up the window shown in Figure 6-13 below:

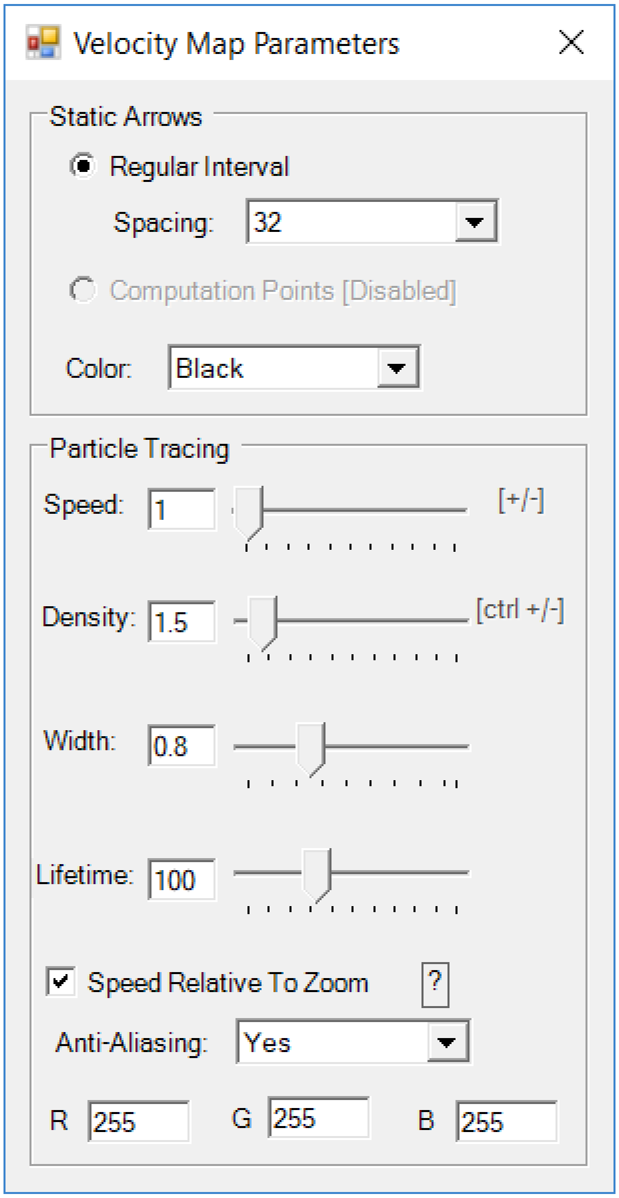


Figure 6-13. Velocity Map Parameters.

The **Velocity Map Parameters** settings window (see Figure 6=-13) allows the user to control the spacing between arrows by selecting a **Spacing** (pixel width for the spacing between arrows). When the arrows are turned on, they are displayed in the direction of the velocity. The magnitude of the velocity is reflected in the size of the arrows (i.e. larger arrows equates to higher velocity). Show in Figure 6-14 is a velocity plot with magnitude/direction arrows turned on.

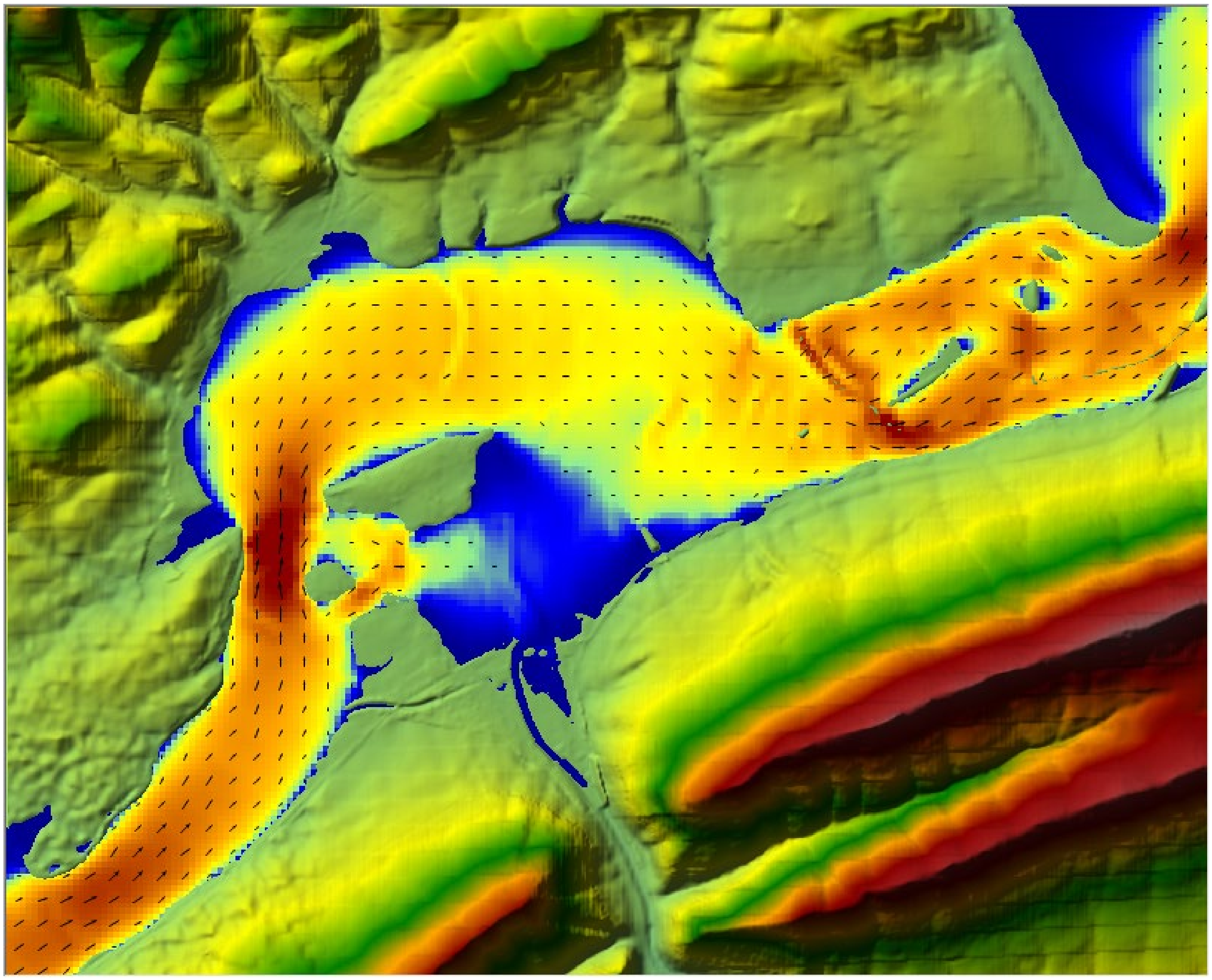


Figure 6-14. Example Velocity Plot with Color and Direction/Magnitude Arrows.

Another extremely cool option for velocity plotting is the option called **Particle Tracing**. When this option is turned on, the user will see what appears to be particles of water moving through the flow field. This is a visualization of water particle movement to improve the understanding of the velocity and the direction of the flow. To turn this option on, press the **Particle Tracing** button  above the map window. Once this option is turned on, from the **Velocity Map Parameters** window (Figure 6-13) the user can change the parameters that control the particle tracing visualization. These parameters are: **Speed** (Speed the particles move. The speed is a relative speed, it is not the actual speed of the particles); **Density** (density of the particles); **Width** (how thick they appear); **Lifetime** (how long a particle trace will last); **Speed Relative to Zoom** (this automatically changes the speed with the zoom level); and **Anti-Aliasing** (**Yes** provides smoother lines for the particle traces, but takes more compute power. **No** produces particle lines that are not as smooth, but takes much less compute power) The user can toggle **Particle Tracing** on and off by using the **F5** key (The plot window must be the active window when using the F5 Key to turn particle tracing on and off). Additionally the user can increase or decrease the speed of the particles (to improve the visualization), by using the **+ key** to speed up the particles and the **– key** to slow down the particles. The user can also change the density of the particles. Holding down the **Cntrl key** and then pressing the **+ key** will increase the density of the particles, while holding down the **Cntrl key** and then the **– key** will reduce the density of the particles.

The particle tracing visualization option can be turned on over the top of the colored velocity plot, or over top of the depth and elevation plots. This option is extremely helpful in visualizing where water is going, and the relative magnitude of the velocity. Try it out; it's really fun and informative!!! Shown in Figure 6-15 is an example plot with the velocity particle tracking option turned on, and being displayed on top of a depth layer plot.

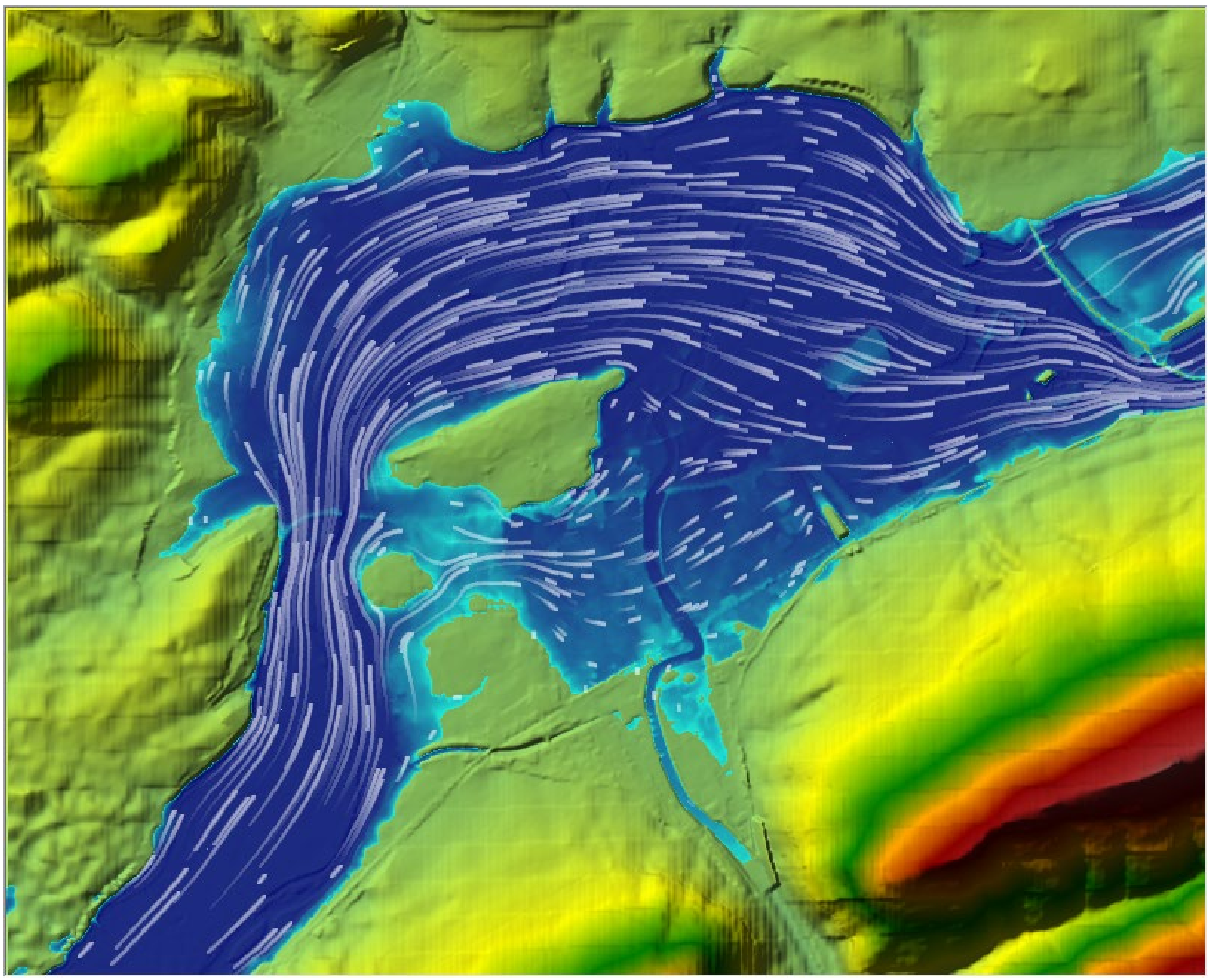


Figure 6-15. Example of the Particle tracing visualization option on top of a depth layer.

Querying RAS Mapper Results

When a **Map Layer** is being displayed in the map window, the results of that map can be queried to display the point value by simply moving the mouse over the map layer. To do this, first click on the results map layer, to make it the active layer. When a results map layer is being displayed, clicking on that layer will turn the label to magenta in the layer directory tree. Once the desired results layer is turned on, and it is set to the active layer, move the mouse pointer over that layer, and everywhere the user moves the mouse pointer the numerical value of that result will be displayed next to the mouse pointer. See the example below in Figure 6-16.

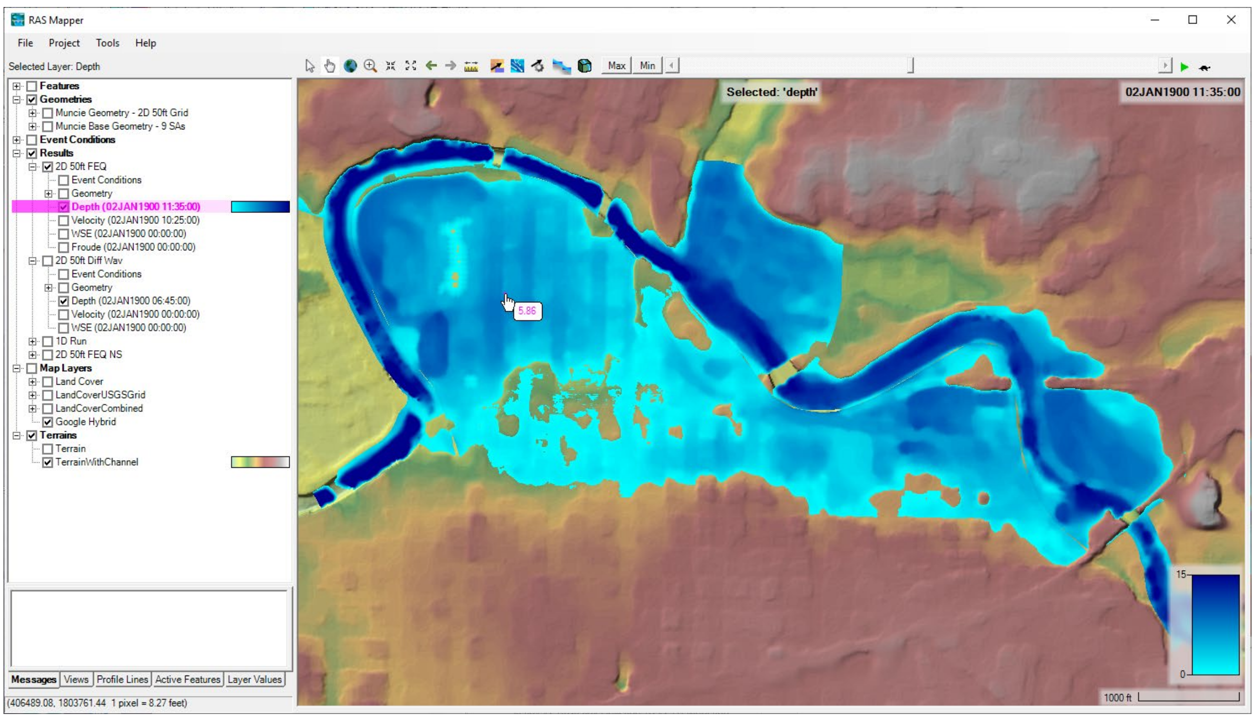


Figure 6-16. Example of Querying a value from the active map layer.

Time Series Output Plots and Tables

When **Results Layer(s)** are turned on for display, the user can also get time series plots and tables for those results layers. If a results map layer is displayed, move the mouse pointer over that graphic layer and then right click the mouse, a pop up window will appear with additional output options for displaying time series plots. For example, if the "Depth" results map layer is turned on, right click on that map window over that layer and an option for plotting a "**Unsteady Time Series**" will be available in the popup window. When this option is selected, there will be a sub menu for selecting the available variable types (i.e. Depth, WSE, and Velocity). Only the map layers that are turned on will be available to plot. Also, if more than one of the same map layer type is turned on for two or more Plans, then all of those map layers will show up on the same time series plot (Figure 6-17).

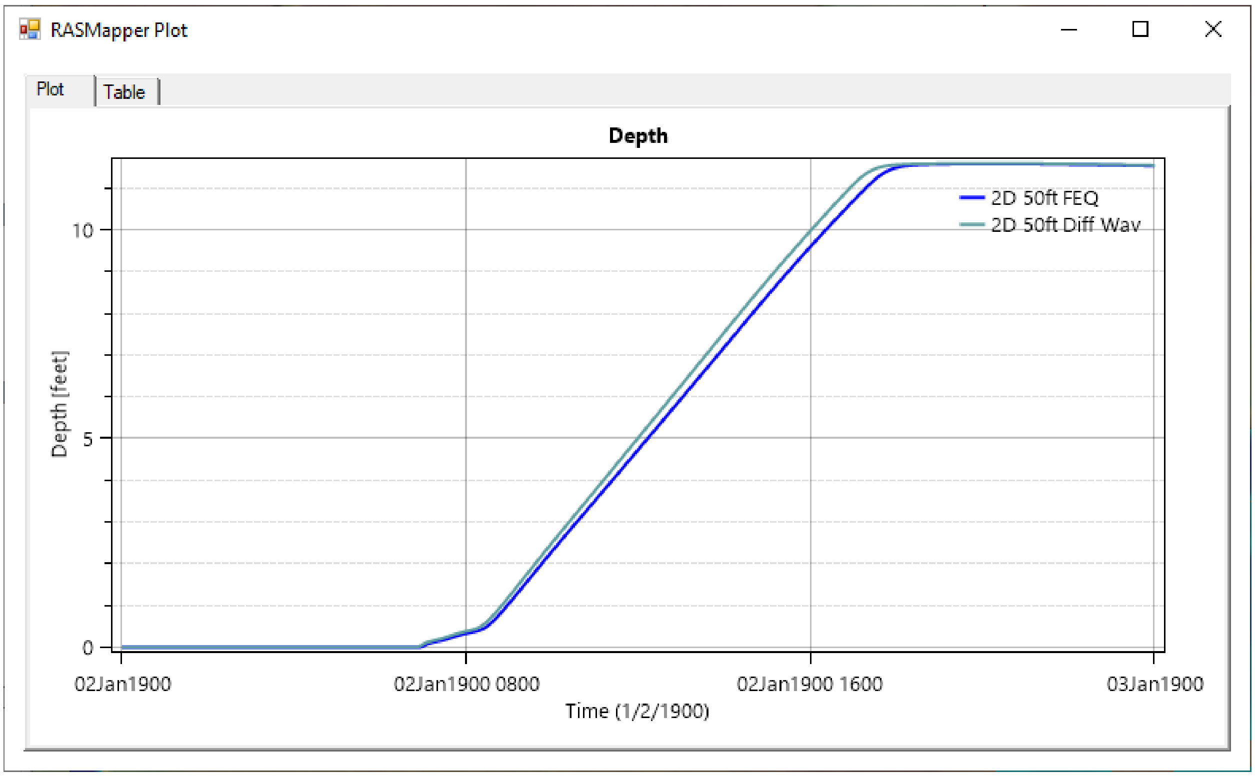


Figure 6-17. Example Time Series Plot of Depth from two different Plans.

If 2D flow areas are in the model, additional 2D model results in the form of time series plots and tables are available. To get 2D model specific results, first turn on the **2D flow area** grid, from the Geometry layer of the desired plan, listed in the **Results** layer. When the 2D flow area grid is turned on, right click the mouse over top of it, the popup menu will show options for plotting (Figure 6-18) the following results time series: Cell Water Surface Time Series; Face Point Velocity Time series (this is a point velocity of the closest Cell Face Point when selected); Face Perpendicular Velocity Time Series (the component of the velocity that is perpendicular to this face); Face flow (flow across this face in time); Face Shear Stress (the average shear stress across the cell face that is closest to the mouse pointer when selected); and the Property Tables (the pre-computed cell elevation vs volume and face property tables (elevation vs area, wetted perimeter, and roughness) that are used in the solution of the equations). An example 2D cell water surface elevation time series plot from RAS Mapper is shown in Figure 6-19.

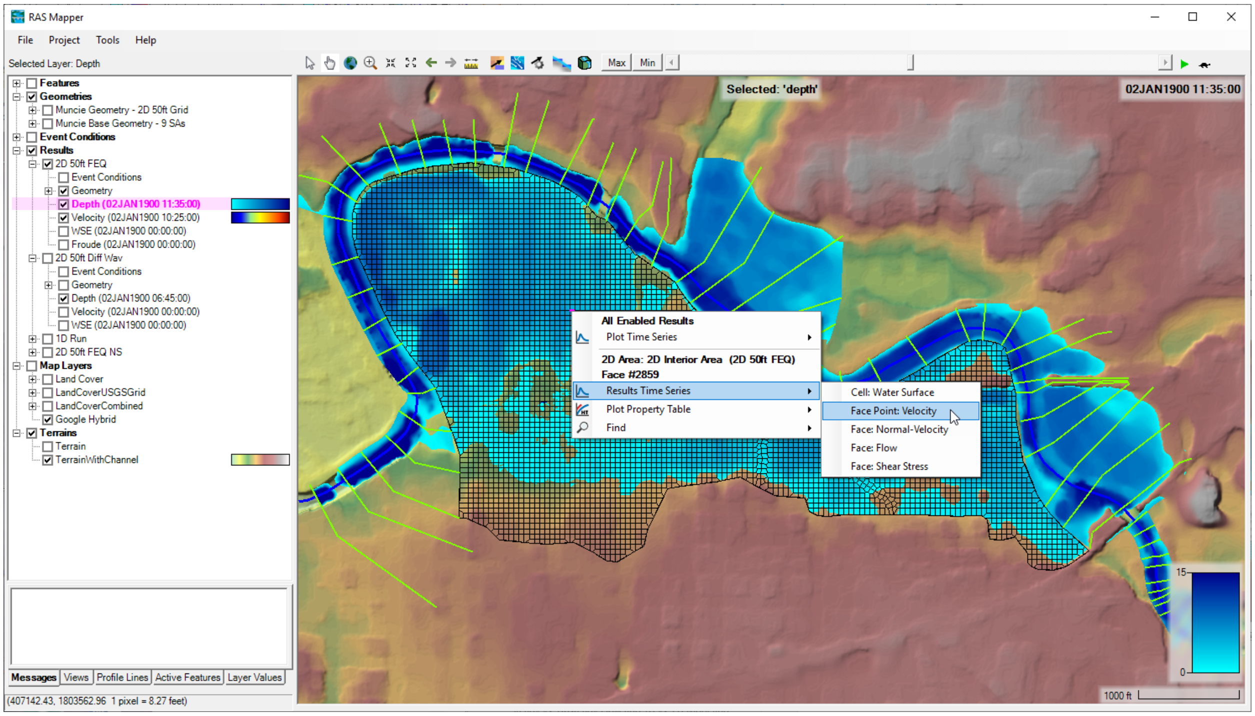


Figure 6-18. Example showing options for displaying 2D Model Output Time Series Results.

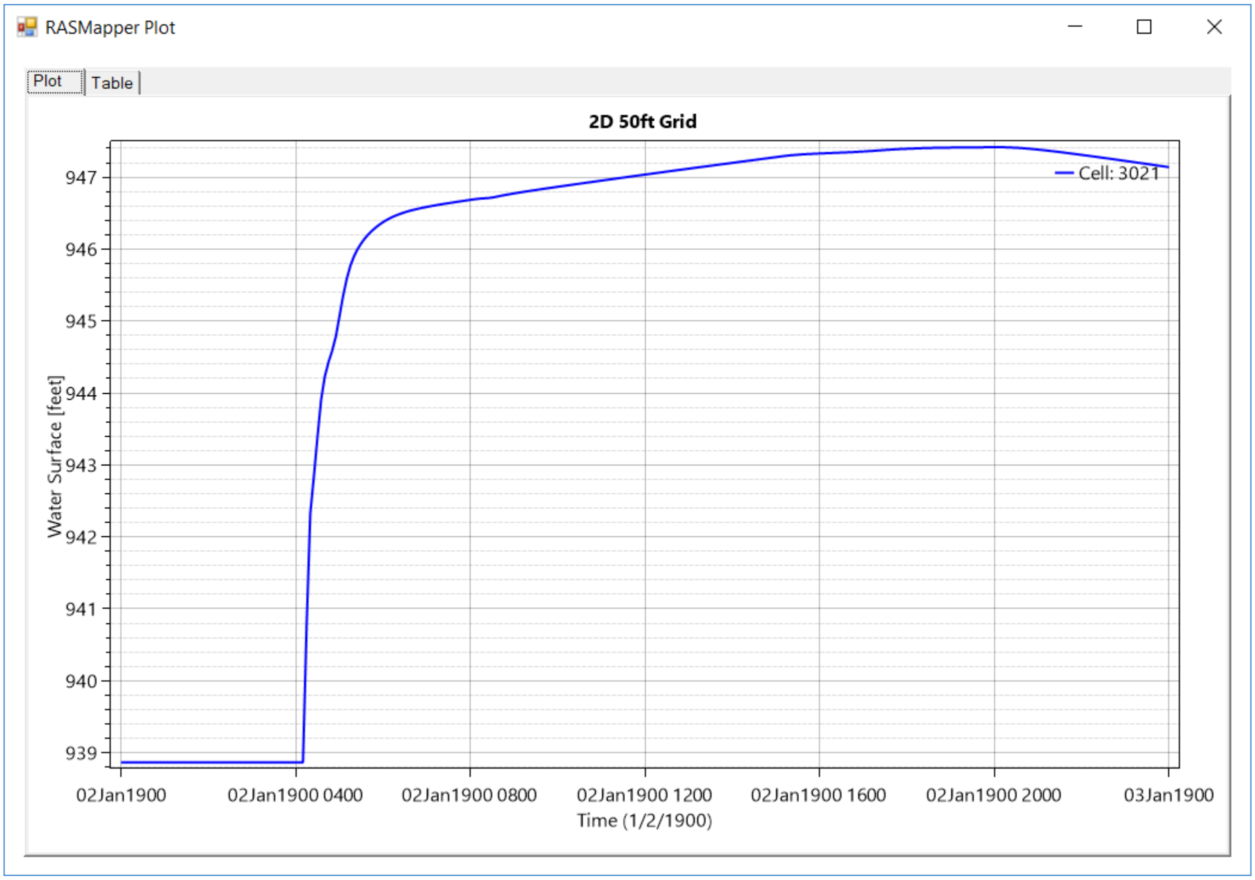


Figure 6-19. Example Time Series Hydrograph Plot of a 2D cell.

Once a time series is plotted from RAS Mapper, there is also the option to display the results in a table. Tabular results are displayed by selecting the **Table** tab on the time series plot. Users can highlight data in the table and use **Ctrl-C** to copy the highlighted information to the Windows Clipboard. Also, when viewing the plot, right clicking brings up a popup menu with the options to "Copy Values to Clipboard" (which copies all of the data) and to "Zoom to the Full Extent" of the data. Additionally, the mouse wheel can be used to Zoom In and Out on the plot; holding down the **Ctrl** key and using the mouse allows for measuring on the plot; and holding down the "**Z"** key and using the mouse allows the user to draw a Zoom Window; pressing the **Esc** key will zoom to the full extent.

Profile Lines

HEC-RAS Mapper has the option for user to draw a line on the map, give that line a name, then use that line to plot whatever results are turned on underneath the line. To create or modify Profile Lines, expand the Features Layer at the top of the tree and turn on the **Profile Lines**. Right click on the Profile Lines layer and select **Edit Layer** to begin editing. The HEC-RAS Mapper editing tools will show up allowing you to select the **Add New** or **Edit Feature** tool. If you select "Add New Feature" tool you can draw a new Profile Line. If you select "Edit

Feature" tool, you can edit existing Profile Lines. Another way to add new Profile Lines is by selecting the **Profile Lines** tab at the bottom left of the HEC-RAS Mapper window. Once this Tab is selected there is a container for the user defined profile lines, to add a profile line select the **+** (add a profile line) button, and to delete a profile line select the **X** (delete profile line) button. The line can be drawn as a multi-point line anywhere on the map. Once the line is drawn you will be asked to give it a unique name. You can change the name by right clicking on the name and selecting **Rename**.

To plot computed results underneath a user defined profile line, first click on the profile line name in the container box on the lower left hand side of HEC-RAS Mapper (See Figure 6-20). This will activate that line. Next, right click on the line, this will bring up a popup menu, of which one of the options will be labeled "**Profile Line: Profile line 1**". Below that will be an option called "**Unsteady Profile Plots**", then a sub menu of that will contain all of the available data layers that can be plotted as a profile plot. Select the layer you want, for example WSE (Water Surface Elevation). Once a data layer is selected then the profile plot for that data layer will show up in a separate window (Figure 6-21).

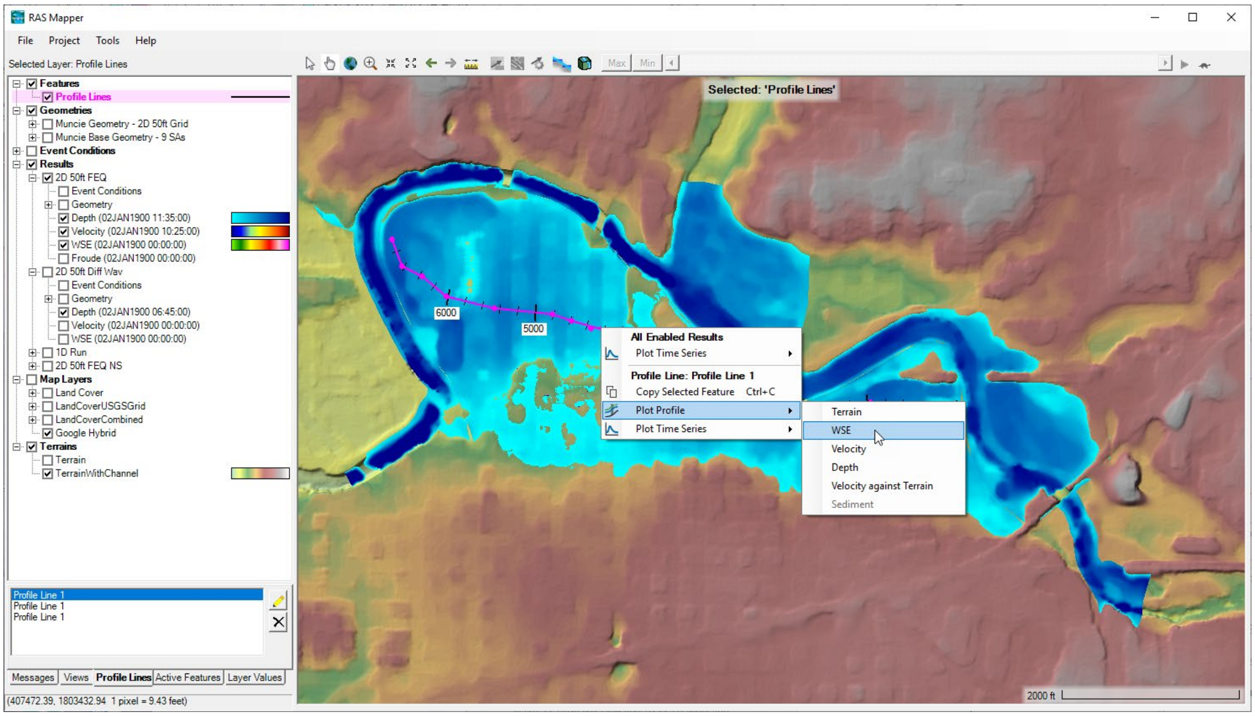


Figure 6-20. Profile Line turned on and selected for plotting options.

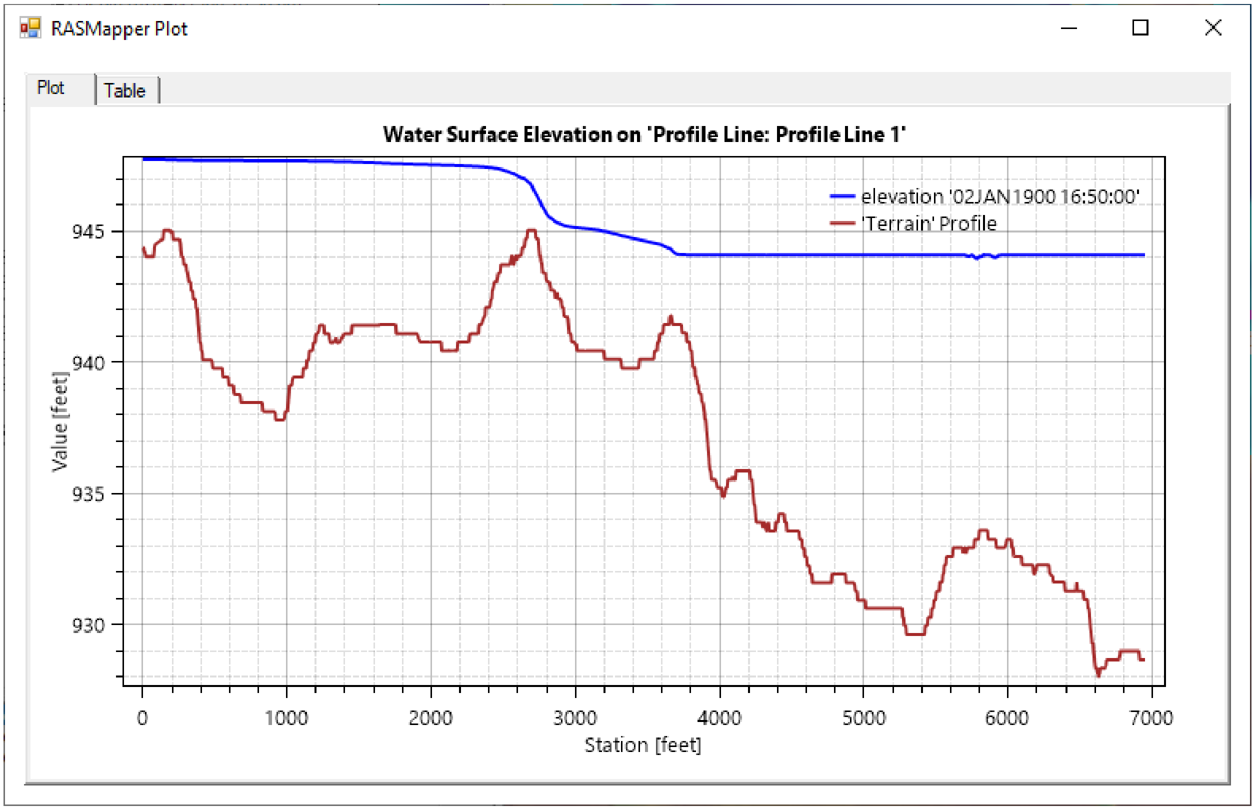


Figure 6-21. Example Profile line Plot of Water Surface Elevation (WSE)

Additionally, Profile Lines can be used to plot **Time Series** Output. For the Example shown in Figure 6-20, there is a second Profile line called "FlowLine1". This line was drawn perpendicular to the direction of flow, for the purpose of plotting time series data for the flow crossing that line. If the user right clicks on a profile line, there is an option called **Plot Time Series**. Sub menu options of this menu include: Flow; Volume Accumulation; and Rating Curve. An example Flow time series is shown in Figure 6-22.

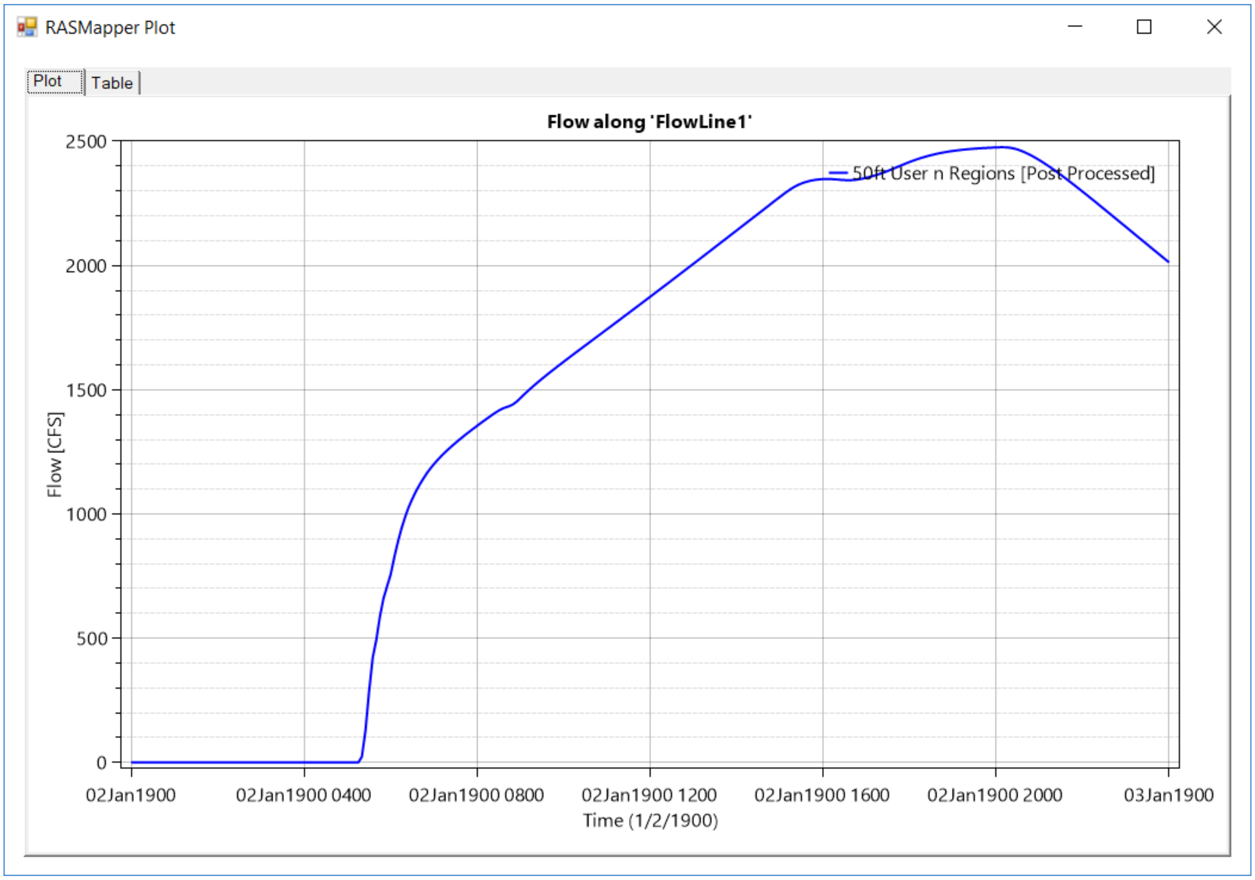


Figure 6-22. Example Flow Time Series Plot.

User Defined Views

HEC-RAS Mapper has the option to allow the user to store specific Views. Views are the current extents of the map view window. This can be a very handy feature when you have a large model. The user can zoom in to an area of interest, then store that view with a user defined name. Then go to another area of interest, and store that view, etc…

To use the **Views** option, select the **Views** tab from the lower left portion of the HEC-RAS Mapper window. Zoom into the location of interest, and the exact extents for which you want to save as a view. Press the +key to save the view. The user will be asked to give a unique name for each view saved. Once you have saved one or more views, then just click on a particular view name, and the map will automatically transition to that view extents.

Background Map Layers

HEC RAS Mapper has several options for bringing in other data layers/formats to be used as background maps below the computed results. For example, the following file formats are supported: web imagery; ESRI Shapefiles; VRT (Virtual Raster) files; GML (Geospatial

Markup Language) files; and many other file formats (115 file formats are currently supported). The GML file type supported is the "Simple Features version 3" format.

To use the Web imagery capability, first set the spatial reference system for the project. Select the **Project | Set Projection** menu item from the RAS Mapper menu bar. When this option is selected the window shown in Figure 6-23 will appear.

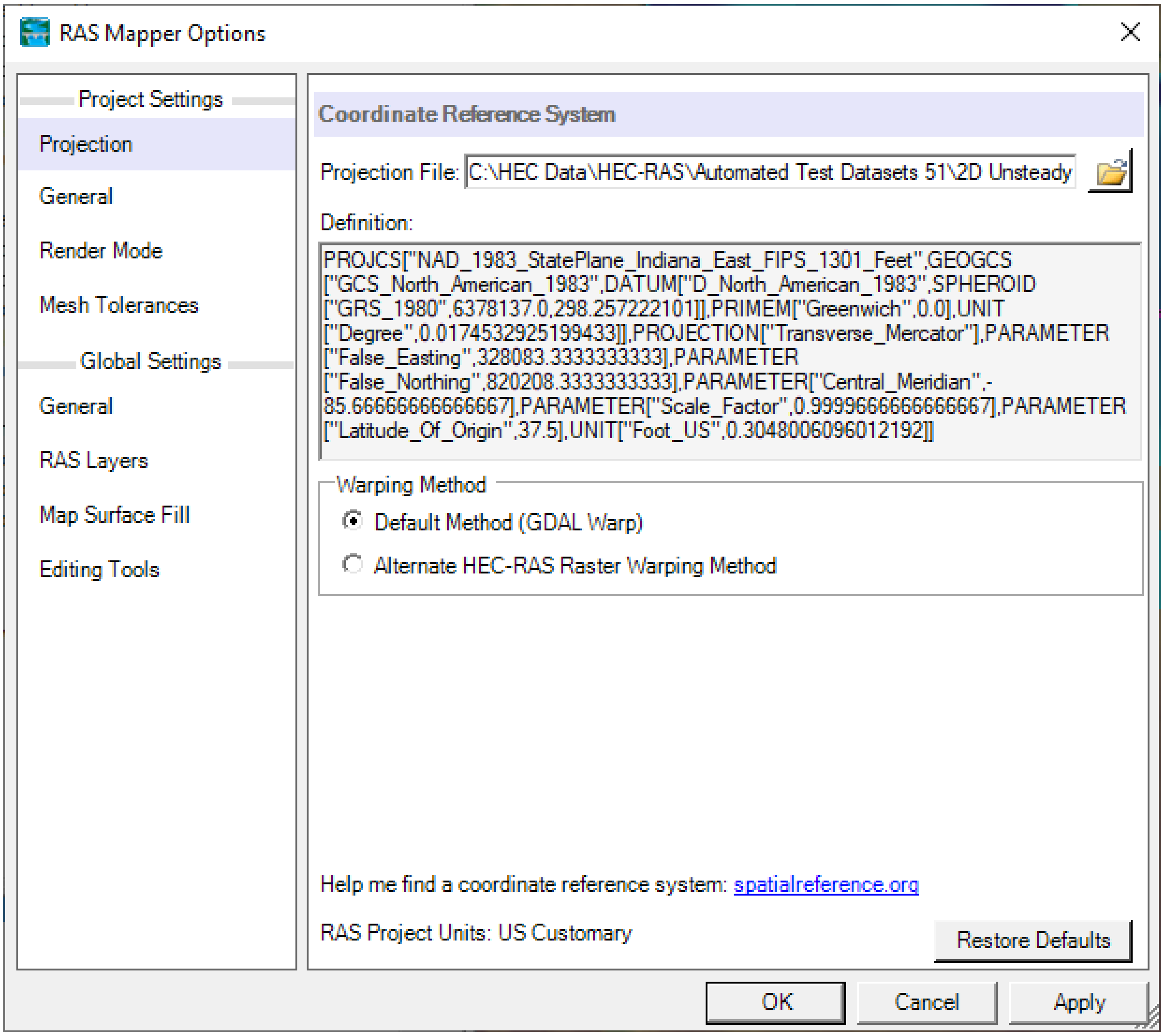


Figure 6-23. Editor to set the RAS project's spatial reference system.

To set the spatial reference system (coordinate system), browse and select an existing ".prj" file (ESRI projection file) that contains the correct coordinate system. If ArcGIS 10 or lower is installed the user can browse to the ArcGIS directory that contains a listing of all the available coordinate systems and select the appropriate one. The default directory path where ArcGIS 10 or earlier stores a listing of all the available coordinate systems is listed in the "Projection File" text box, shown in Figure 6-23. For this example, "NAD 1983 State Plane Indiana East" was selected.

Web Imagery:

Once the correct coordinate system is set in RAS Mapper, the **Web Imagery** option can be used by selecting it from the **Tools** menu (or right click on **Map Layers** and select **Add Web Imagery Layer**). When this option is selected, a window will appear with the list of available web services for downloading web based imagery and map layers (Figure 6-24). Select one of the available options and press the **OK** button. When a web service is selected, RAS Mapper will send the limits of the currently viewed area to that server and request the imagery/map data. Once the data is received it is displayed on the screen. This data is not saved to the hard disk. It is only for real-time display of the imagery and map layers. When using this option, every time the user zooms in or out, or pans, it makes a new request for the data, receives it, and then displays it. See an example of Web imagery used as a background layer in Figure 6-25.

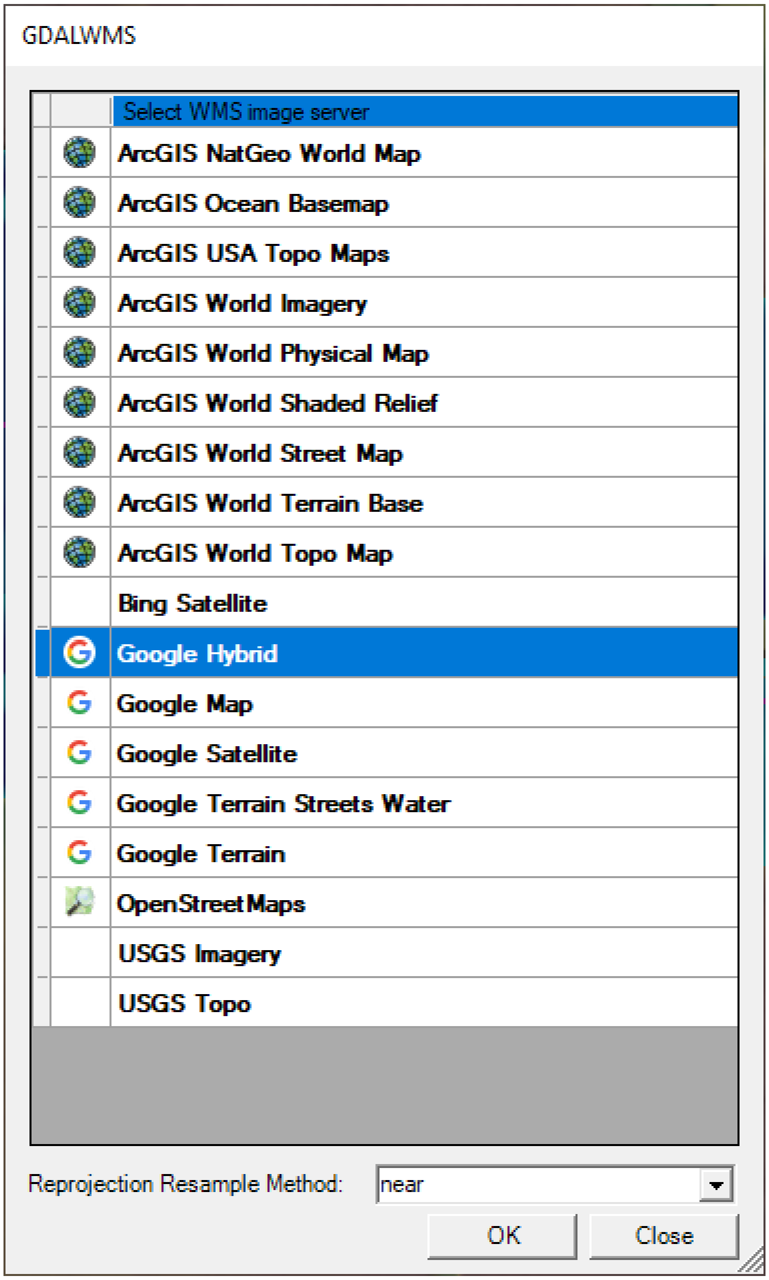


Figure 6-24. Web mapping services available in RAS Mapper.

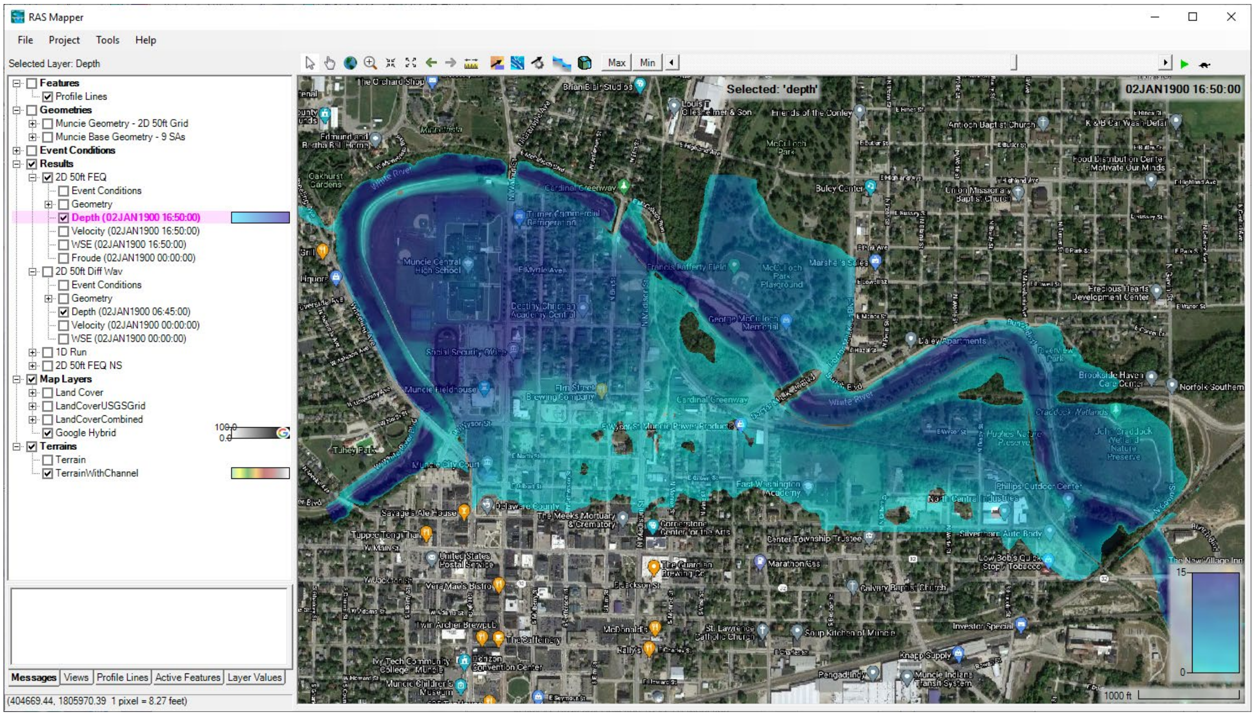


Figure 6-25. RAS Mapper with background Web imagery loaded with an inundation depth grid overlaid.

Other Map Layer Formats

As mentioned previously, in addition to web imagery, RAS Mapper supports many different file formats for displaying map layers. Some of the more popular formats are: ESRI Shapefiles; GeoTIFF; MrSID; JPEG; Arc/Info Grids; Bitmaps; NetCDF; USGS ASCII DEM; etc…

To use this option, right click on **Map Layers**, then select the **Map Data Layers/Add Existing Layer** option. The file browser window will appear, allowing the user to navigate to the desired file and select it. See Figure 6-26 below:

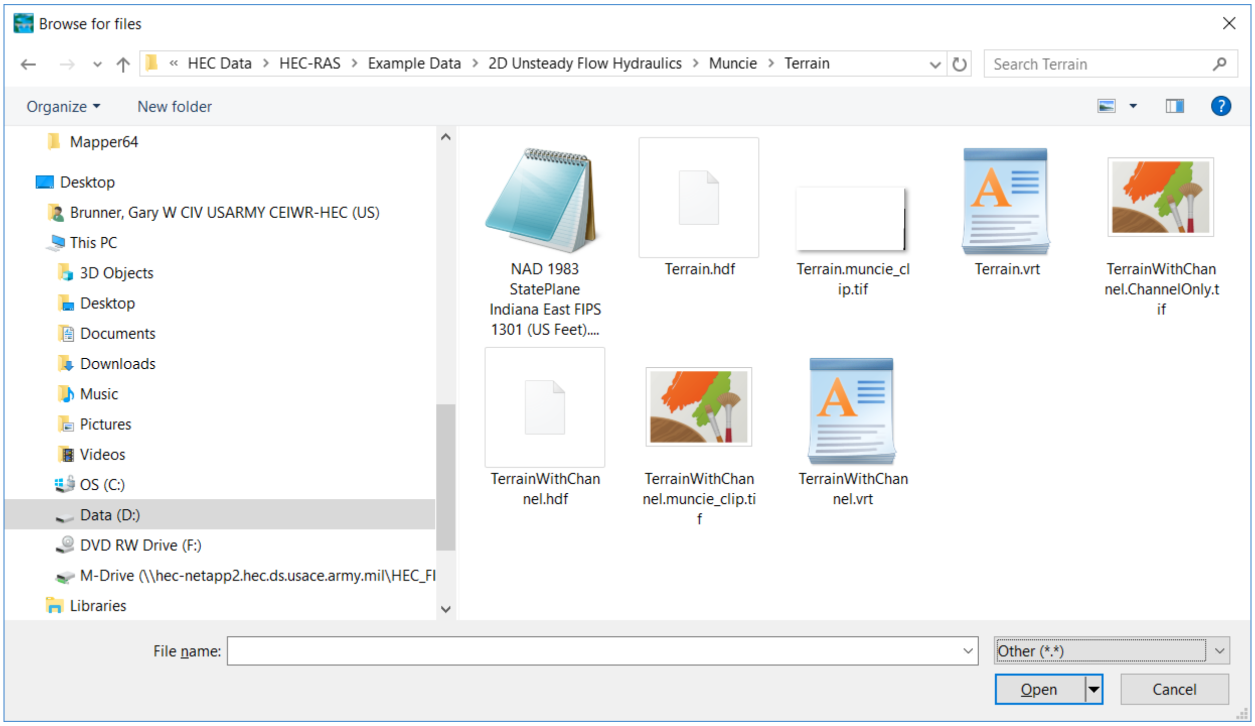


Figure 6-26. Example File chooser for bringing in Map Layers to be used for background display.

National Levee Database

The last tool to discuss is the link to the National Levee Database (NLD). **Note: This option is only available to U.S. Army Corps of Engineers Employees, using an ACE-IT approved computer.** If the user selects **Import NLD** from the RAS Mapper **Tools** menu, then a window will appear as shown in Figure 6-21. The user can select to query all the levees and floodwalls that are within the current view extents (the area shown on the screen when fully zoomed out), or within the view of selected map layers. Once the **Query** button is pressed, the software calls the NLD for all of its information. The NLD will send a list back to HEC-RAS and a window will appear on the screen with that list of levees/floodwalls (Figure 6-27).

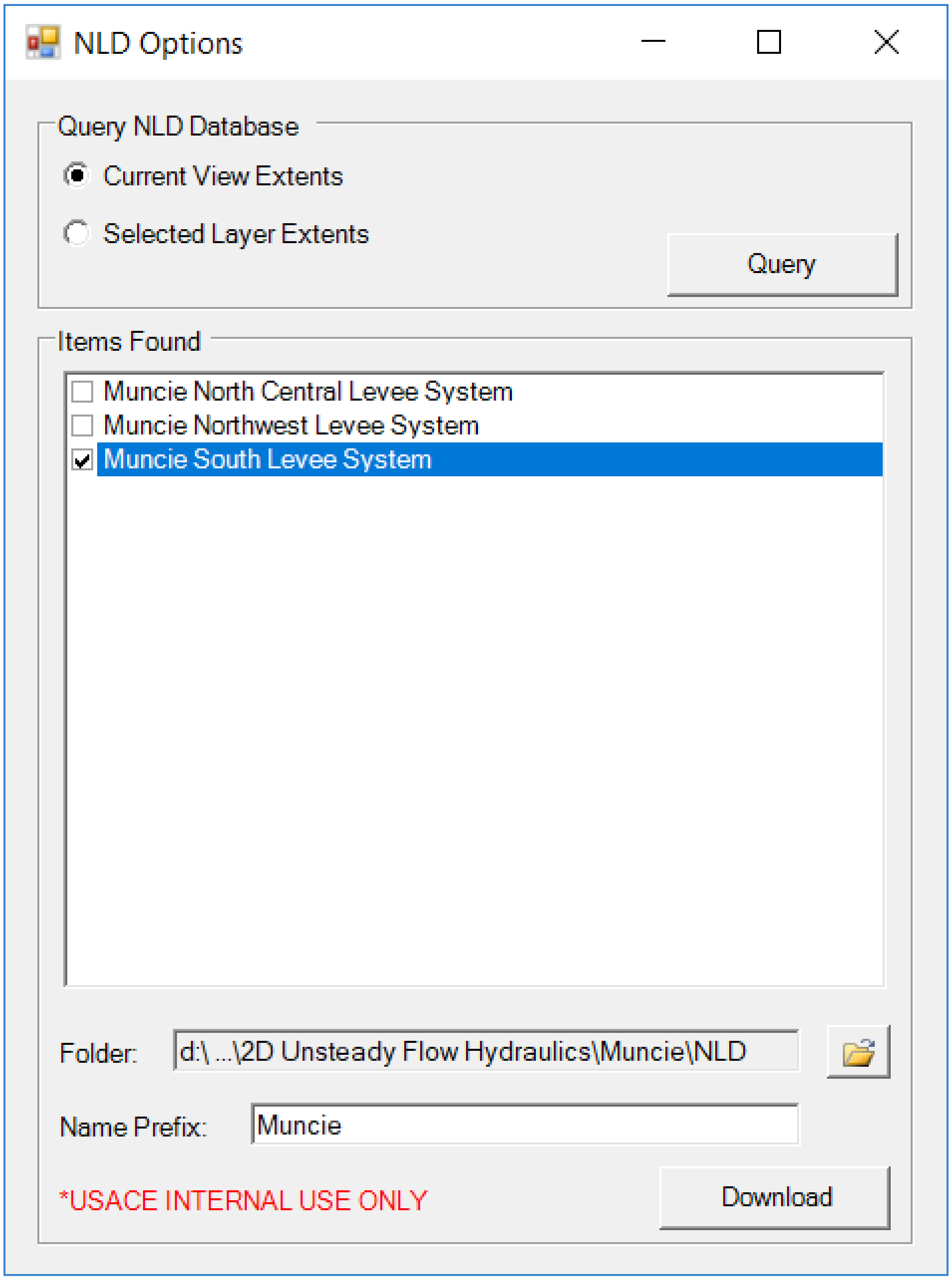


Figure 6-27. List of levees and floodwalls sent to RAS from the NLD.

The user then selects what information they want. Finally, the user presses the **Download** button, and then they will be asked to select a directory to store the data in. Once a directory is selected, the software will download a levee 3D centerline, a floodwall 3D centerline, and a polygon of the protected area for each of the levees listed in the window shown in Figure 6-27. Currently the information is stored in a ".gml" file format (Geospatial Markup Language). The next step for HEC-RAS in using this data is to automate the process of converting it into an HEC-RAS Lateral Structure to represent the levees and floodwalls, as well as use the protected area for 2D flow and/or storage area boundaries. This is not available in yet.

2D Output File (HDF5 binary file)

The Output for the 2D flow area computations, as well as most of the 1D output, is contained in a binary file that is written in the HDF (Hierarchical Data Format) file format (The HDF Group, 2014). Similar to an XML document, HDF files are self-describing and allow users to specify complex data relationships and dependencies. However, unlike XML files, HDF files can contain many different types of data and all are stored in an efficient binary form. Furthermore, HDF allows direct access to different parts of the file without first having to parse the entire contents. Specifically, we are using the HDF-5 file format (Version 5 of HDF).

To view and or use some of the output outside of the HEC-RAS interface, the user can access it directly from the HDF files.

A Free HDF file viewer can be downloaded from The HDF Group at the following location:

https://www.hdfgroup.org/downloads/hdfview/

Download and Install the Windows 64-bit version. Once the HDF file viewer is installed you can open the files, view their contents, display tabular data, and even plot results. Shown in Figure 6-28 is an example HDF file output from an HEC-RAS 1D/2D model run. As shown in Figure 6-28, the user can get to the Unsteady flow output for the 2D areas (as well as 1D objects) by drilling down through the directories… **Results/Unsteady/Output/Output Blocks/Base Output/Unsteady Time Series/2D flow areas/**, then clicking on the folder name of the 2D flow area and the user can see all the output that was computed and stored for that specific 2D flow area. Currently available HDF file time series data that is output for a 2D area is:

1. Water Surface: Water surface elevation for each cell (ft or m)
2. Face Velocity: Normal face velocity (the component of the velocity perpendicular to that face) (ft/s or m/s)
3. Time Step Current computational time step at that point in time.

In addition to the Unsteady Time Series output, there is also Summary Output. The Summary Output includes:

1. Maximum Face Velocity: Maximum face velocity in the entire 2D area for each time step (ft/s or m/s)
2. Maximum Water Surface: Maximum water surface in the entire 2D area each time step (ft or m)
3. Minimum Face Velocity: Minimum face velocity in the entire 2D area for each time step (ft/s or m/s)
4. Minimum Water Surface: Minimum water surface in the entire 2D area each time step (ft or m)

NOTE: The node velocities (Node X Vel and Node Y vel) are not automatically written to the HDF output file. HEC-RAS Mapper does not need these velocities to perform any of the mapping (It can compute the node velocities on the fly from the Face Normal velocities). If you want these velocities output to the HDF file you must go to the "**Unsteady Flow Analysis**" window, then select "**Options**", then select "**Output Options**". From here select the Tab labeled "**HDF5 Write Parameters**". Then check on the option labeled "Face Point (Node) Velocities".

There are several optional variables that can be written to the HDF output file. These include: Cell Depth; Cell center velocity; Cell flow balance (inflow minus outflow); Cell and Face Eddy Viscosity values; Face flow; Face WSEL; Face Tangential Velocity (both sides of each face); Face Shear Stress; and Face Point (Node) velocities.

As you can see by looking at the file format, there is also time series output in this file for the 1D objects (cross sections, storage areas, lateral structures, inline structures, etc…). Over time all of the HEC-RAS binary output will be switched to HDF file format. For now the traditional ".O##" files are still written to and used for the post processing output, which users can view from the graphics and tables in the HEC-RAS interface. Even after HEC-RAS has switched over to using HDF, HEC-RAS will still fully support DSS (import of data and user selected output of results).

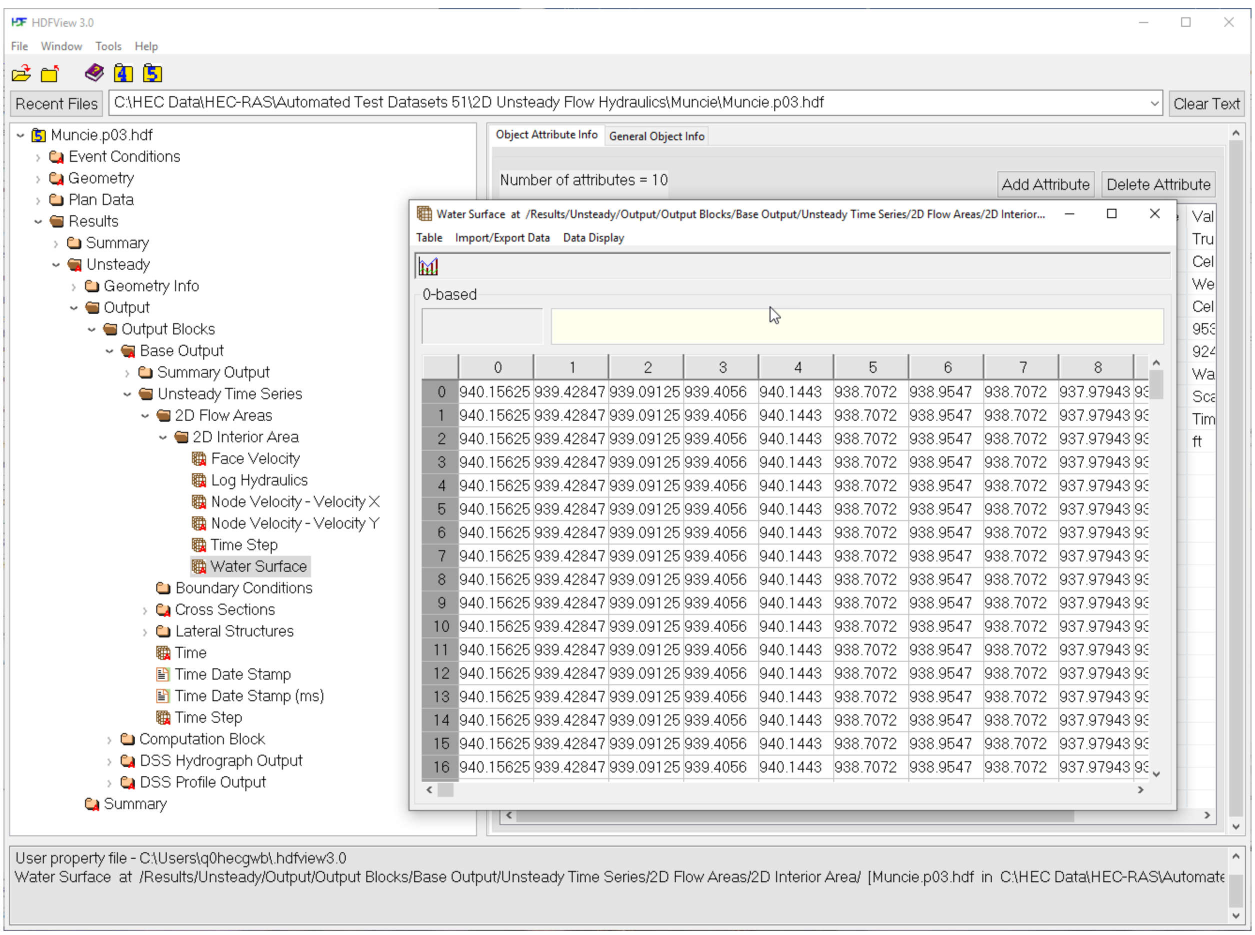


Figure 6-28. Example HDF File Output from HEC-RAS 1D/2D Model Run

3D Perspective Plots

Another type of graphic available to the user is the 3D Perspective Plot. The 3D plot is a 3-dimensional plot of the terrain on the computed results (Depth, water surface, velocity, etc.).

The HEC-RAS 3D Viewer was developed to help engineers convey hydraulic modeling results to decision-makers. The HEC-RAS 3D Viewer is accessed from either the HEC-RAS program or inside RAS Mapper. The 3D Viewer provides a three-dimensional visualization of HEC-RAS simulation results and terrain data.

To access the 3D Viewer through the HEC-RAS program interface, go to **View | 3D View ...** menu item or press the **3D Viewer** button, shown below.

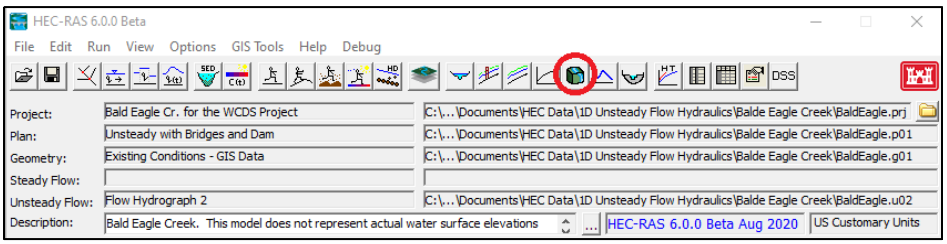


Figure 6-29. HEC-RAS Main Window with 3D Viewer Button.

To access the 3D Viewer through RAS Mapper, press the **3D Viewer** button, shown below.

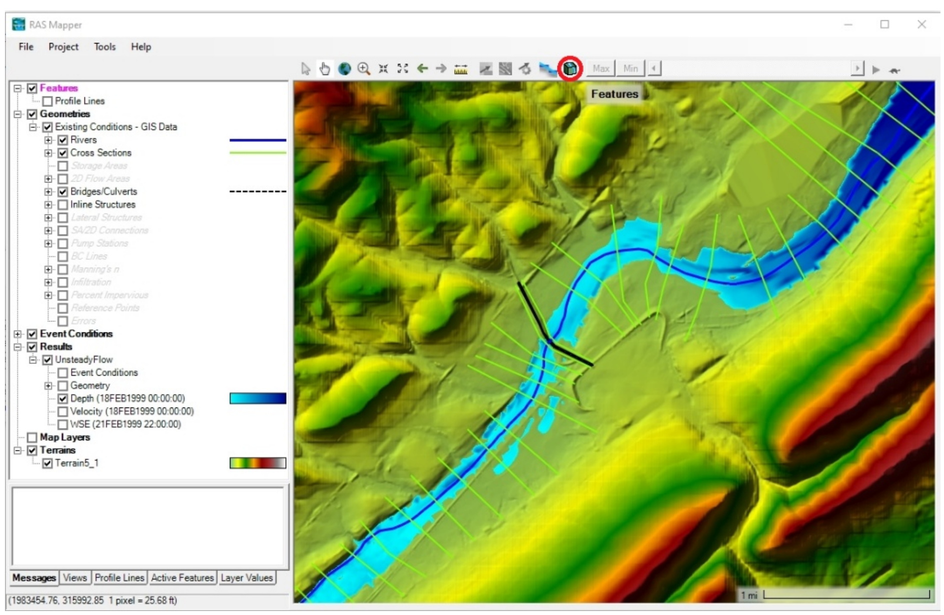
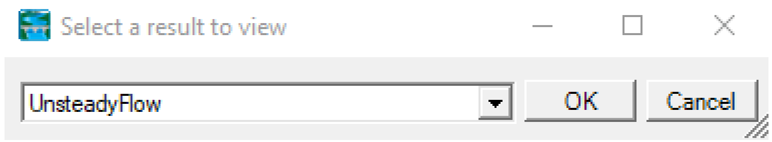


Figure 6-30. 3D Viewer Access Button in HEC-RAS Mapper.

Then select the result to show in the 3D Viewer



Another way to access the 3D Viewer is by right clicking on a particular result and selecting the **View Result in 3D** menu item.

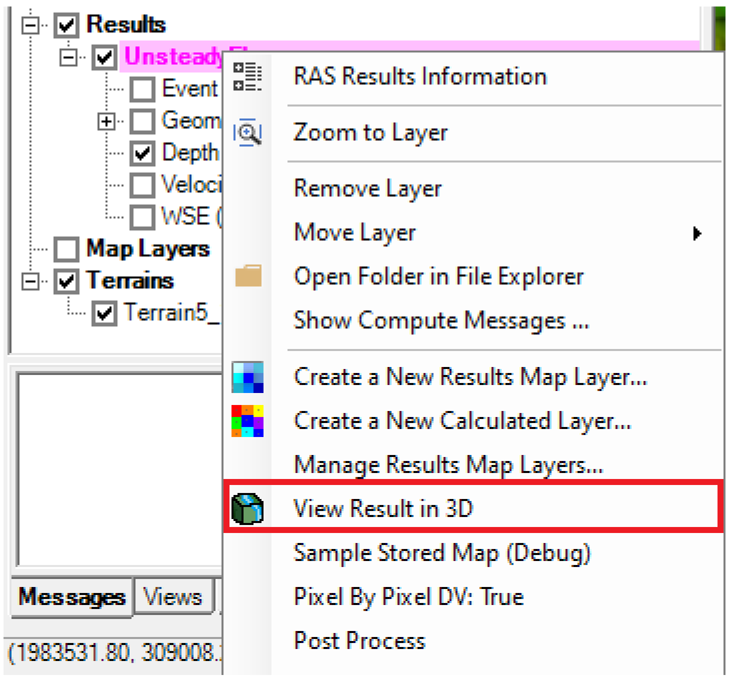


Figure 6-31. Selecting the 3D Viewer from HEC-RAS Mapper Menu.

The last way to access the 3D Viewer is by right clicking a particular result map and selecting the **View Map in 3D Viewer** menu item.

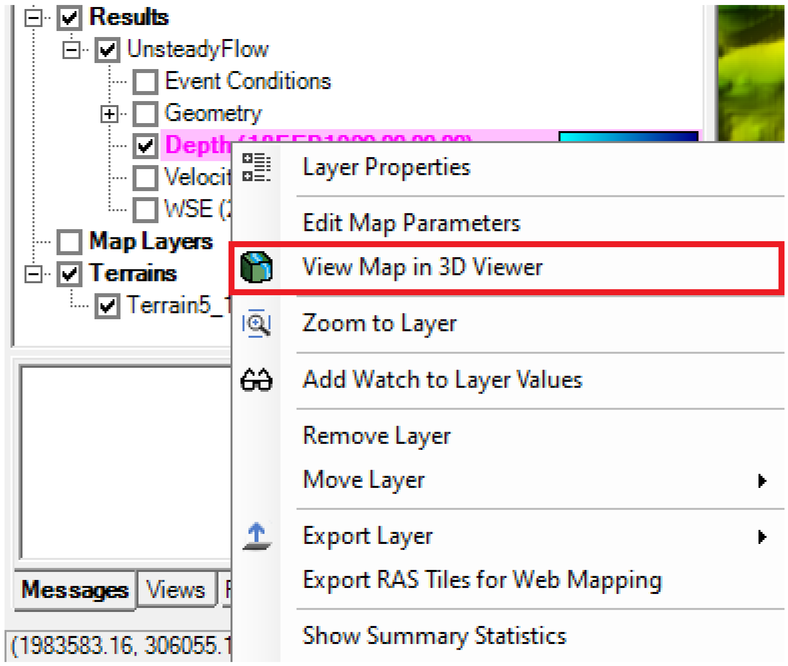
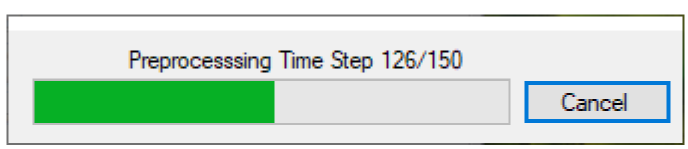


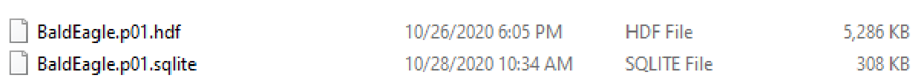
Figure 6-32. Accessing 3D Viewer from a Result Map Layer.

Pre-processing Results for 3D Viewer

Performing any of the various ways to access the 3D Viewer will bring up a pre-proccessing window if this is the first time you have run the 3D Viewer or if you have cancelled pre-processing the last time you opened this result in the 3D Viewer.



The 3D Viewer has to do much more processing compared to RAS Mapper to show a time step in the simulation. Pre-processing offloads the processing to a file in the same directory as the result file that was selected. It will be named the same except the extension will be sqlite.



Pre-processing will make subsequent loading for this result to be a smoother experience. It will also make playing the results animation smoother. Pre-processing is optional, press the Cancel button if you do not wish to pre-process at this time.

3D Viewer

The 3D Viewer interface, shown below, is comprised of a menu, a toolbar, a mini-map and the view itself.

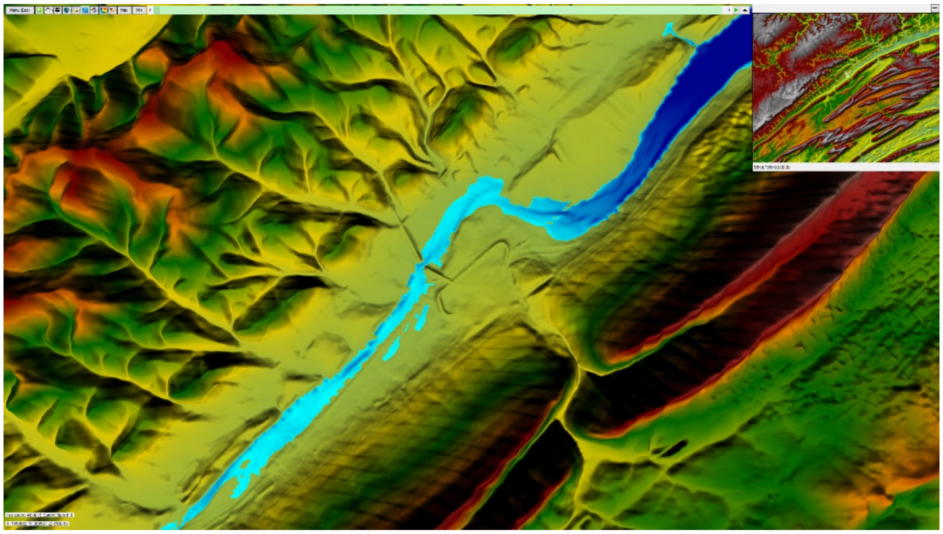


Figure 6-33. 3D Viewer Interface.

Menu and Options

To access the menu to see options, help, and to quit the application, either click on the menu button on the top left corner or press the escape key.

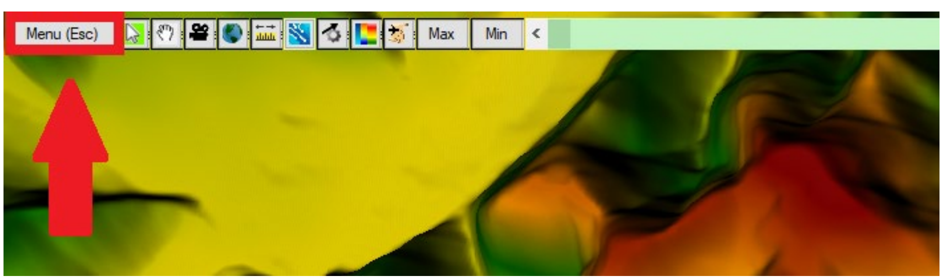
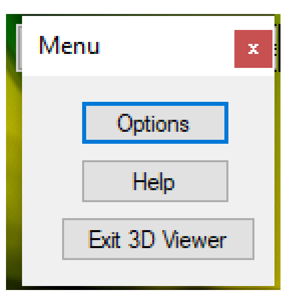


Figure 6-34. 3D Viewer Menu and Options Buttons.



When the Options button is selected a window will appear with four Tabs (General; Graphics; Controls; and Particle Tracing). The following tables describe each of the options on the four Tabs.

General Options

Table 8-3. General Options

Option	**Description**
Water During Animation	When this option is turned on, the water resolution will be the same resolution as the terrain, based on Level of Detail settings. Animation may play slower than usual with this option turned on
Z Scale	How much the terrain elevation is scaled by. Higher value exaggerates the elevations of the terrain, making it easier to see subtle changes in terrain. **(Not Implemented Yet)**

Controls

There are three ways to control the 3D Viewer, mouse and keyboard, just mouse, and a game controller. These controls options controls certain aspects of using the mouse, keyboard, and game controller.

Table 8-4. Controls for mouse and controller sensitivity.

Option	**Description**
Mouse Sensitivity	Controls how much the view changes with mouse movement. Higher sensitivity means more view change with mouse movement. Default is 0.1
Controller X Sensitivity	Control how much the view changes with the right stick of the game controller, horizontal axis only. Default is 0.3
Controller Y Sensitivity	Control how much the view changes with the right stick of the game controller, vertical axis only. Default is 0.15
Invert Y Axis	When this is turned on, moving the vertical axis on either the mouse or game controller will change the view in the opposite direction. Default is off

Pressing the **Change Key/Controller bindings** button will bring up a different window where you can change the various bindings for all the controls of the 3D Viewer.

Table 8-5. Controls for Moving around within the 3D Viewer.

Action	Default Key Binding	Default Controller Binding	Description
Move Forward	W	Left Stick up	Moves the viewer forward in space
Move Backward	S	Left Stick down	Moves the viewer backward in space
Strafe Left	A	Left Stick Left	Moves the viewer in a left side-step fashion in space
Strafe Right	D	Left Stick Right	Moves the viewer in a right side-step fashion in space
Increase Elevation	Space	Right Shoulder Button	Moves the viewer up in space
Decrease Elevation	Left Control	Right Trigger Button	Moves the viewer down in space
Change Results Map	M		Changes the results map between 4 different maps, a realistic map, depth map, velocity map, and water surface elevation map
Toggle Particles	P		Turns on or off the particle tracing effect
Flight Path Play/Pause	Return (Enter)		While a flight path is active, will either play the path or pause it.
Go Forward On Flight Path	Up Arrow	Up Directional Arrow	While a flight path is active, will make the viewer go forward on the flight path
Go Backward On Flight Path	Down Arrow	Down Directional Arrow	While a flight path is active, will make the viewer go backward on the flight path
Increase Viewer Speed	Right Arrow	Left Directional Arrow	Makes the viewer travel faster. The viewer can only go so fast however.
Decrease Viewer Speed	Left Arrow	Right Directional Arrow	Makes the viewer travel faster. The viewer can only go so slow however.
Turn Left	Unbound	Right Stick Left	Rotates the view to the left
Turn Right	Unbound	Right Stick Right	Rotates the view to the right
Change View Up	Unbound	Right Stick Up	Rotates the view up **(No changeable binding yet)**

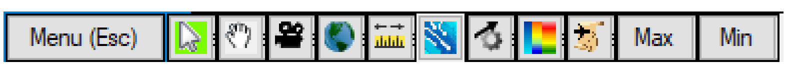
Change View Down	Unbound	Right Stick Down	Rotates the view down **(No changeable binding yet)**
Toggle Mouse Pointer	Tab	A Button	Will either show or hide the mouse pointer **(No changeable binding yet)**

Particle Tracing

Table 8-6. Controls for Controlling Particle Tracing.

Option	Description
Speed	Refers to the animation speed of the particle trace. Default value is 1.
Density	Refers to concentration of tracers in an area. Default value is 1.
Width	Refers to the width of the particle. Default value is 5.
Lifetime	Refers to how long the particle exists on screen before it disappears and a new particle spawns in its place. Default value is 300.
# Particles	Refers to how many particles are shown at any one time. Default value is 10,000.
RGB	Changes the color of the tracers. Each field accepts an integer between 0 and 255. R corresponds to Red, G corresponds to Green and B corresponds to Blue

Toolbar



The Toolbar is located at the top left of the 3D Viewer window. The following Table describes each of the tools.

Table 8-7. Description of each of the 3D Viewer Tools located on the Toolbar.

Tool		Description
Select		Wherever the select pointer is at, it will show the value of either the terrain elevation or water surface value, dependent on the map type chosen. While using the select pointer it's possible to navigate through the terrain through middle-clicking and dragging on the terrain.
Pan		Left click with the pan pointer to navigate through the terrain by clicking and dragging the terrain.

Change Camera Modes		Allows you to change how the 3D Viewer is controlled. When in helicopter mode , the viewer will move forward, backward, left and right on a plane. Elevation is controlled by the elevation up and down keys When in airplane mode , the viewer will move forward in space in relation to where it is currently looking. For example, this means that looking straight up and going forward will cause the viewer to go straight up. **(Not Implemented Yet)**
Zoom to Entire Extent		Zooms to the maximum viewable extent of the terrain, and forces the viewer to look straight down.
Measure Tool		Measure the distance in map units. **(Not Implemented Yet)**
Toggle Particle Tracing		Toggles whether particles show on the water surface.
Particle Tracing Options		A shortcut to get to Particle Tracing Options
Change Results Map		Changes the results map between 4 different maps, a realistic map, depth map, velocity map, and water surface elevation map
Select a Flight Plan		Opens the flight plan window to choose a flight plan. See Flight Plans/Paths section for more information.
Set to Simulation Maximum	Max	Sets the water surface to simulation maximum. **(Not Implemented Yet)**
Set to Simulation Minimum	Min	Sets the water surface to simulation minimum. **(Not Implemented Yet)**
Animation Bar		Change the animation bar position to change the time of the simulation. When a portion of the animation bar is grey, it means that the simulation has not loaded at that time yet.
Play/Pause	/	Plays or pauses the animation

Change Animation Speed		Changes the delay before changing time step in the animation. Note that there is an inherent delay that is unavoidable for each time step. That delay depends on whether you pre-processed the dataset, and whether you have high resolution water turned on during animation. **(Not Implemented Yet)**

Min map

The mini map is shown to assist with acquiring bearings when using the 3D Viewer. Shown in the Figure below is an example of the mini map, which is displayed in the upper right hand corner of the 3D Viewer.

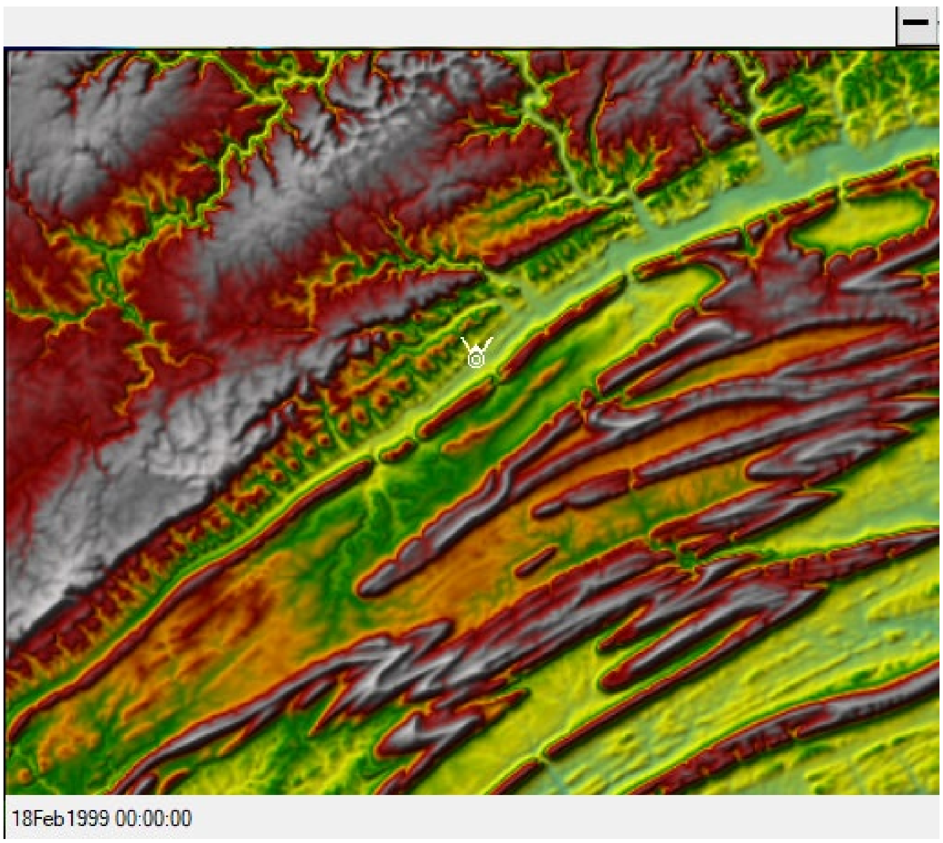


Figure 6-35. Example of the 3D Viewer Mini Map.

There are three components to the mini map, which are explained in the Table below.

Table 8-8. Components of the 3D Viewer Mini Map.

Component	Description
Hide/Show Button	On the top right of the mini-map is a button that will hide or show the mini-map.
Mini-map	A character on the mini-map  shows where the viewer is in relation to the map, and also shows what direction the viewer is currently facing. Here the character is facing north The mini-map also grants the ability for the viewer to be moved anywhere on the map. With either the Select  or Pan  tool, left click anywhere to move the viewer to that location. The left mouse button can also be held down to move the viewer with the mouse drag.
Simulation Date/Time	Shows the current Date/Time for the simulation

Speed/Position Information

On the bottom left corner of the viewer is information about the viewer's speed and its position. A description of these components is in the Table below.

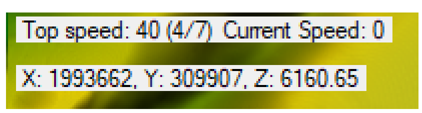


Table 8-9. Speed and Location Components.

Component	Description
Top Speed	The maximum speed that the viewer can currently travel. Can be increased/decreased with the Increase/Decrease Viewer Speed key bindings. The fraction shows what top speed is currently selected out of the possible top speeds. The top speeds are generated based on terrain size.
Current Speed	While the viewer is moving, the current speed will update to show the viewer's current speed.

X,Y	The position of the viewer on the XY plane. These coordinates are the same ones used in RAS Mapper
Z	The elevation of the viewer

How To Use the 3D Viewer

Movement

How movement works on the keyboard/controller depends on what camera mode the 3D Viewer is currently set. By default, the 3D Viewer is set to Helicopter mode ( ), which means the movement keys will move the viewer as if it's on a geometric plane. For example, pressing the forward key will move the viewer forward.

Movement for the mouse works the same for either camera mode. If the cursor is currently the Select cursor ( ), middle click and drag on the terrain will move the viewer proportional to how much the mouse is dragged. If the cursor is the Pan cursor ( ), then a left click and drag is all that is needed.

Changing View/Rotation

Changing the view of the viewer is accomplished through the mouse or the right stick on the controller. It can also be done through the keyboard but the keys are unbound by default to discourage using the keyboard.

To change the view with the mouse right click and drag on the screen. This only applies when the mouse cursor is visible. If the view changes too little with the mouse drag then you can change the mouse sensitivity in the options. When the mouse cursor is not visible there is no need to right click, the view will change with mouse movement. See Mouse Lock/Unlock for more information.

To change the view with a game controller, use the right stick. This only applies when the mouse cursor is not visible. If the view changes too little with the right stick then you can change the controller sensitivity in the options. When the cursor is visible the left stick controls the mouse. See Mouse Lock/Unlock for more information.

Changing Elevation

To change the elevation with the mouse, scroll up to decrease in elevation, scroll down to increase in elevation

Changing elevation with the keyboard or game controller depends on what camera mode the 3D Viewer is currently set. By default, the 3D Viewer is set to Helicopter mode ( ), in Helicopter mode elevation is changed by pressing the Elevation Up or Elevation Down keys. In Airplane mode ( ), these keys are disabled, and to change elevation in this mode look in the direction you wish to ascend/descend to

Mouse Lock/Unlock

By default, the mouse is shown and it considered to be "unlocked". The purpose of an unlocked mouse is to allow easy access to the buttons on the user interface.

To make changing the view with the mouse less tedious, the mouse can be "locked" to the screen by pressing the Tab key. This will hide the mouse cursor but will no longer require the mouse to right click to change view.

Flight Plans/Paths

The 3D Viewer allows the user to specify a flight path polyline in RAS Mapper for the viewer to follow like a train on tracks

Laying Out the Flight Path in RAS Mapper

To lay out a flight path, first open RAS Mapper. To open RAS Mapper, on the main RAS window select the **GIS Tools | RAS Mapper** menu item or by pressing the  **RAS Mapper** button, shown below.

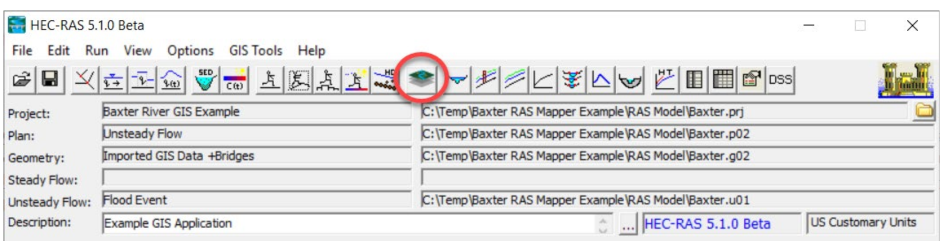


Figure 6-36. HEC-RAS Main Window with HEC-RAS Mapper Button Highlighted.

A flight path layer will be needed to lay out flight paths. To do that right click on **Features** Group in the tree view, and then select **Create New Layer > Flightpath Layer**

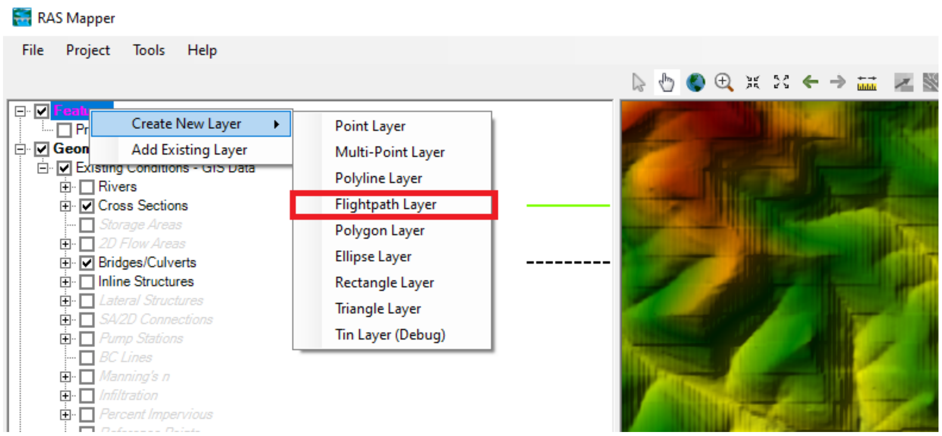


Figure 6-37. HEC-RAS Mapper Menu with Flightpath Layer Selection shown.

Then lay out the polyline you want to use for the flight plan. When you have finished right click the layer and select Stop Editing.

A shape file will be generated under the Flight Paths folder in the root project directory.

Go back to the 3D Viewer and press the Select a Flight Plan button in the toolbar ()

An example 3D Perspective plot is shown in Figure 6-38. The older X-Y-Z Perspective Plot of 1D cross sections is still available from the View menu.

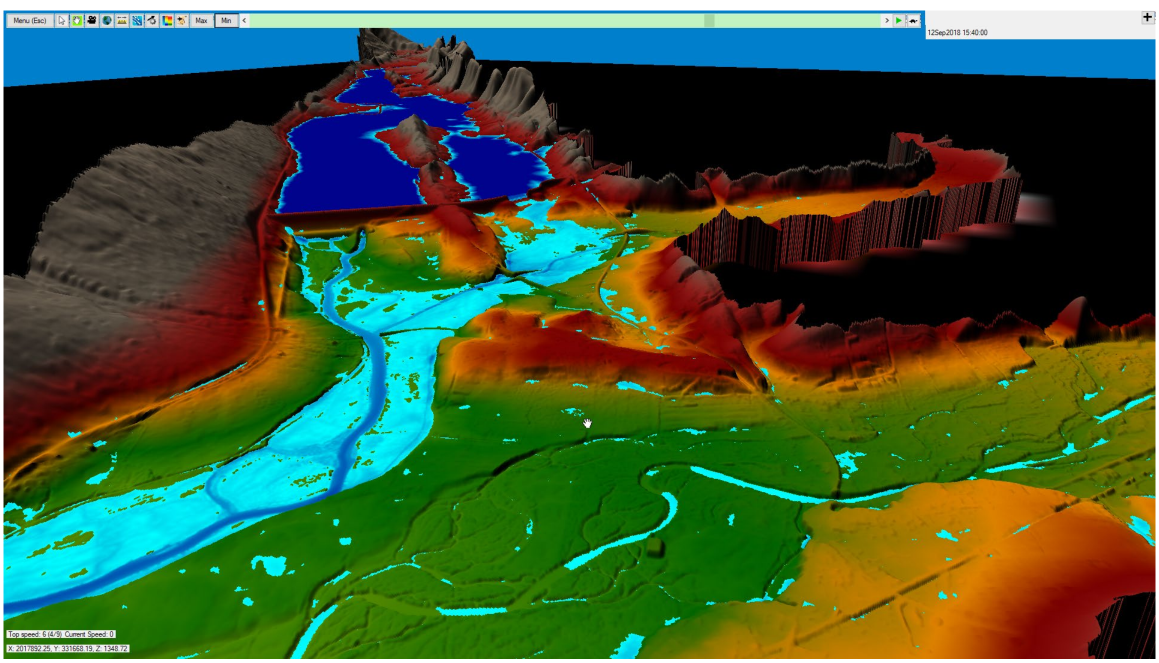


Figure 6-38. Example 3D Perspective Plot.

CHAPTER 7

Steady vs. Unsteady Flow and 1D vs. 2D Modeling

For a detailed discussion on this topic, go to the HEC-RAS Documentation Webpage and download the document called "Modeler Application Guidance for Steady vs. Unsteady, and 1D vs 2D vs 3D Hydraulic Modeling". Here is a link to that document:

https://www.hec.usace.army.mil/publications/TrainingDocuments/TD-41.pdf

The following is a short summary of what is in that document. First the user should think about "When do I need unsteady flow modeling over steady flow modeling? Where should I use 2D Flow Areas over 1D river reaches and/or storage areas ?"

To answer the questions of Steady vs Unsteady Flow, and 1D vs 2D modeling approaches, requires additional information regarding the purpose of the model and the data available. Each river system will have site specific information that must be considered in order to answer the questions of Steady versus Unsteady flow and 1D versus 2D modeling. The following is a partial list of some of the things that the modeler should typically consider when trying to make a modeling approach decision:

1. Physical description of the river channels, floodplain areas, bridges/culverts, other hydraulic structures, levees, roads, etc. that the model will be applied to.
2. What is the typical size, length, and complexity of the systems that these models will be applied to? Is it a 1 mile, 10, 50, 100, 500, or 1000 mile river system
3. What is the source and level of accuracy of the terrain data, cross section data, and hydraulic structure data?
4. What is the general level of accuracy of the hydrology used to drive the models?
5. Will this model be used for Planning type studies, or will it be used for real time modeling and mapping?
6. What type of events (hydrology and boundary conditions) will the models be used to analyze (Dambreak, flash floods, normal rainfall runoff events, steady flows)?
7. What is the typical duration of a flood event on this river system? (1-3 days, 1- week, 1-month, 6-months, years)
8. Is the flow path of the water generally known for the full range of events?
9. Are there unique aspects of the system that will significantly affect the computed results? Such as: is the river tidally influenced; do wind speed and directions affect the water surface elevations; is the river affected by floating ice or ice jams; does there tend to be

debris issues during flood, and does the debris tend to pile up at hydraulic structures; will levee overtopping, breaching, and interior flow routing need to be addressed, Are there any significant bridges and culverts that will cause water to backup behind them during significant flood events, etc…?

10. What are the required outputs from the model (water surface elevations, water depths, arrival times, average velocities, detailed velocities in two dimensions at specific point locations, etc…)?
11. What is the model purpose and expected level of accuracy required?
12. How much time and money do you have to get it done?
13. What experience does the modeler have with 1D and/or 2D modeling?

With that said, I will try to offer both a theoretical opinion and a practical application opinion to the question posed above.

Note: a much more detailed and extensive discussion on this subject is contained in **HEC Training Document 41 (TD-41) "Modeler Application Guidance for Steady vs Unsteady, and 1D vs 2D vs 3D Hydraulic Modeling".** This document can be obtained from the Publications area of HEC's website.

Steady vs. Unsteady Flow Modeling

Steady flow models, or even running an unsteady flow model (1D or 2D) in a steady flow mode (constant flow) should generally not be used when the following situations exist in the river system being analyzed (this is not an exhaustive list):

- The river is tidally influenced, and the tide has a significant effect on the water surface elevations for the area of interest.
- The events being modeled are very dynamic with respect to time (i.e., dam break flood waves; flash floods; river systems in which the peak flow comes up very quickly, stays high for a very short time, and then recedes quickly).
- Complex flow networks and/or flow reversals occur during the event.
- Dynamic events such as levee overtopping and breaching occur during the event.
- Extremely flat river systems, where gravity, hydrostatic pressure, and friction are not necessarily the only significant force acting on the flow (i.e. local and convective acceleration forces).
- Systems with Pump stations that move a significant amount of water.

- Systems with structures that have complex gate operations based on stages and flows in the system

In addition to the specific items listed above, a successful application of any steady-flow model requires that flow rates have already been accurately computed by a hydrologic model or measured by an accurate and complete set of stream gages. If a hydrologic model is being used to not only compute the rainfall-runoff over the watershed, but perform all of the routing within the system, then the flow rates used in the steady-flow model are only as accurate as the hydrologic model could compute them. So, the use of a steady-flow hydraulic model, is predicated on the fact that a hydrologic model was considered to be appropriate for not only developing the flow rate from rainfall runoff computations, but also routing all of the flows through the system during the event. Therefore, a large part of the decision of steady-flow versus unsteady flow hydraulic modeling comes down to the question: is hydrologic stream flow routing accurate enough to produce flow rates that can be used in the corresponding steady-flow hydraulics models.

Even considering all of what is stated above. There are still many areas in which a good hydrologic model (one that is representative of the watershed and has been well calibrated) can be used in conjunction with a steady flow hydraulics model to perform watershed studies requiring hydraulic model results. However, it is up to the modeler to decide when using a steady flow modeling approach is not appropriate.

1D vs. 2D Hydraulic Modeling

The question of 1D versus 2D hydraulic modeling is a much tougher question than steady versus unsteady flow. There are definitely some areas where 2D modeling can produce better results than 1D modeling, and there are also situations in which 1D modeling can produce just as good of results or better than 2D models… with less effort and computational requirements. Unfortunately, there is a very large range of situations that fall into a gray area, and one could list the positive and negative aspects of both methodologies for specific applications.

Here are some areas where I think 2D modeling can give better results than 1D modeling:

- When modeling an area behind a leveed system, and the levee will be overtopped and/or breached, the water can go in many directions. If that interior area has a slope to it, water will travel overland in potentially many directions before it finds its way to the lowest point of the protected area, and then it will begin to pond and potentially overtop and/or breach the levee on the lower end of the system. However, if a protected area is small, and ultimately the whole area will fill to a level pool, then 1D model is fine for predicting the final water surface and extent of the inundation.

- Bays and estuaries in which the flow will continuously go in multiple directions due to tidal fluctuations and river flows coming into the bay/estuary at multiple locations and times.
- Areas and/or events in which the flow path of the water is not completely known.
- Highly braided streams
- Alluvial fans – however, this is very debatable that any numerical model can capture a flood event accurately on an alluvial fan, due to the episodic nature of flow evolutions that can change the whole direction of the channels during the event.
- Flow around abrupt bends in which a significant amount of super elevation will occur during the event.
- Very wide and flat flood plains, such that when the flows goes out into the overbank area, the water will take multiple flow paths and have varying water surface elevations and velocities in multiple directions.
- Applications where it is very important to obtain detailed velocities for the hydraulics of flow around an object, such as a bridge abutment or bridge piers, etc…

The following are areas in which 1D modeling can potentially produce results as good as 2D modeling (from the perspective of computed water surface elevations, and flow/stage hydrographs), with less effort (from a model development, calibration, and application viewpoint, as well as a computational time viewpoint):

- Rivers and floodplains in which the dominant flow directions and forces follow the general river flow path. This covers a lot of river systems in my opinion, but it is obviously debatable as to the significance that lateral and vertical velocities and forces impact the computed water surface elevations and the resulting flood inundation boundary.
- Steep streams that are highly gravity driven and have small overbank areas.
- River systems that contain a lot of bridges/culvert crossings, weirs, dams and other gated structures, levees, pump stations, etc…. and these structures impact the computed stages and flows within the river system. I have not seen any 2D model yet that has a comprehensive set of hydraulic structure modules/capabilities that can handle the full range of hydraulic flow situations that can come up on many of our river systems. This is an area that the current state of the art in 1D models is far ahead of the 2D models. This statement does not mean that these capabilities cannot be incorporated into a 2D model, It just means that I have not seen a widely used 2D model that has such a comprehensive set of capabilities.
- Medium to large river systems, where we are modeling a large portion of the system (100 or more miles), and it is necessary to run longer time period forecasts (i.e. 2 week to 6 month forecasts). Even with the tremendous advancements in multi-processor

computing, and GPU (Graphics Processor Units) computing, there are still significant spatial and simulation time limitations on what we can effectively use 2D models for in the real time forecasting domain. This will obviously be changing over time.

- Areas in which the basic data does not support the potential gain of using a 2D model. If you do not have detailed overbank and channel bathymetry, or you only have detailed cross sections at representative distances apart, many of the benefits of the 2D model will not be realized due to the poor accuracy of the terrain data.

With all of that said, there are many areas in which it will be highly debatable as to the relevant accuracy of using a 1D or a 2D modeling approach for a specific application. There are many aspects to consider, other than purely “am I solving the full Saint Venant equations in one dimension or two dimensions”. I believe that there are both knowledge gaps in understanding when 1D versus 2D should be used, and there are tool gaps. I personally believe that combined 1D/2D models will play an important role in our modeling efforts in the near term. This is an area where the hydraulic modeling tools need to be improved.

I am also of the view point that the majority of uncertainty and ability to accurately forecast stages and flows in river systems is mostly due to poor estimation of rainfall both spatially and temporally, and hydrologic modeling, which often includes large portions of ungaged areas in which little to no calibration could be performed. This can often be a much greater contributor to forecast/modeling error than any differences arising from 1D versus 2D model choices.

CHAPTER 8

Optimize Your Computer for Fast Computations

Now that 2D modeling is becoming widespread in the HEC-RAS community, a lot of modelers are wanting to know what kind of computer to get to maximize computation speed when running those large 2D datasets.

Before moving into suggestions for 2D modeling, first in 1D modeling, multiple processing cores are NOT currently used. If you plan to only do 1D modeling, having extra cores will not help you with speed. In this case, the processor speed is everything. So get the fastest processor you can for 1D model (e.g. 3.4 Ghz or higher)

For the rest of this post, I'll assume you want to optimize your computer for 2D HEC-RAS modeling, since those are the models that typically will take longest to run.

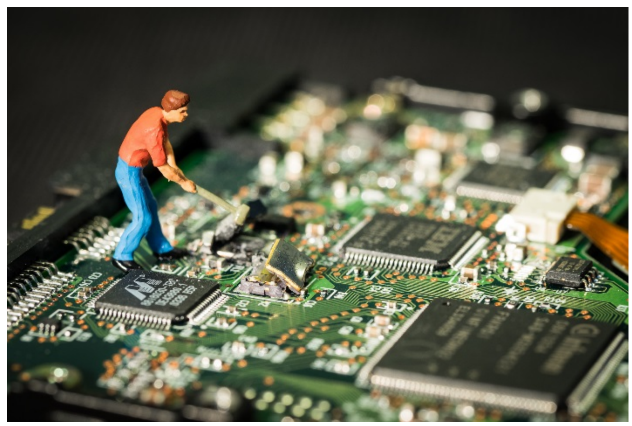
1. <u>More cores is not always better</u>. In fact, it has been found that for smaller 2D areas (e.g. less than 10,000 cells or so), 8 cores may indeed run slower than 4 or 6 cores. The reason behind this is that there is a level of computing overhead used just to transfer data between cores. Fortunately, HEC-RAS has an option to change the number of cores you wish to use in the Computation Options and Tolerances window (from the unsteady flow analysis window…Options…Calculation Options and Tolerances…2D Flow Options tab). For smaller datasets, I suggest experimenting with this to optimize computation speed. "All Available" may not necessarily be the fastest. But for large numbers of cells, you're going to want as many cores as you can get your hands on.

   So for the number of Cores, get as many as you can, but not at the expense of processor speed. Make sure you get at least 3.2 to 3.4 Ghz or higher processors, no matter how many cores you get.

   Hyper-threading is an Intel technology that attempts to keep CPU resources as busy as possible. Each real CPU core has what appears to be two cores. However, there is really only one true core. For example, the typical Intel I7 chip has four real CPU cores, but if you open Task Manager and go to the Performance Tab, you will see four across the top, and what appears to be four more below it. These are virtual cores. Each real CPU core has only one true math processing unit, but with Hyper-threading it has two instruction feeders. Hyper-threading tries to eliminate stalls by always having another thread at the ready in a second virtual core. If one thread stalls (not requiring the math unit) on virtual core A, virtual core B will instantly start picking up the slack, so the execution units keep working at 100%.

The RAS 2D compute engine is extremely math heavy. So for each core it utilizes, it is almost always using 100% of the math unit. So the second virtual core (Hyper-thread) is never used. So back to our Intel I7 chip example. An Intel I7 has 4 real cores, but appears to have 8 cores (4 virtual cores). RAS will only use the four real cores, and it will keep them almost 100% busy. However, Task Manager reports this as only 50% utilization of the CPU. However, this is truly 100% utilization of the four real cores math units.

2. Processor speed is still paramount. Do NOT think you will have fast HEC-RAS model run times just because you have a computer with 16 cores or more. If all of your cores have slow processor speeds, you'll get some benefit out of the number of cores, but you will be disappointed in the overall speed for a wide range of model types (1D/2D) and sizes. So make sure even if you get a large number of cores, you are not doing so at the expense of fast processor clock speeds. As of the date of this article (August 2016) and the current version of HEC-RAS (5.0.1), 3.2 to 3.4 GHz or higher is a good clock speed for fast running models.



3. Your hard drive is important. Especially if you are producing a lot of output. Small detailed output intervals, small mapping output intervals, writing computation level output, etc. All of these settings affect how much and how often output is written to the hard drive during run time. Solid state hard drives (SSD) are typically going to be better than the traditional spinning hard drive (HDD).

4. RAM is important, but not as much as you might think. While RAM is definitely important, it is not as important for 2D modeling as number of cores and processor speed. You do want enough RAM to run your operating system and have your entire HEC-RAS model in memory, without the operating system having to swap things in and out of memory. That being said, if you plan to do multiple HEC-RAS models at the same time, or you have a habit of keeping lots of programs open and running in the background of your computer, you may want to get a computer with a lot of RAM. I would venture to guess that if you are buying a computer with a lot of cores with fast clock speeds, I'm

guessing that computer will have enough RAM. But RAM is cheap, so you might as well load up on it while you're building the HEC-RAS computer of your dreams.

5. Graphics card does not matter. While some of your other programs run best on a super-charged graphics card, HEC-RAS currently does not support GPU computing. For HEC-RAS modeling, don't waste your money on an expensive graphics card. However, you may seem so noticeable improvement in the snappiness of image rendering or particle tracing with a better graphics card. If money is no object, get a top-of-the-line graphics card, but this is one area you can sacrifice if you need to save some dough.

To sum up, my recommendation for building a computer to optimize 2D runs in HEC-RAS is as follows:

Get as many processing cores as you can, but do not do so in expense of processor speed. Make sure your computer has processors that are 3.2 to 3.4 Ghz or even higher (the faster the better). Get an SSD hard drive, and max out your RAM. 32 Gb is a good amount and can run large models (millions of cells). If you plan to run really large models (many millions of cells), you may want more RAM. Pretty simple really. And by the way, 24-inch (or larger) dual monitors really helps with viewing all those HEC-RAS windows you have open.

APPENDIX A

References

Alonso, Santillana and Dawson. 2008. On the diffusive wave approximation of the shallow water equations. Euro. J. Appl. Math. 2008, Vol. 19.

Balzano. 1998. Evaluation of methods for numerical simulation of wetting and drying in shallow water flow models. Coastal Eng. 1998, 34.

Casulli. 2008. A high-resolution wetting and drying algorithm for free-surface hydrodynamics. Int. J. Numer. Meth. Fluids. 2008.

Casulli and Cattani. 1994. Stability, accuracy and efficiency of a semi-implicit method for three-dimensional shallow water flow. Computers Math. Applic. 1994, Vol. 27.

Casulli and Cheng. 1992. Semi-implicit finite difference methods for three dimensional shallow water flow. Int. J. Numer. Meth. Fluids. 1992, Vol. 15.

Casulli and Cheng. 1990. Stability analysis of Eulerian-Lagrangian methods for the one-dimensional shallow water equations. Appl. Math. Modelling. 1990, Vol. 14.

Casulli and Stelling. 1998. Numerical simulation of 3D quasi-hydrostatic free-surface flows. J. Hydraulic Eng. 1998.

Casulli and Stelling. 2010. Semi-implicit subgrid modelling of three dimensional free-surface flows. Int. J. Numer. Meth. Fluids. 2010.

Casulli and Walters. 2000. An unstructured grid, three-dimensional model based on the shallow water equations. Int. J. Numer. Meth. Fluids. 2000, 32.

Casulli and Zanolli. 2005. High resolution methods for multidimensional advection-diffusion problems in free surface hydrodynamics. Ocean Modelling. 2005, 10.

Casulli and Zanolli. 2002. Semi-implicit numerical modelling of nonhydrostatic free-surface flows for environmental problems. Math. Comp. Modelling. 2002, 36.

Casulli. 1997. Numerical simulation of three-dimensional free surface flow in isopycnal co-ordinates. Int. J. Numer. Meth. Fluids. 1997, Vol. 25.

Casulli. 1990. Semi-implicit finite difference methods for the two-dimensional shallow water equations. J. Comp. Physics. 1990, 86.

Fischer, et al. 1979. Mixing in Inland and Coastal Waters. 1979.

Geospatial Data Abstraction Library (GDAL), 2014. GDAL: http://www.gdal.org/

Green, W.H., and Ampt, G.A. 1911. Studies on soil physics, Journal of Agricultural Science, 4(1), 1–24.

Ham, Pietrzak and Stelling. 2006. A streamline tracking algorithm for semi-Lagrangian advection schemes based on the analytic integration of the velocity field. J. Comp. Appl. Math. 2006, 192.

HDF Group, 2014. Hierarchical Data Format (HDF). The HDF Group: http://www.hdfgroup.org/HDF5/

Hromadka, et al. 2010. Manning's equation and two dimensional flow analogs. J. Hydrology. 2010, 389.

Krol. 2009. Momentum exchange as a common physical background of a transparent and physically coherent description of transport phenomena. Turbulence, Heat and Mass Transfer. 2009, 6.

Lamb, Crossley and Waller. 2008. A fast two-dimensional floodplain inundation model. Water Management. 2008, 162.

Lang, G. 2012. FuE-Vorhaben UnTRIM SubGrid-Topografie Abschlussbericht. Report A39550370150 [In German].

Leer. 1979. Towards the ultimate conservative difference scheme. A second-order sequel to Godunov's method. J. Comp. Physics. 1979, 32.

Meyer, et al. 2002. Generalized barycentric coordinates on irregular polygons. Journal of Graphics Tools. 2002, Vol. 7.

Ogden, F.L., and Saghafian, B. 1997. Green and Ampt infiltration with redistribution. Journal of Irrigation and Drainage Engineering ASCE, 123(5), 386–393.

Pathirana, et al. 2008. A simple 2-D inundation model for incorporating flood damage in urban drainage planning. Hydrol. Earth Syst. Sci. Discuss. 2008, 5.

Russel, Takano and Abramopoulos. 1987. Comparison of horizontal difference schemes for the shallow water equations on a sphere. Short- and Medium-Range Numerical Weather Prediction. 1987.

Santillana and Dawson. 2010. A local discontinuous Galerkin method for a doubly nonlinear diffusion equation arising in shallow water modeling. Comp. Meth. Appl. Mech. Eng. 2010, 199.

Santillana and Dawson. 2009. A numerical approach to study the properties of solutions of the diffusive wave approximation of the shallow water equations. Comput. Geosci. 2009.

Schenk and Gärtner. 2006. On fast factorization pivoting methods for symmetric indefinite systems. Elec. Trans. Numer. Anal. 2006, 23.

Schenk and Gärtner. 2011. Parallel Sparse Direct and Multi-Recursive Iterative Linear Solvers: Pardiso User Guide Version 4.1.2. 2011.

Schenk and Gärtner. 2004. Solving unsymmetric sparse systems of linear equations with PARDISO. J. Future Generation Comp. Systems. 2004, 20(3).

Schenk, Bollhoefer and Roemer. 2008. On large-scale diagonalization techniques for the Anderson model of localization. SIAM Review. 2008, 50.

Schenk, Waechter and Hagermann. 2007. Matching-based preprocessing algorithms to the solution of saddle-point problems in large-scale nonconvex interior-point optimization. J. Comp. Optim. App. 2007, Vol. 36, 2-3.

SCS 1985. National engineering handbook. Section 4-Hydrology. Washington, DC.

Sehili, A., L.G. Gunther, C. Lippert 2014. High-resolution subgrid models: background, grid generation, and implementation. Ocean Dynamics, Vol. 64, 519-535.

Skaggs, R.W. and Khaheel, R. 1982. Chapter 4: Infiltration. Hydrologic Modeling of Small Watersheds. Ed. by C.T. Haan, H. P. Johnson and D.L. Brakensiek, 139-149, St. Joseph, MI, ASCE. 121–168.

Smith, R.E., Corradini. C., and Melone, F. 1993. Modeling infiltration for multistorm run-off events. Water Resources Research 29(1), 133-144.

Tayefi, et al. 2007. A comparison of one- and two-dimensional approaches to modelling flood inundation over complex upland floodplains. Hydrol. Processes. 2007, 21.

Thacker. 1981. Some exact solutions to the non-linear shallow-water wave equations. J. Fluid Mech. 1981, Vol. 107.

Versteeg and Malalasekera. 2007. An Introduction to Computational Fluid Dynamics: The Finite Volume Method. 2007.

Wang, Zhao and Fringer. 2011. Reconstruction of vector fields for semi-Lagrangian advection on unstructured, staggered grids. Ocean Modelling. 2011, 40.

Weil, R.R., and Brady, N.C. 2016. The Nature and Properties of Soils (15th ed.). Columbus, Ohio: Pearson. 221 p.

Yu and Lane. 2006. Urban fluvial flood modelling using a two-dimensional diffusion-wave treatment, part 1: mesh resolution effects. Hydrol. Processes. 2006, 20.

Yu and Lane. 2006. Urban fluvial flood modelling using a two-dimensional diffusion-wave treatment, part 2: development of a sub-grid-scale treatment. Hydrol. Processes. 2006, 20.

APPENDIX B

RAS Mapper Supported File Formats

The following is a list of the file formats that can currently be imported into HEC RAS Mapper:

VRT :   Virtual Raster

GTiff :   GeoTIFF

NITF :   National Imagery Transmission Format

RPFTOC :   Raster Product Format TOC format

ECRGTOC :   ECRG TOC format

HFA :   Erdas Imagine Images (.img)

SAR_CEOS :   CEOS SAR Image

CEOS :   CEOS Image

JAXAPALSAR :   JAXA PALSAR Product Reader (Level 1.1/1.5)

GFF :   Ground-based SAR Applications Testbed File Format (.gff)

ELAS :   ELAS

AIG :   Arc/Info Binary Grid

AAIGrid :   Arc/Info ASCII Grid

GRASSASCIIGrid :   GRASS ASCII Grid

SDTS :   SDTS Raster

DTED :   DTED Elevation Raster

PNG :   Portable Network Graphics

JPEG :   JPEG JFIF

MEM :   In Memory Raster

JDEM :   Japanese DEM (.mem)

GIF :   Graphics Interchange Format (.gif)

BIGGIF :   Graphics Interchange Format (.gif)

ESAT : Envisat Image Format

BSB : Maptech BSB Nautical Charts

XPM : X11 PixMap Format

BMP : MS Windows Device Independent Bitmap

DIMAP : SPOT DIMAP

AirSAR : AirSAR Polarimetric Image

RS2 : RadarSat 2 XML Product

PCIDSK : PCIDSK Database File

PCRaster : PCRaster Raster File

ILWIS : ILWIS Raster Map

SGI : SGI Image File Format 1.0

SRTMHGT : SRTMHGT File Format

Leveller : Leveller heightfield

Terragen : Terragen heightfield

GMT : GMT NetCDF Grid Format

netCDF : Network Common Data Format

ISIS3 : USGS Astrogeology ISIS cube (Version 3)

ISIS2 : USGS Astrogeology ISIS cube (Version 2)

PDS : NASA Planetary Data System

TIL : EarthWatch .TIL

ERS : ERMapper .ers Labelled

JPEG2000 : JPEG-2000 part 1 (ISO/IEC 15444-1)

L1B : NOAA Polar Orbiter Level 1b Data Set

FIT : FIT Image

GRIB : GRIdded Binary (.grb)

MrSID : Multi-resolution Seamless Image Database (MrSID)

JP2MrSID : MrSID JPEG2000

MG4Lidar : MrSID Generation 4 / Lidar (.sid)

RMF : Raster Matrix Format

WCS : OGC Web Coverage Service

WMS : OGC Web Map Service

MSGN : EUMETSAT Archive native (.nat)

RST : Idrisi Raster A.1

INGR : Intergraph Raster

GSAG : Golden Software ASCII Grid (.grd)

GSBG : Golden Software Binary Grid (.grd)

GS7BG : Golden Software 7 Binary Grid (.grd)

COSAR : COSAR Annotated Binary Matrix (TerraSAR-X)

TSX : TerraSAR-X Product

COASP : DRDC COASP SAR Processor Raster

R : R Object Data Store

MAP : OziExplorer .MAP

PNM : Portable Pixmap Format (netpbm)

DOQ1 : USGS DOQ (Old Style)

DOQ2 : USGS DOQ (New Style)

ENVI : ENVI .hdr Labelled

EHdr : ESRI .hdr Labelled

GenBin : Generic Binary (.hdr Labelled)

PAux : PCI .aux Labelled

MFF : Vexcel MFF Raster

MFF2 : Vexcel MFF2 (HKV) Raster

FujiBAS : Fuji BAS Scanner Image

GSC : GSC Geogrid

FAST : EOSAT FAST Format

BT : VTP .bt (Binary Terrain) 1.3 Format

LAN : Erdas .LAN/.GIS

CPG :   Convair PolGASP

IDA :   Image Data and Analysis

NDF :   NLAPS Data Format

EIR :   Erdas Imagine Raw

DIPEx :   DIPEx

LCP :   FARSITE v.4 Landscape File (.lcp)

GTX :   NOAA Vertical Datum .GTX

LOSLAS :   NADCON .los/.las Datum Grid Shift

NTv2 :   NTv2 Datum Grid Shift

CTable2 :   CTable2 Datum Grid Shift

ACE2 :   ACE2

SNODAS :   Snow Data Assimilation System

ARG :   Azavea Raster Grid format

RIK :   Swedish Grid RIK (.rik)

USGSDEM :   USGS Optional ASCII DEM (and CDED)

GXF :   GeoSoft Grid Exchange Format

HTTP :   HTTP Fetching Wrapper

NWT_GRD :   Northwood Numeric Grid Format .grd/.tab

NWT_GRC :   Northwood Classified Grid Format .grc/.tab

ADRG :   ARC Digitized Raster Graphics

SRP :   Standard Raster Product (ASRP/USRP)

BLX :   Magellan topo (.blx)

GeoRaster :   Oracle Spatial GeoRaster

Rasterlite :   Rasterlite

SAGA :   SAGA GIS Binary Grid (.sdat)

KMLSUPEROVERLAY :   Kml Super Overlay

XYZ :   ASCII Gridded XYZ

HF2 :   HF2/HFZ heightfield raster

PDF : Geospatial PDF

OZI : OziExplorer Image File

CTG : USGS LULC Composite Theme Grid

E00GRID : Arc/Info Export E00 GRID

ZMap : ZMap Plus Grid

NGSGEOID : NOAA NGS Geoid Height Grids

MBTiles : MBTiles

IRIS : IRIS data (.PPI, .CAPPi etc)