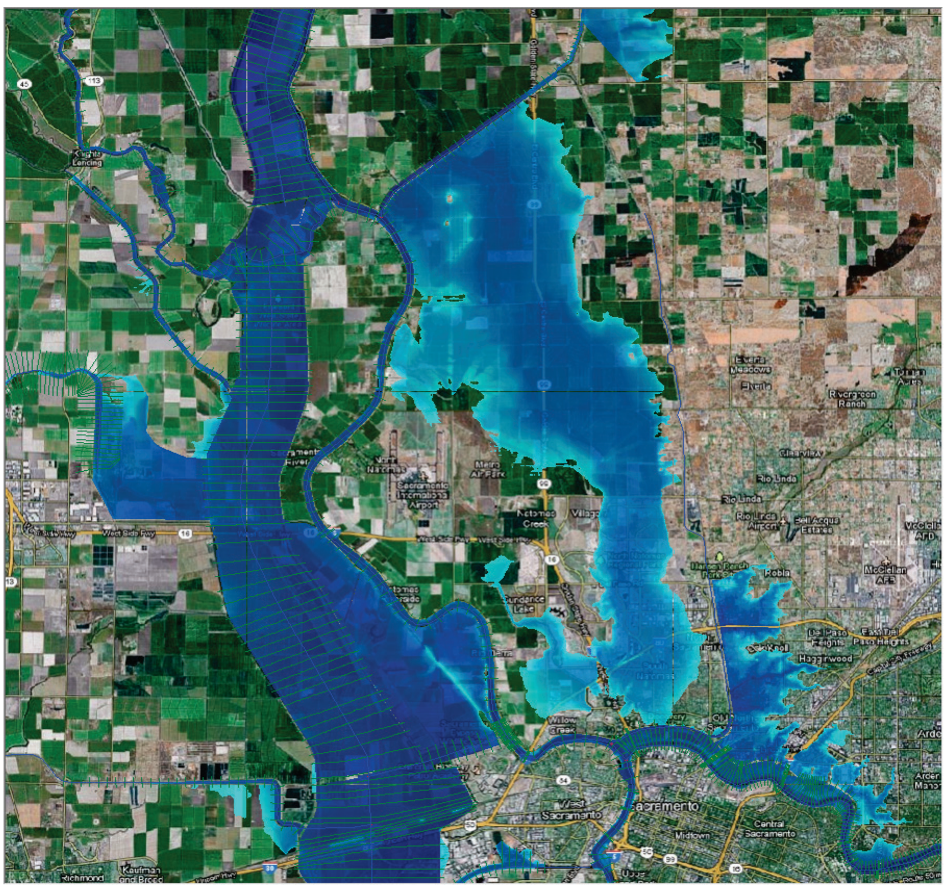
US Army Corps
of Engineers
Hydrologic Engineering Center

HEC-RAS
River Analysis System



2D Modeling User's Manual

Version 5.0
February 2016

Approved for Public Release. Distribution Unlimited.

CPD-68A

REPORT DOCUMENTATION PAGE

Form Approved OMB No. 0704-0188

1. REPORT DATE *(DD-MM-YYYY)* February 2016	**2. REPORT TYPE** Computer Program Documentation	**3. DATES COVERED** *(From - To)*

4. TITLE AND SUBTITLE HEC-RAS River Analysis System, 2D Modeling User's Manual Version 5.0	**5a. CONTRACT NUMBER** **5b. GRANT NUMBER** **5c. PROGRAM ELEMENT NUMBER**
6. AUTHOR(S) Gary W. Brunner, CEIWR-HEC	**5d. PROJECT NUMBER** **5e. TASK NUMBER** **5F. WORK UNIT NUMBER**
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) US Army Corps of Engineers Institute for Water Resources Hydrologic Engineering Center (HEC) 609 Second Street Davis, CA 95616-4687	**8. PERFORMING ORGANIZATION REPORT NUMBER** CPD-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** U	**c. THIS PAGE** U	UU	171	**19b. TELEPHONE NUMBER**

HEC-RAS
River Analysis System

2D Modeling

User's Manual

February 2016

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:

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

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

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

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

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

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

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

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

Waiver of Warranty:

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

Limitation of Liability:

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

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

Indemnity:

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

Assent:

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

Table of Contents

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 the user to perform one-dimensional steady flow hydraulics; one and two-dimensional unsteady flow river hydraulics; 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 and now Version 5.0 in 2015.

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. The steady flow water surface profiles computation module and the majority of the one-dimensional unsteady flow computational module were programmed by Mr. Steven S. Piper. The Skyline one-dimensional unsteady flow matrix solution algorithm was developed by Dr. Robert L. Barkau (author of UNET).

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

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 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, and Alex J. Kennedy.

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: Mr. Vern R. Bonner, Mr. Richard Hayes, Mr. John Peters, Mr. Al Montalvo, and Dr. 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. SQLite – Reading and writing database files. SQLite: https://www.sqlite.org/

6. cURL - HTTP support for GDAL http://curl.haxx.se/

7. Clipper – an open source freeware library for clipping and offsetting lines and polygons. http://www.angusj.com/delphi/clipper.php

CHAPTER 1

Introduction

HEC has added the ability to perform two-dimensional (2D) hydrodynamic routing within the unsteady flow analysis portion of HEC-RAS. Users can now perform one-dimensional (1D) unsteady-flow modeling, two-dimensional (2D) unsteady-flow modeling (Saint Venant equations or Diffusion Wave equations), 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 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.

HEC-RAS Two-Dimensional Flow 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. **Saint-Venant or Diffusion Wave Equations in 2D.** The program solves either the 2D Saint Venant equations (with optional momentum additions for turbulence 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 Saint-Venant 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 Saint-Venant 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).

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 treaded the same as an unstructured mesh, except the software takes advantage of cells that are orthogonal to each other (i.e. this simplifies some of the computations required). 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. The computational mesh does not need to be orthogonal but if the mesh is orthogonal the numerical discretization is simplified and more efficient.

6. **Detailed Hydraulic Table Properties 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.

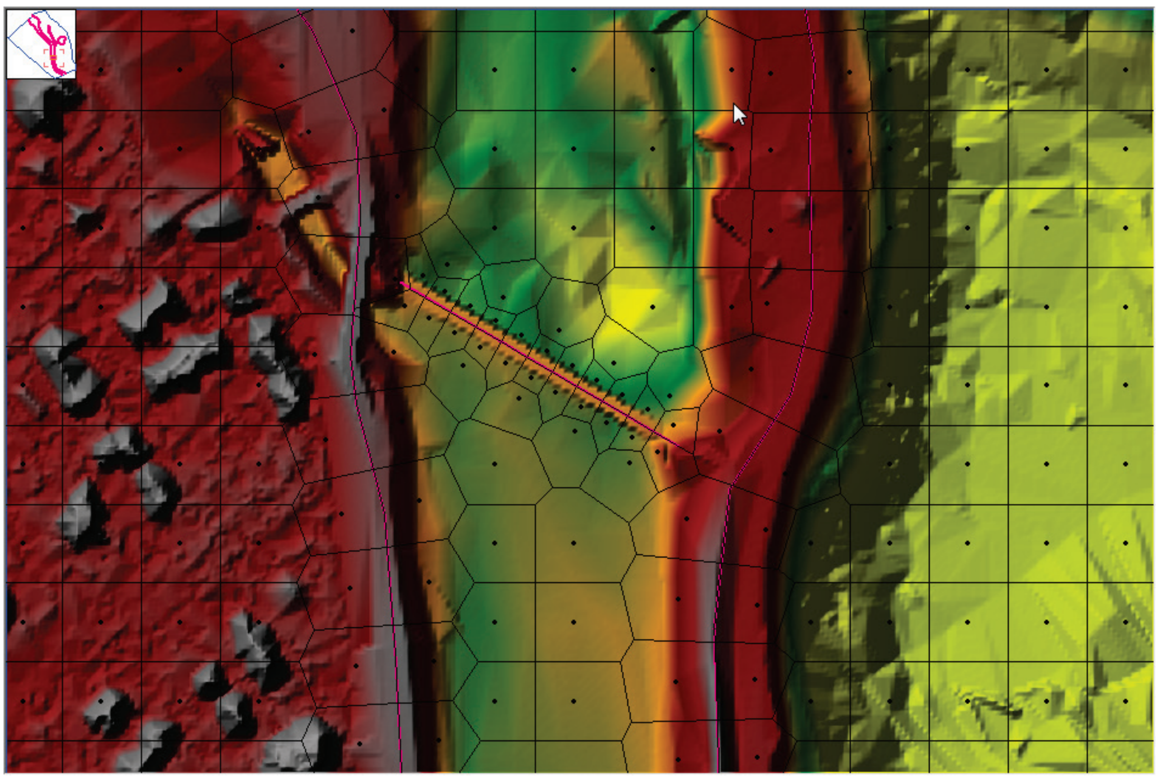


Figure 1-1. Unstructured computational mesh with detailed sub-grid terrain data.

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.

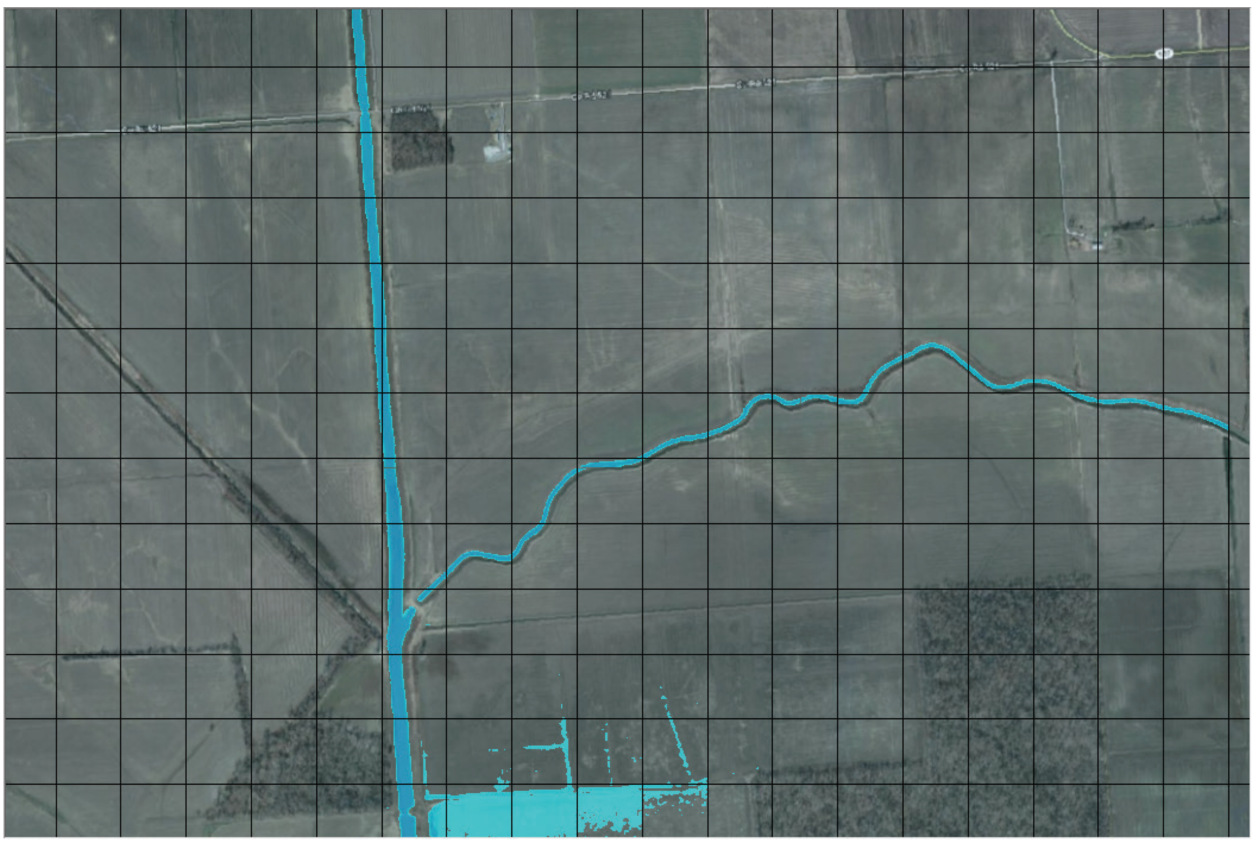


Figure 1-2. Example showing the benefits of using the detailed subgrid terrain for the cell and face hydraulic properties.

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 and 32-Bit Computational Engines.** HEC-RAS now comes with both 64 bit and 32-bit computational engines. The software will use the 64-bit computational engines automatically if installed on a 64-bit operating system. The 64-bit computational engines run faster than the 32-bit and can handle much larger data sets.

Overview of how to Develop a Combined 1D/2D Unsteady Flow Model with HEC-RAS

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 other 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 Land classification data set within HEC-RAS Mapper in order to establish Manning’s n values within the 2D Flow Areas. Additionally HEC-RAS has option for user defined polygons that can be used to override the Land Classification data or 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 Geometry editor, 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; increase or decrease cell density as needed; Add, Move, or Delete cell centers where needed.
9. Run the 2D geometric pre-processor from RAS Mapper in order to create the cell and face hydraulic property tables.
10. Connect the 2D Flow Areas to 1D Hydraulic elements (river reaches, Lateral structures, storage area/2D flow area hydraulic connections) as needed.
11. Add any necessary hydraulic structures inside of a 2D Flow Area.
12. From 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 in HEC-RAS

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. More flexibility for adding internal hydraulic structures inside of a 2D flow area.
2. Cannot currently perform sediment transport erosion/deposition in 2D flow areas.
3. Cannot current perform water quality modeling in 2D flow areas.
4. Cannot connect Pump stations to 2D flow area cells.

5. Cannot use the HEC-RAS bridge modeling capabilities inside of a 2D flow area. You can do culverts, weirs, and breaching by using the **SA/2D Area Conn** tool.

CHAPTER 2

Developing a Terrain Model for use in 2D Modeling and Results Mapping

It is absolutely 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, create a good terrain model, 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. 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. For more details on creating terrain models with HEC-RAS Mapper, please review the chapter on HEC-RAS Mapper in the HEC-RAS 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.

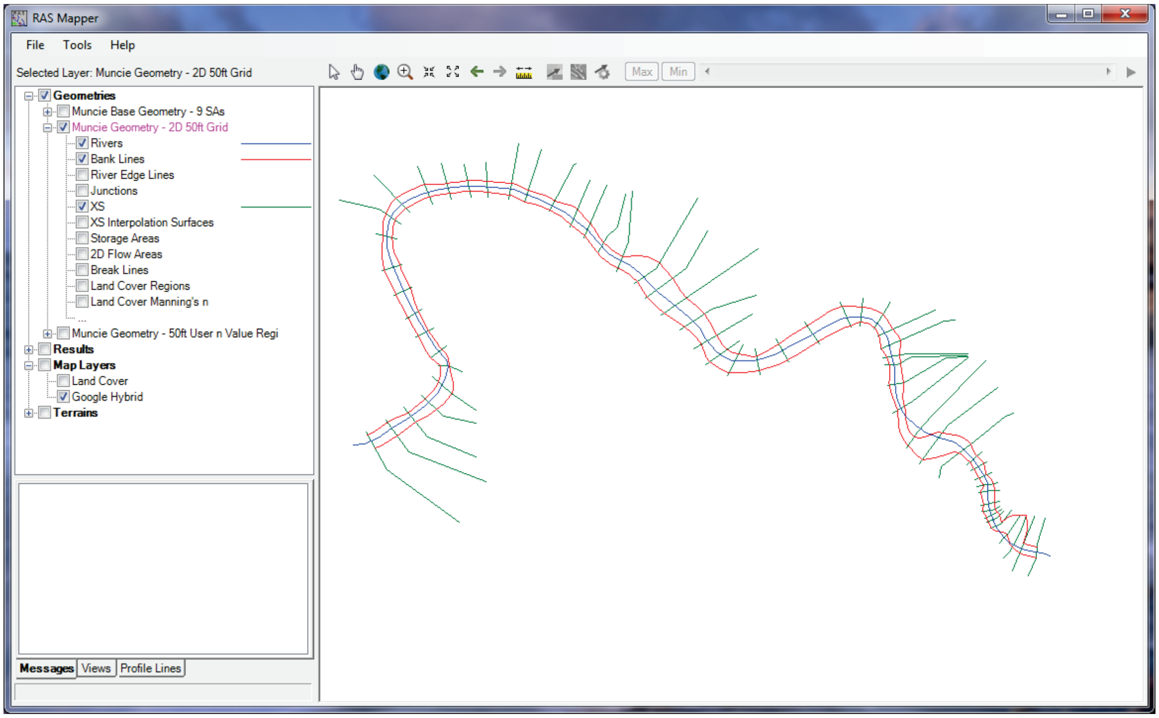


Figure 2-1. RAS Mapper with no terrain or other map layers loaded.

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 **Tools | Set Projection for Project** menu item from the RAS Mapper menu bar. When the **Set Projection** option is selected the window shown below will appear (Figure 2-2).

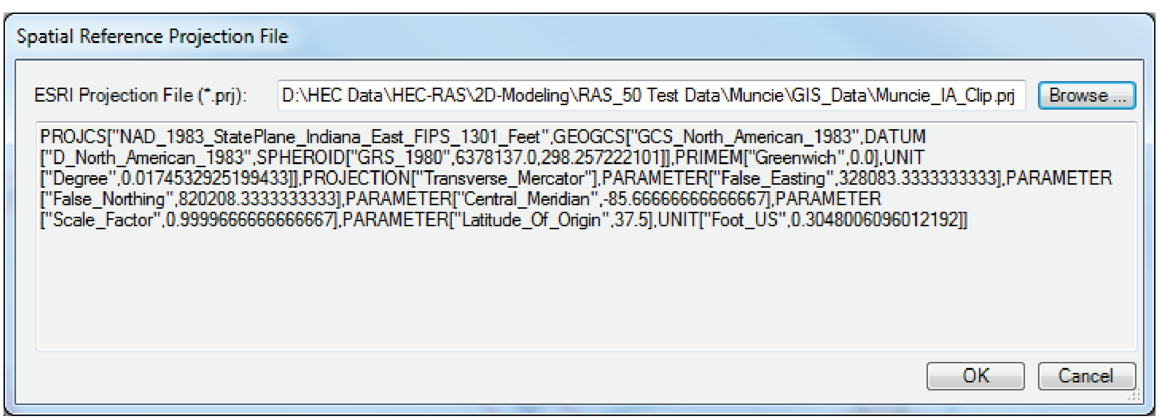


Figure 2-2. Editor to set the RAS project's spatial reference system.

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 State Plane 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. To develop a new terrain data set (terrain model), select the **Tools** | **New Terrain** menu item from the **RAS Mapper** main menu bar. 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 processes 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.

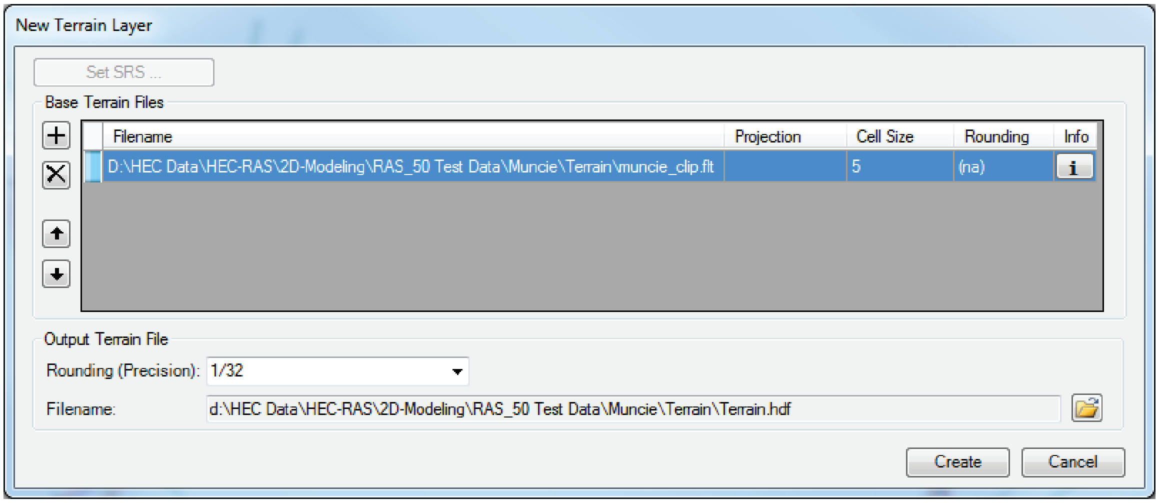


Figure 2-3. Example New Terrain Layer dialog.

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.

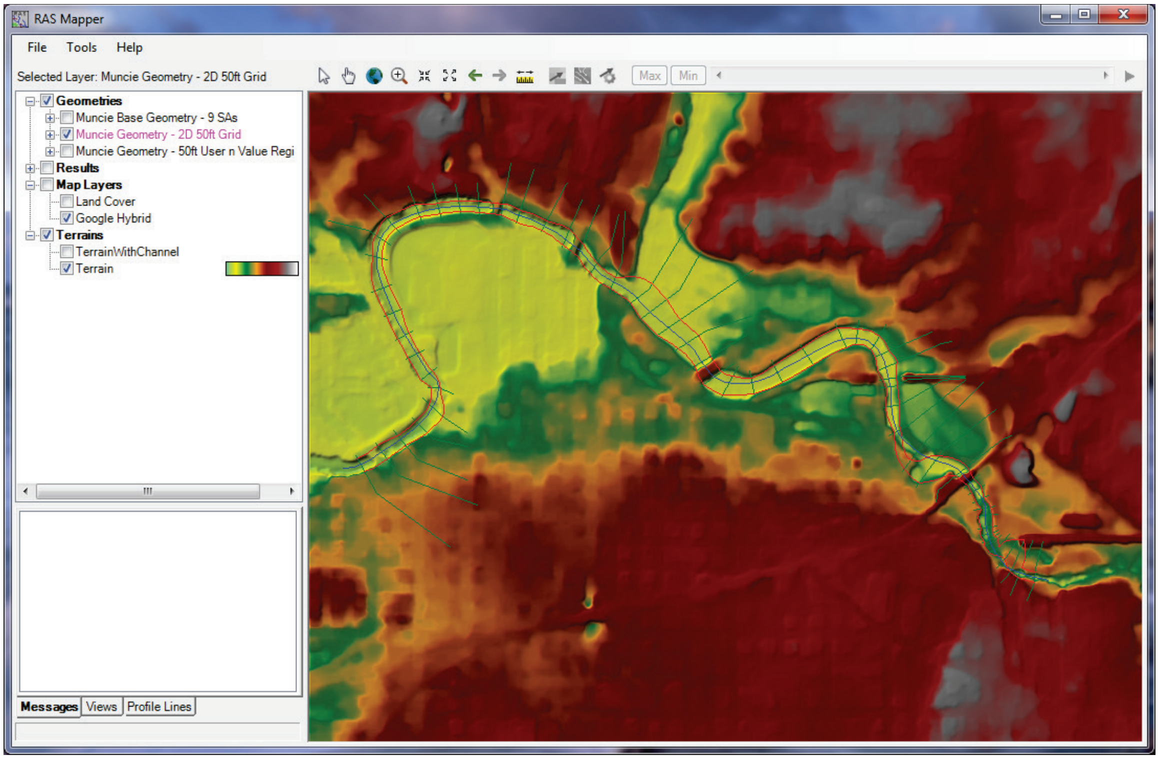


Figure 2-4. RAS Mapper with a Terrain Data Layer added.

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 **Layer Properties**. The **Layer 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).

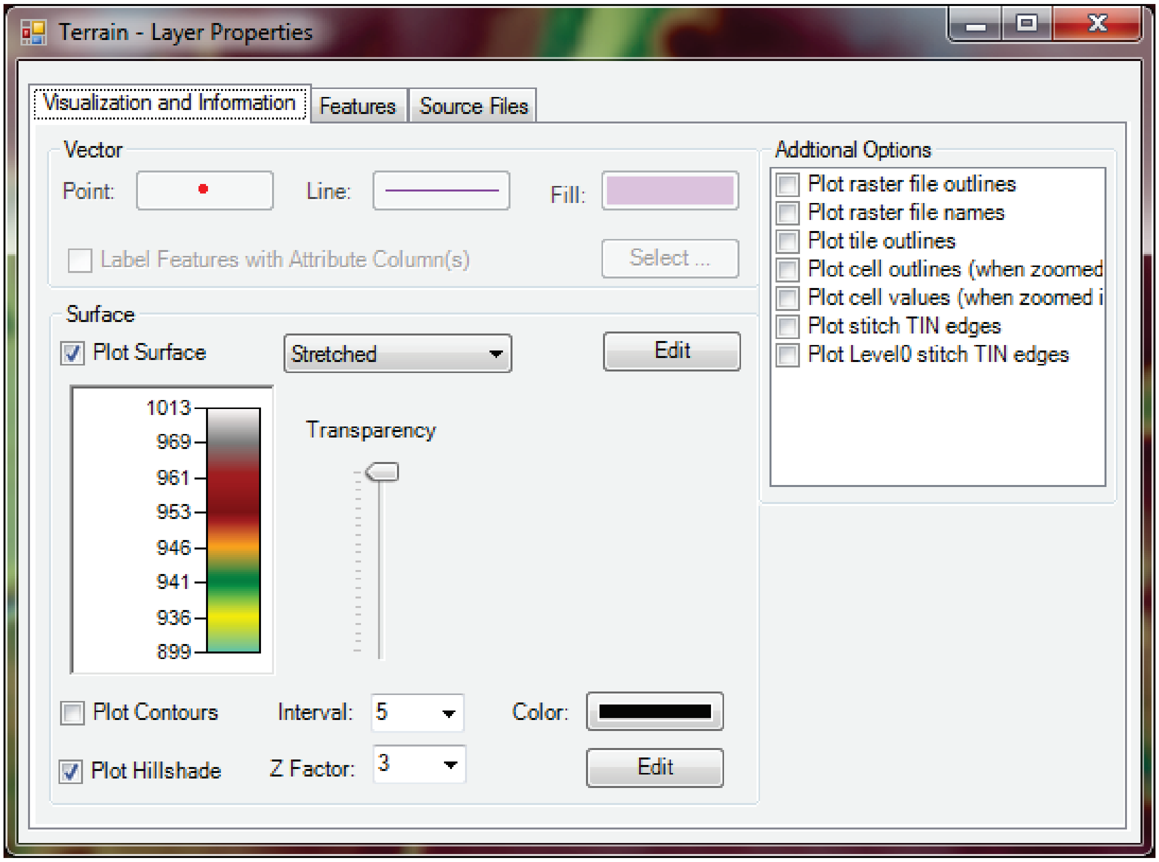


Figure 2-5. Layer Properties Window for the Terrain Data Layer.

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.

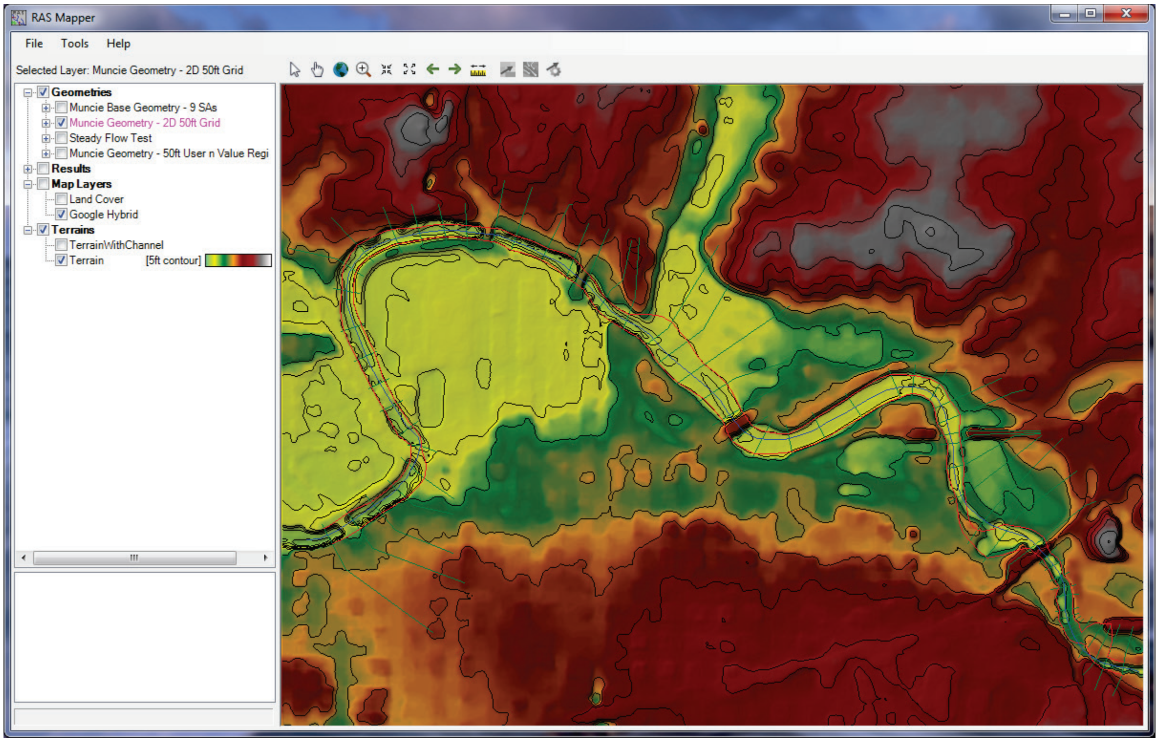


Figure 2-6. Terrain Data with Hill Shading and Contour Lines Turned On.

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. RAS Mapper can now 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.

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 geometry data to be used in creating the channel terrain model. Also turn on the following sublayers: **River (stream centerline)**; **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. This is the information (along with the terrain) that is used to create the new channel geometry. See an example with the desired geometry sublayers selected in Figure 2-7 below.

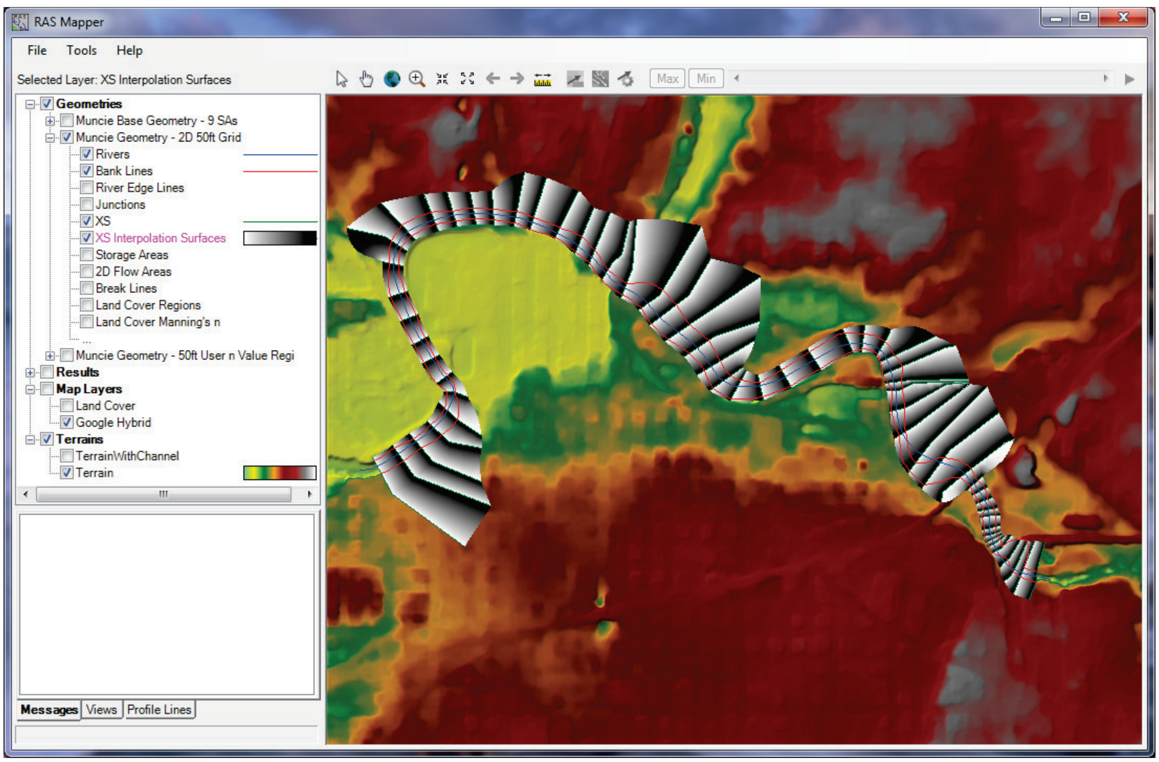


Figure 2-7. RAS Mapper with base terrain and Geometry Layers Displayed.

Once the geometry layers are completed, the channel terrain model is created by right clicking on the Geometry layer 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.

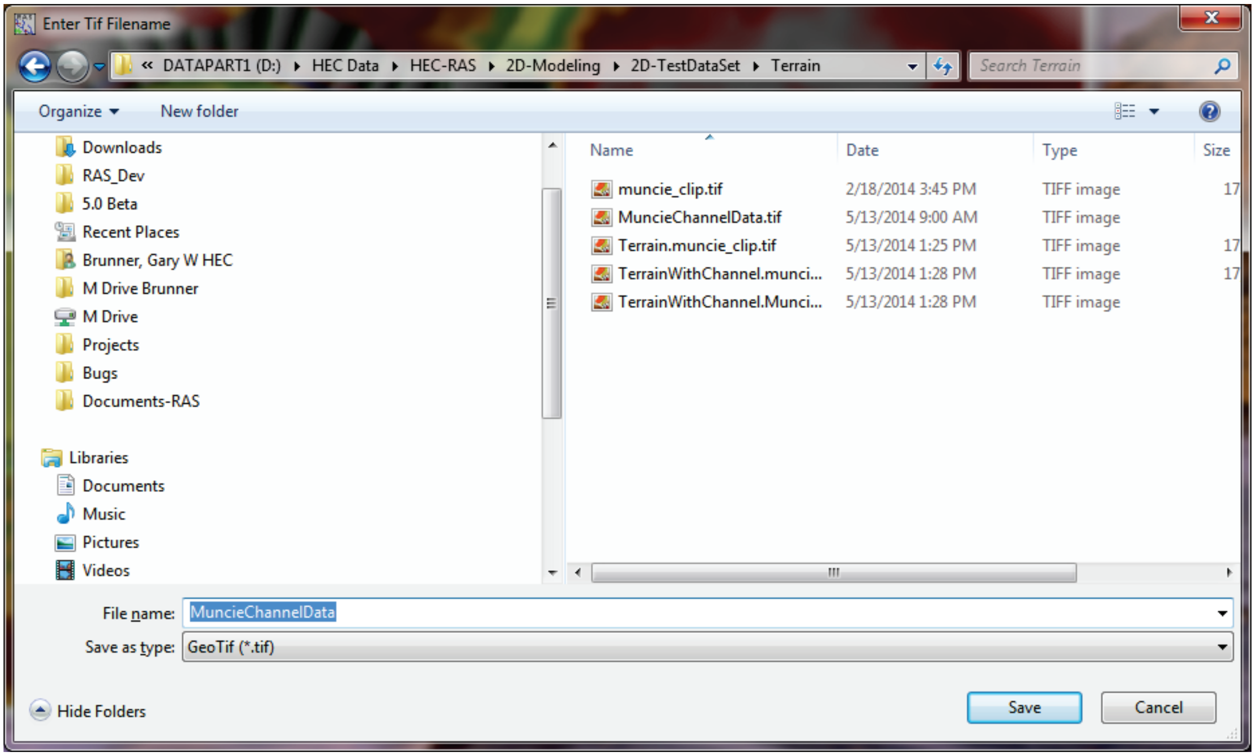


Figure 2-8. Terrain Export File Choose shown with Example Name and Directory.

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 "5.0", then the new terrain model will have grids that are 5 x 5 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.

NOTE: The user may want to make a copy of their 1D river model, then move the main channel bank stations down within the channel, such that the area between the bank stations only represents the portion of the cross section that is new, and not in the existing terrain model. Then when a terrain model is made from the cross section data, between the main channel bank stations, it will only represent new information, that is not in the current terrain model, and thus it will not replace any good data that is already in the terrain model.

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.

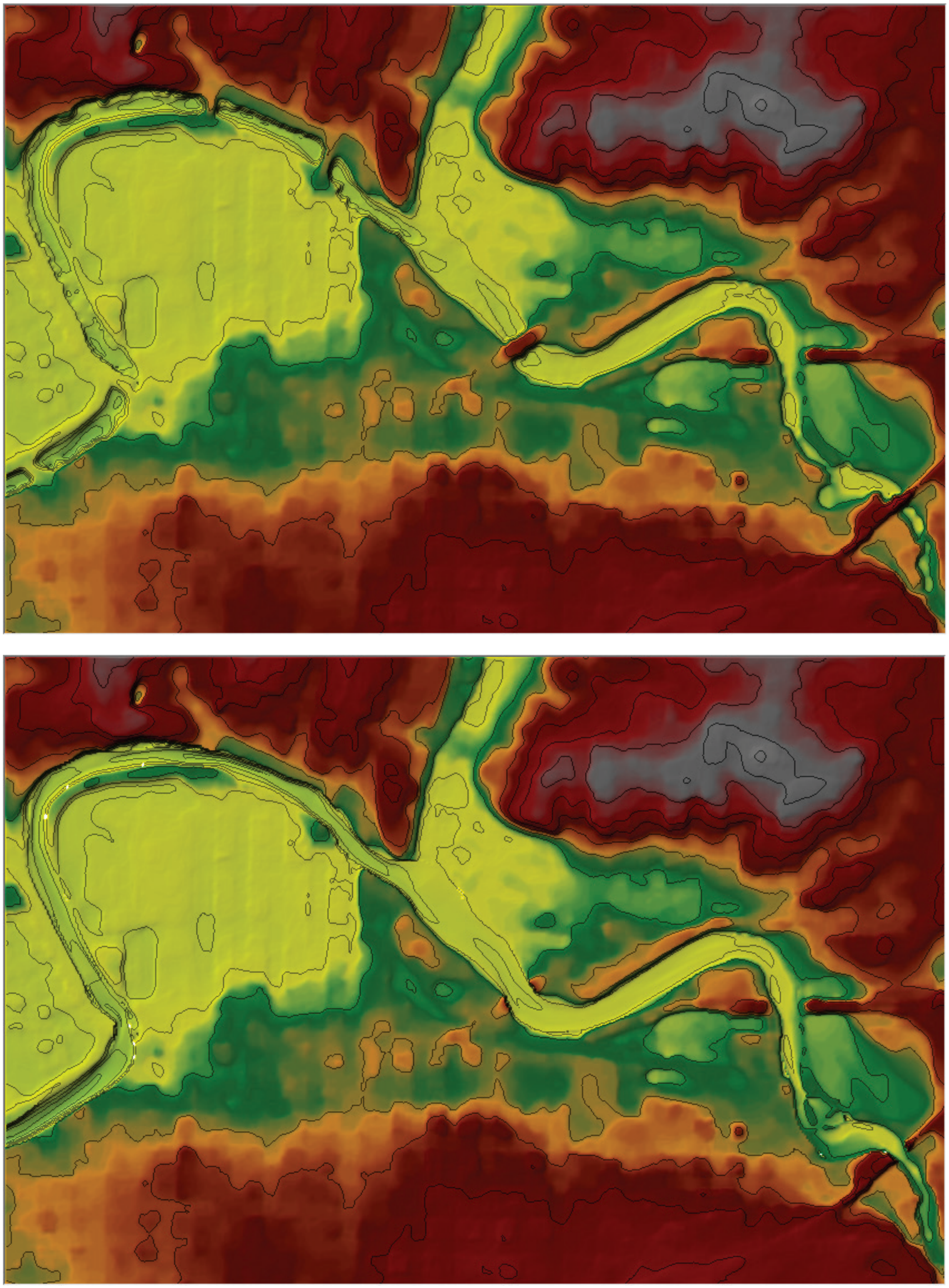


Figure 2-9. Original Terrain model (Top) and New Terrain model with Channel Data (Bottom).

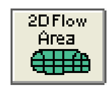
CHAPTER 3

Development of a Combined 1D/2D Model

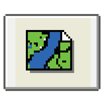
Development of the 2D Computational Mesh

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 and the mesh editing tools. A 2D computational mesh is developed in HEC-RAS 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 2D flow area drawing tool in the Geometric Data editor (just as the user would create a Storage Area). The best way to do this in HEC-RAS is to first bring in terrain data and aerial imagery into HEC-RAS Mapper. Once you have terrain data and various Map Layers in RAS Mapper, they can be displayed as background images in the HEC-RAS Geometry editor. Additionally, the user may want to bring in a shapefile that represents the protected area, if they are working with a leveed system. The background images will assist the user in figuring out where to draw the 2D flow area boundaries in order to capture the tops of levees, floodwalls, and any high ground that will act as a barrier to flow.



Use the background mapping button on the HEC-RAS Geometry editor to turn on the terrain and other Map Layers, in order to visualize where the boundary of the 2D Flow Area should be drawn. If you created a Terrain layer in RAS Mapper, and you want to display it in the geometry editor, after turning that layer on you will need to go to the Geometry editor's **View** menu, then select **Set Schematic Plot Extents**. From this window select the option called **Set to Computed Extents**. This option will reset the extents of the geometric data editor view window to the extents of the terrain model you created and associated to the geometry data.

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 main river to the 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, use the **2D Flow Area** tool (the button on the Geometric Editor **Tools** bar labeled **2D Flow Area**, highlighted in red on Figure 3-1). 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.

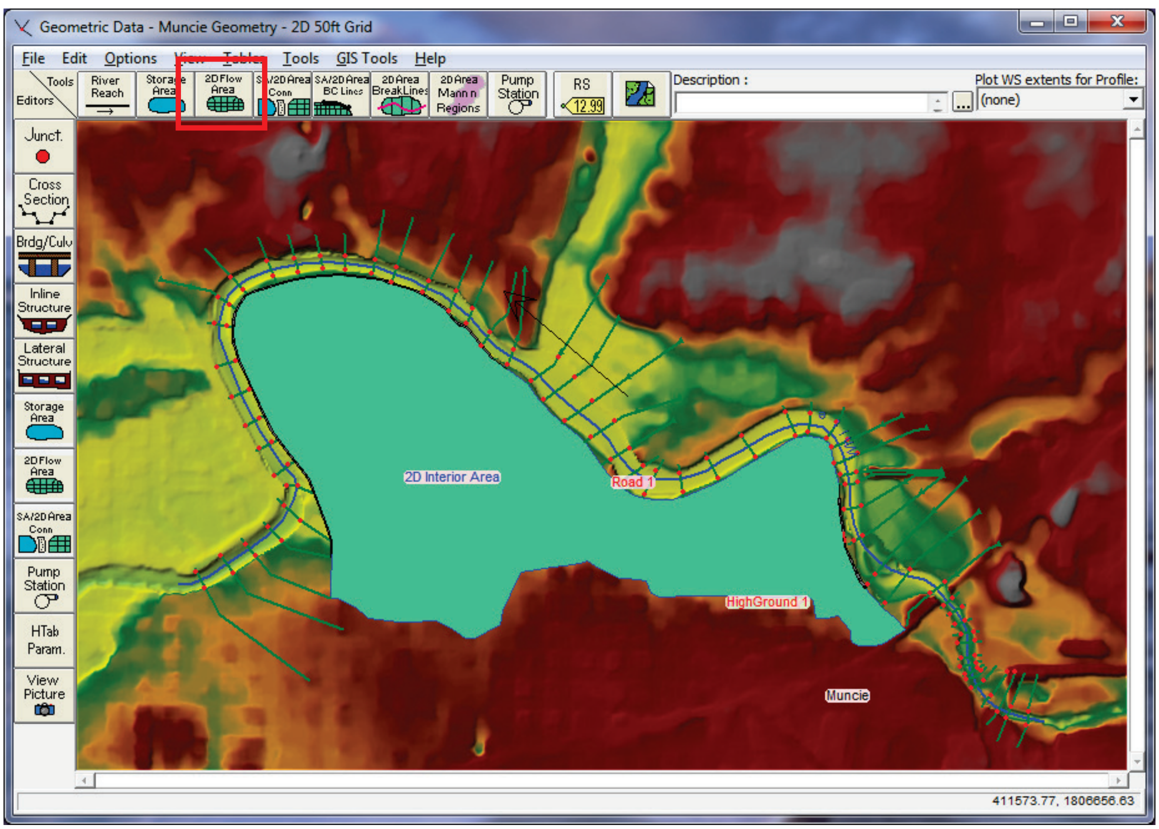
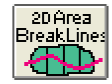


Figure 3-1. Example 2D flow area polygon.

Adding Break Lines inside of the 2D Flow Area



Before the computational mesh is created the user may want to add break lines to enforce the mesh generation tools to align the computational cell faces along the break lines. Break lines can also be added after the main computational mesh is formed, and the mesh can be regenerate just around that break line. In general, break lines should be added to any location that is a barrier to flow, or controls flow/direction.

Break lines can be imported from Shapefiles (**GIS Tools/Breaklines Import from Shapefile)**; drawn by hand; or detailed coordinates for an existing breakline can be pasted into the break line coordinates table (**GIS Tools/Breaklines Coordinates** Table). To add break lines by hand into a 2D flow are, select the **2D Area Break Line** tool (highlighted in Red in Figure 3-2), then left click on the geometry window to start a break line and to add additional points. Double click to end a break line. While drawing a breakline, you can right click to re-center the screen in order to have more area for drawing the breakline. Once a break line is drawn the software will ask you to enter a name for the break line. Add break lines along levees, roads, and any high ground that you want to align the mesh faces along. Break lines can also be placed along the main channel banks in order to keep flow in the channel until it gets high enough to overtop any high ground berm along the main channel. An example of using break lines within a 2D flow area for modeling levees is shown in Figure 3-2.

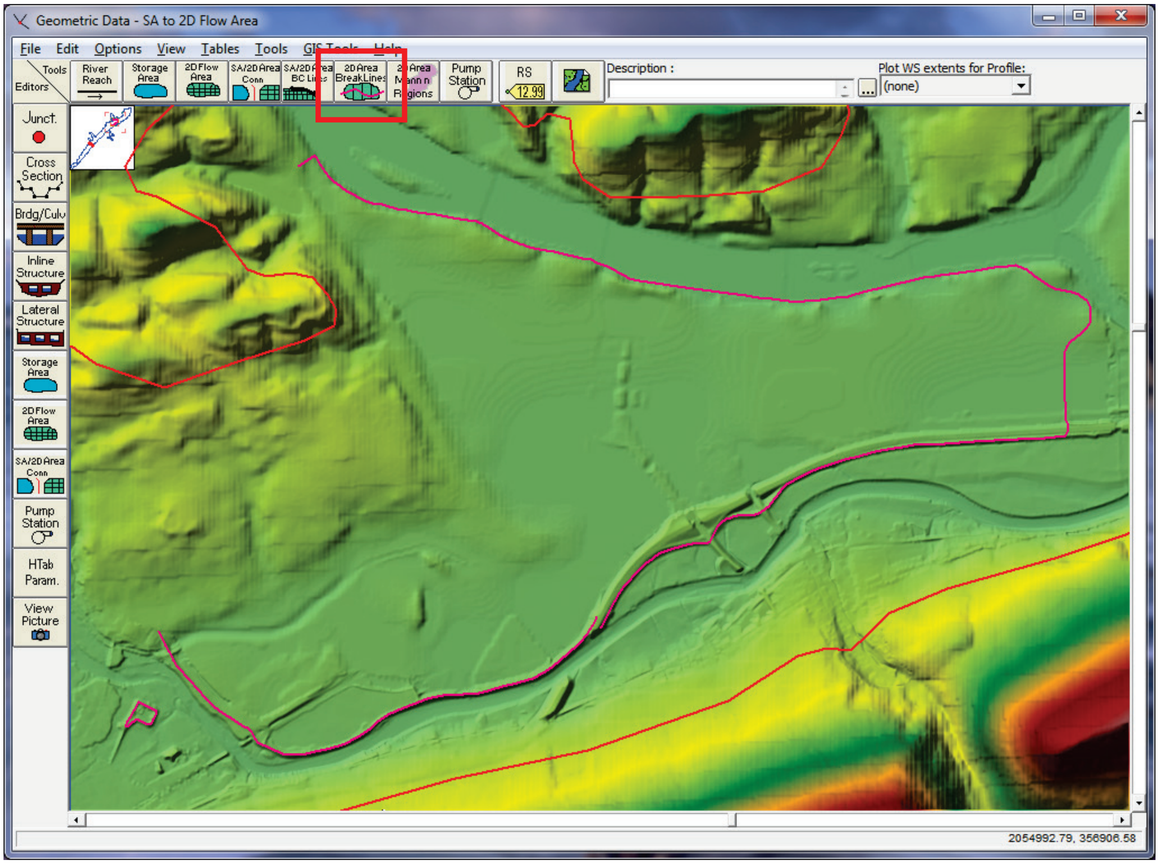


Figure 3-2. Example Break lines for Levees

After all the break lines have been added, the computational mesh can be generated. Keep in mind the user can also add additional break lines after the mesh has been generated, and the computational mesh can be refined around an individual break line at any time. This will be discussed in more detail under the section labeled "Editing/Modifying the Computational Mesh."

Creating 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 3-3).

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.

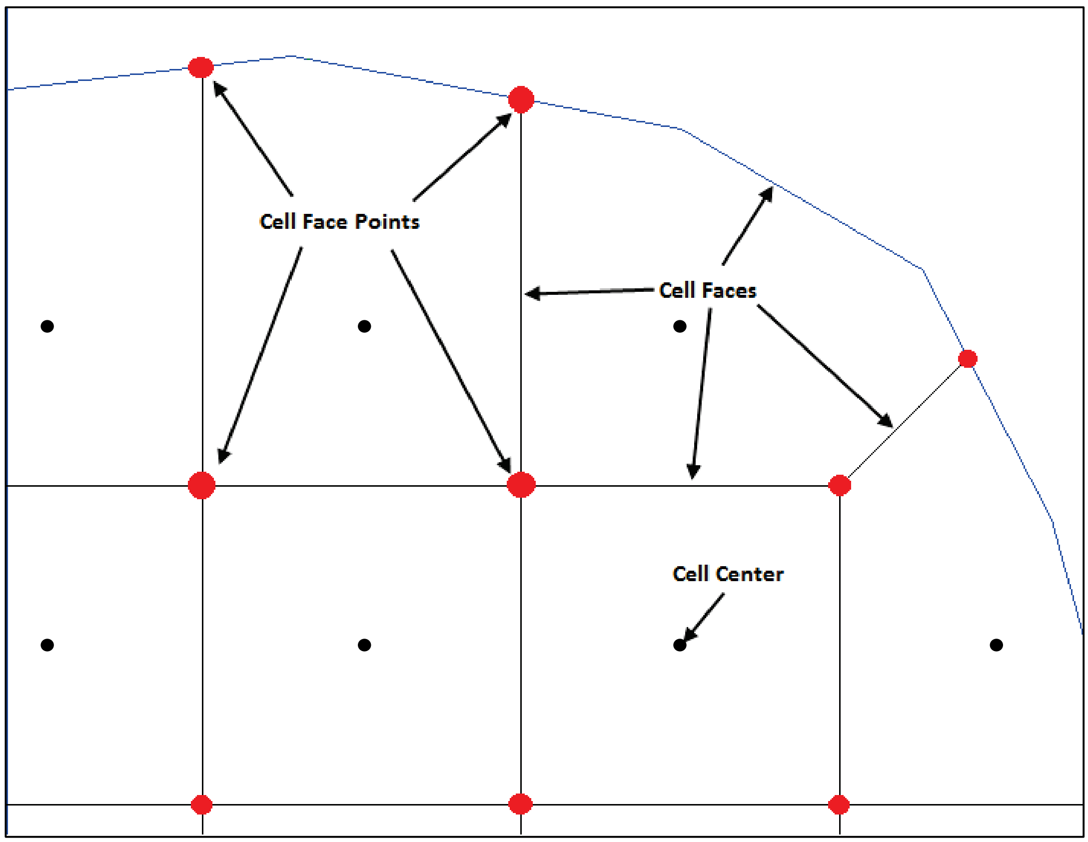


Figure 3-3. HEC-RAS 2D modeling computational mesh terminology.

To create a 2D flow area computational mesh, select the **2D Flow Area** editor button on the left panel of the **Geometric Data** editor (under the **Editors** set of buttons on the left) to bring up the 2D flow area editor window:

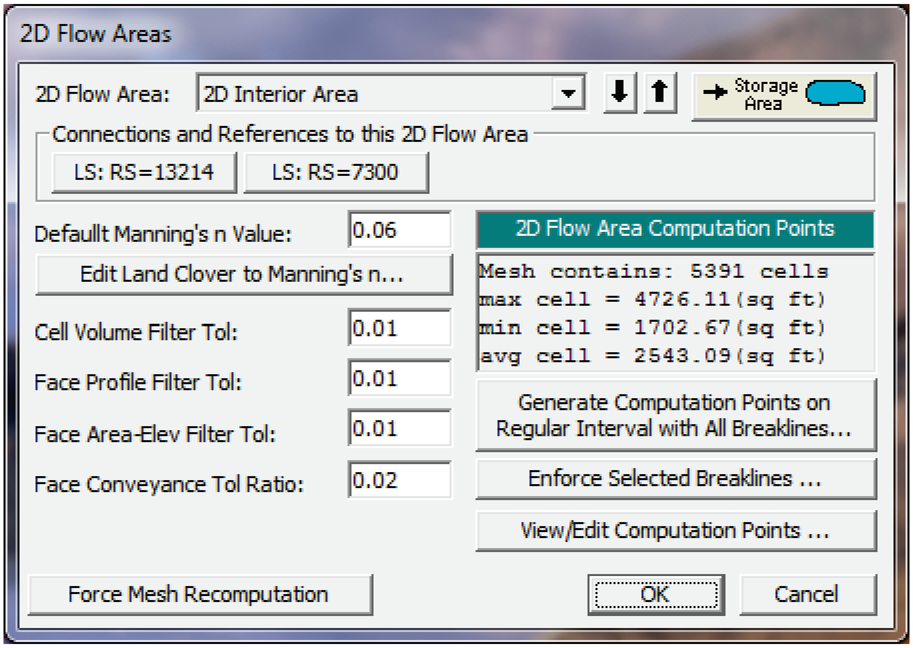


Figure 3-4. 2D flow area Mesh Generation Editor.

The **2D Flow Area** editor allows the user to select a nominal grid size for the initial generation of the 2D flow area computational mesh. To use this editor, first select the button labeled **Generate Computational points on regular Interval ….** This will open a popup window that will allow the user to enter a nominal cell size. The editor requires the user to enter a **Computational Point Spacing** in terms of DX and DY (see Figure 3-5). This defines the spacing between the computational grid-cell centers. For example, if the user enters DX = 50, and DY = 50, they will get a computational mesh that has grids that are 50 x 50 everywhere, except around break lines and the outer boundary. Cells will get created around the 2D flow area boundary that are close to the area of the nominal grid-cell size you selected, but they will be irregular in shape.

Since the user can enter break lines, the mesh generation tools will automatically try to "snap" the cell faces to the breaklines. The cells formed around break lines may not always have cell faces that are aligned perfectly with the break lines. An additional option available is **Enforce Selected Breaklines**. The **Enforce Selected Breaklines** option will create cells that are aligned with the breaklines, which helps ensures that flow cannot go across that cells face until the water surface is higher than the terrain along that break line. When using the Enforce Selected Breaklines option, the software will create cells spaced along the breakline at the nominal cell size entered buy the user. However, the user can enter a different cell spacing to be used for each breakline. This is accomplished by selecting **GIS Tools/Breaklines Cell Spacing Table**, and then entering a user defined cell spacing for each breakline.

The popup editor has an option to enter where the user would like the cell centers to start, in terms of an upper left X and an upper left Y coordinate. These Starting Point Offset fields are not required. By default it will use the upper left corner of the polygon boundary that represents the 2D flow area. Use of the **Shift Generated Points** option allows the user to shift the origin of the grid cell centers, and therefore the location of the cell centers.

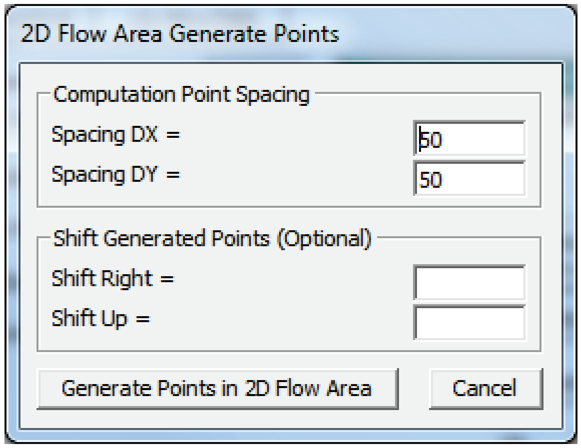


Figure 3-5. 2D Computational Point Spacing Editor.

After the **Computational Point Spacing** (DX and DY) has been entered, press the **Generate Points in 2D flow area** button. 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.

There are five additional fields on the 2D flow areas editor (Figure 3-4) that are used during the 2D pre-processing. These fields are:

Default Manning's n Value: This field is used to enter a default Manning's n values that 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.

There is also a button on this editor labeled **Edit Land Classification to Manning's n**. This button brings up the **Land Classification to Manning's n** table, which allows the user to enter Manning's n values for corresponding land use classifications. This table can also be brought up by going to the "**Tables"** menu and selecting "**Manning's n by Land Classification**".

Cell Volume 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.

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 results 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.

Once a nominal grid size has been selected, breaklines have been entered, a base Manning's n-value has been entered, and tolerances have been set, the user should press the **OK** button to accept the data and close the editor. When the **OK** button is selected the software automatically creates the computational mesh and displays it in the **Geometric Data Editor** graphics window (see Figure 3-6).

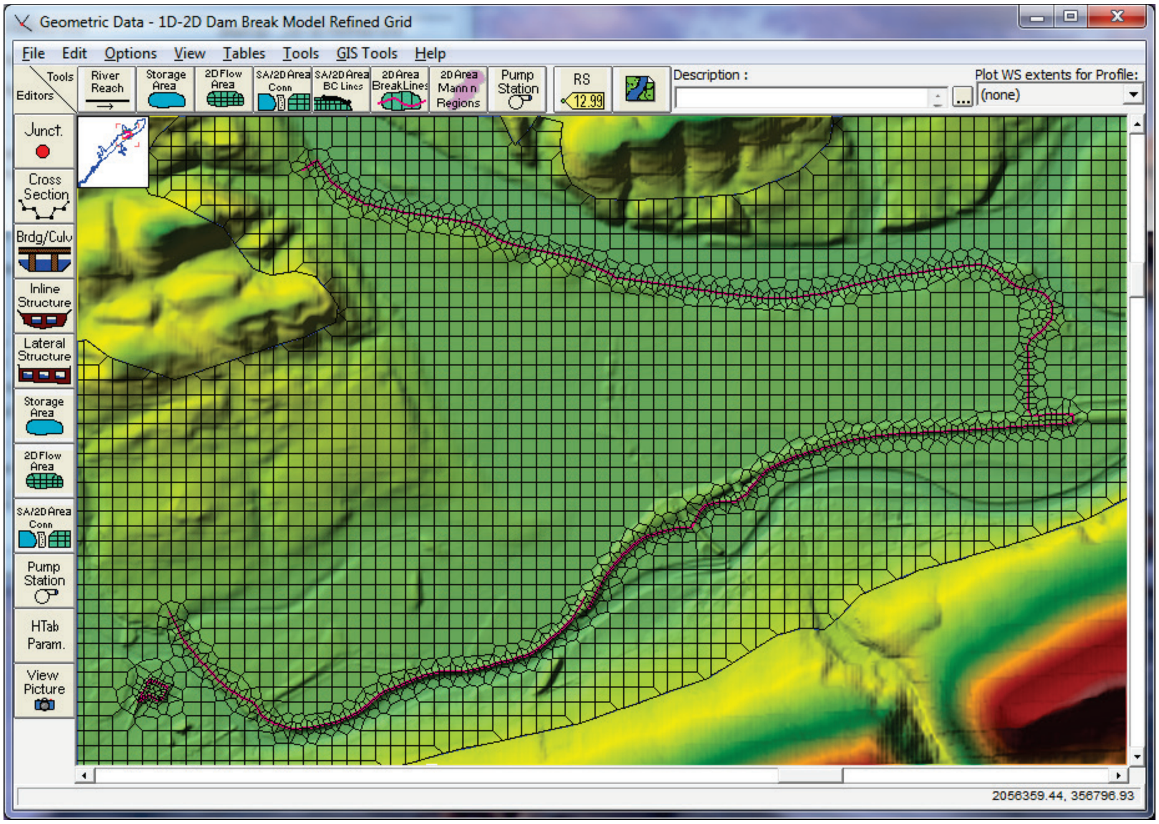


Figure 3-6. Example 2D computational mesh for a levee protected area.

As mentioned previously, cells around the break lines and the 2D flow area boundary will typically be irregular in shape, in order to conform to the user specified break lines and boundary polygon. The mesh generation tools utilize the irregular boundary, as well as try to ensure that no cell is smaller in area than the nominal cell size. The cells around the 2D flow area outer boundary will be equal to or larger than the nominal cell size; therefore, if a boundary cell is going to be smaller than the nominal cell size it gets combined with a neighbor cell. Shown in Figure 3-7, is a zoomed in view of a mesh with break lines on top of levees.

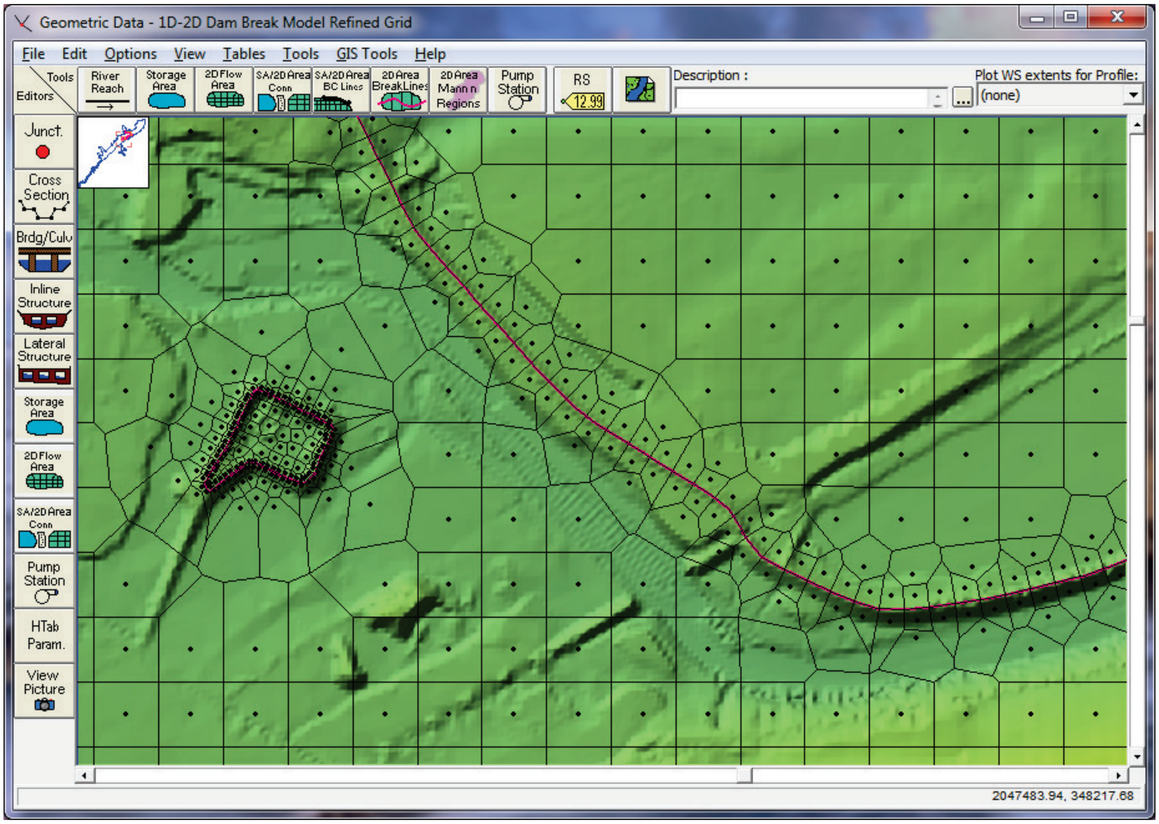


Figure 3-7. Zoomed in view of the 2D flow area computational mesh.

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 also 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 taken into account 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.

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

Break Lines

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. The user can draw a new break line, then left click on the break line and select the option **Enforce Break line in 2D Flow Area**. 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).

Additionally the user can control the size/spacing of cells along the break line. To control the cell spacing along a break line, right click on the break line and select the option **Edit Break Line Cell Spacing**. A window will appear allowing the user to enter a minimum and maximum cell spacing to be used when forming cells along that break line. The minimum cell spacing is used directly along the break line. 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 nominal cell size being used for the mesh. The user can enter a Maximum cell size if desired. If no maximum cell size is entered, the software automatically transitions the cells from the minimum cell size around the break line, to the default mesh cell size. To enforce the new cell spacing, the user must select the **Enforce Break line in 2D Flow Area** option, after entering the break line cell spacing. Break line cell spacing's are saved, such that if the mesh is regenerated, the user defined break line cell spacing will automatically be used. User can also bring up a table that will show all of the break lines and any user entered break line cell spacing values. To open this table, select **GIS Tools**, then **Break Lines Cell Spacing Table**. Once the table is open, users can add or change break line cell spacing values from the table. Then if the user regenerates the whole mesh, or just the area around a specific break lines, the new break line cell spacing will be used.

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. However, transitions from a larger cell size immediately to a smaller cell size, may not produce the most accurate computational model. So it is better to transition cell sizes gradually. The HEC-RAS mesh generation tools allow the user to enter a minimum and a maximum cell spacing to use around break lines. The mesh generation tools will automatically transition from the smaller cell size right at the break line to the larger cell size away from the break line. See an example of this in Figure 3-8 below.

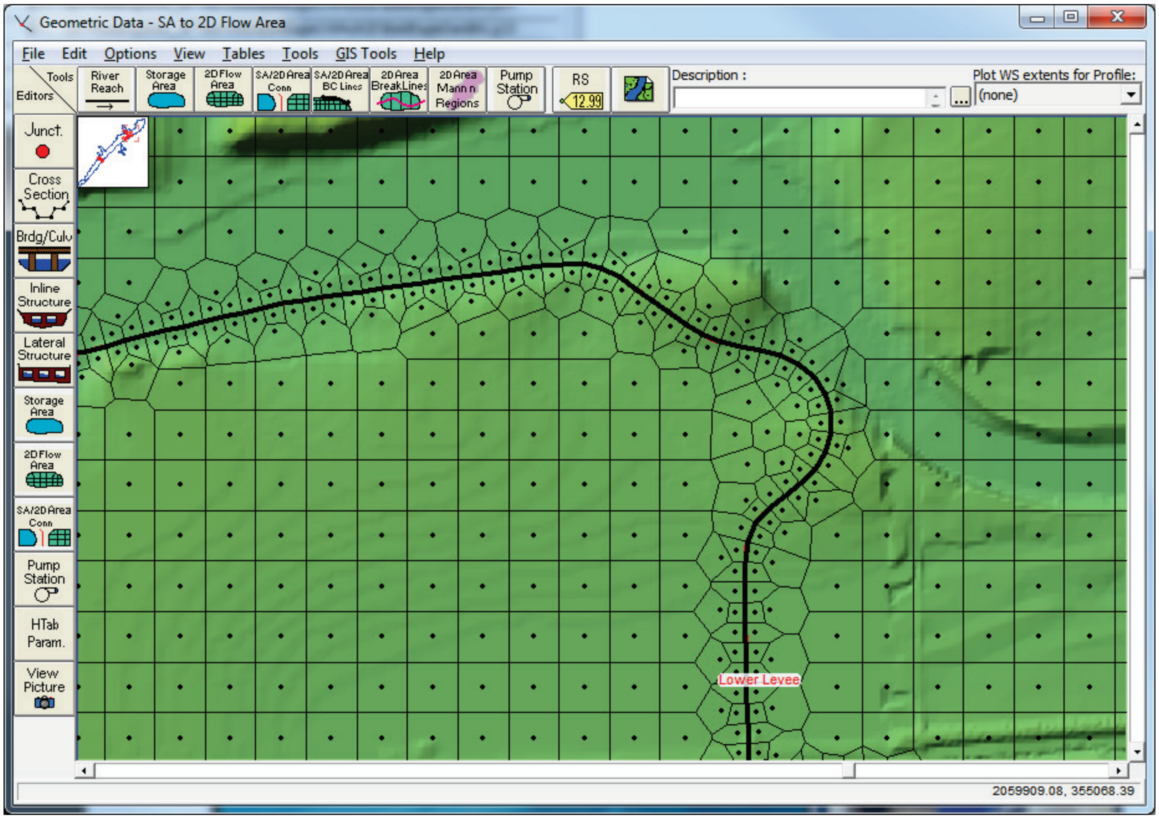


Figure 3-8. Example of gradually transitioning cell sizes away from the break line.

Hand Based Mesh editing Tools

The hand editing mesh manipulation tools are available under the **Edit** menu of the HEC-RAS Geometric Data 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.

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.

HEC-RAS makes the computational mesh by following the Delaunay Triangulation technique and then constructing a Voronoi diagram (see Figure 3-9 below, taken from the Wikimedia Commons, a freely licensed media file repository):

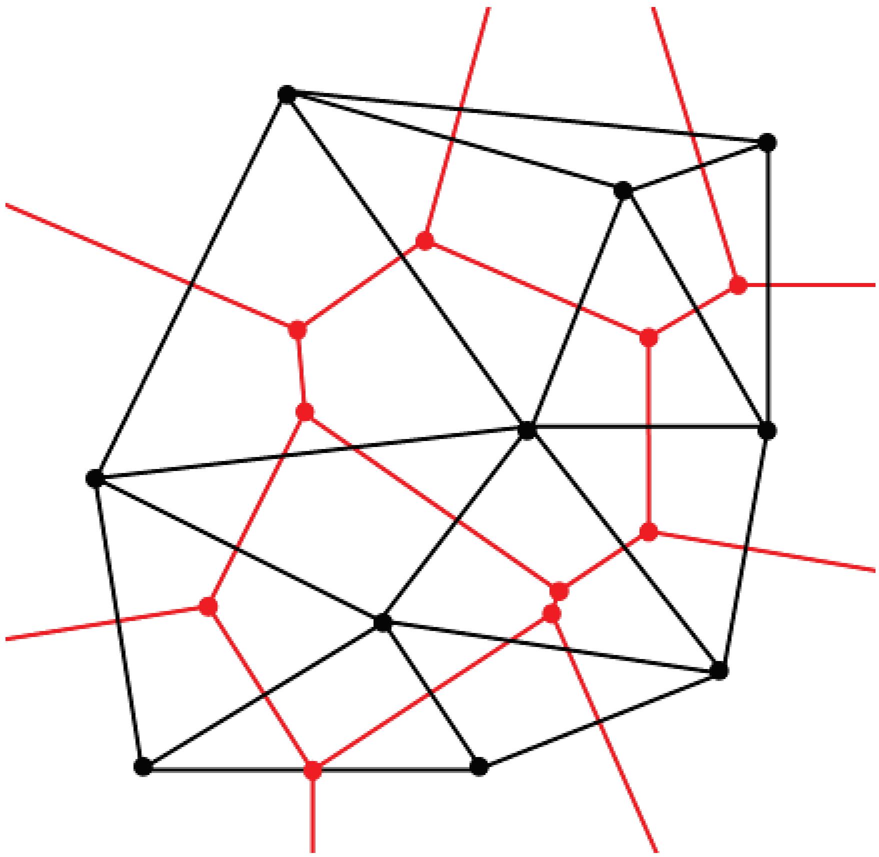


Figure 3-9. Delaunay - Voronoi diagram example.

The triangles (black) shown in Figure 3-9 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.

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 3-10). 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.

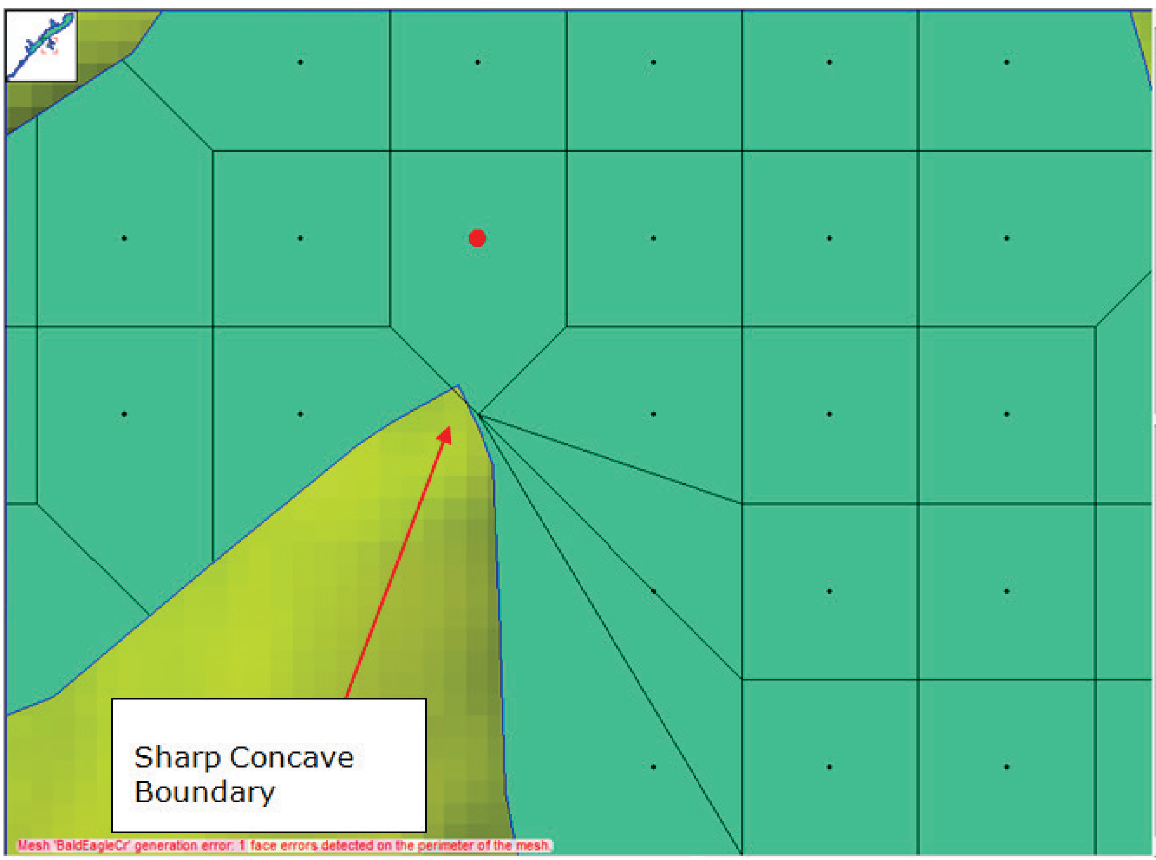


Figure 3-10. Example of a Sharp Concave Boundary Causing Mesh Generation Problem.

Shown in Figure 3-11 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.

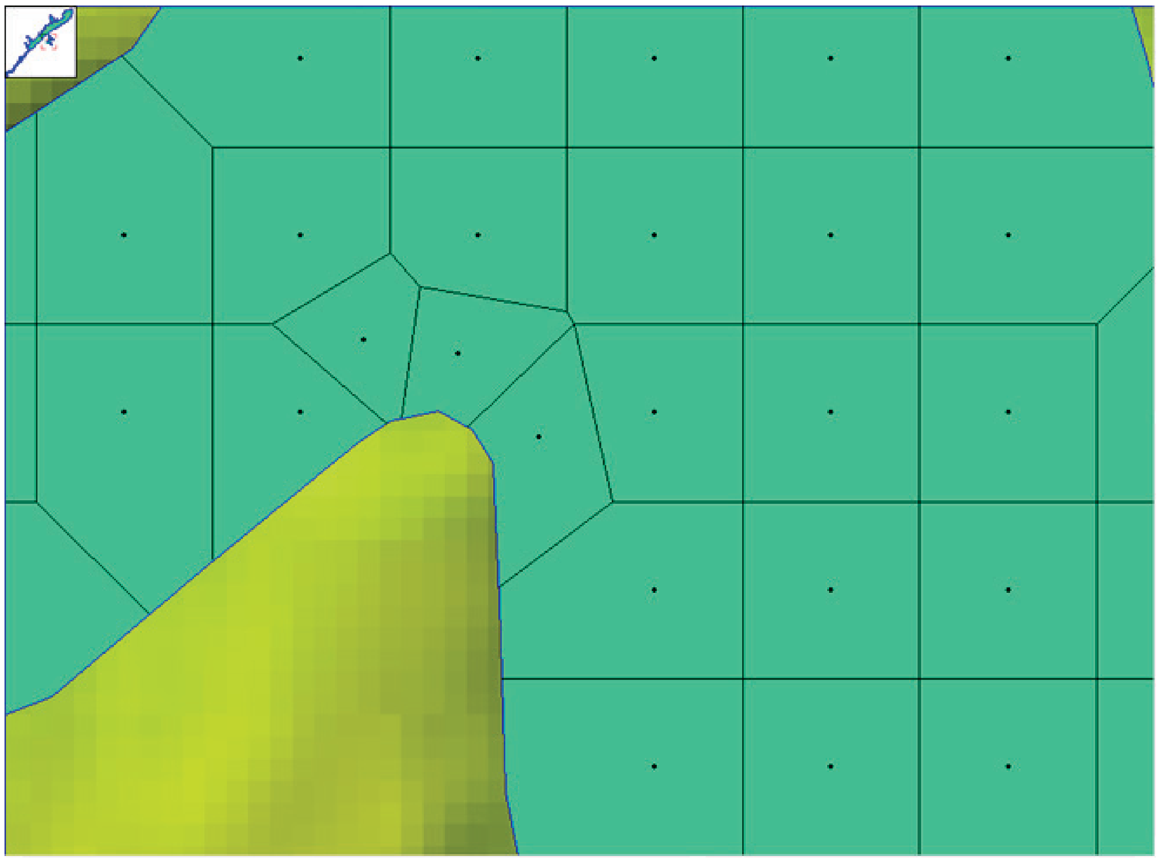


Figure 3-11. Corrected Mesh with Smoothed Boundary and Additional Cell Centers.

- **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 3-12. 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.

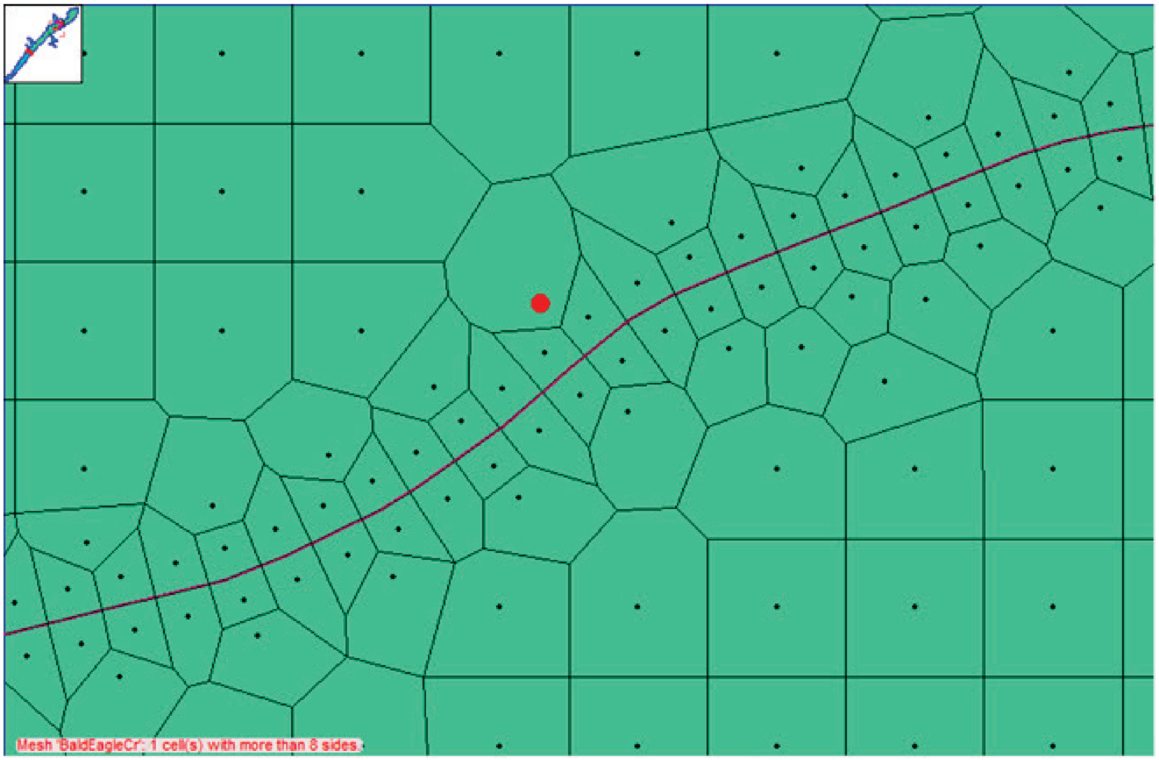


Figure 3-12. Example Mesh with a Cell that has more than 8 sides.

The mesh problem shown in Figure 3-12 (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 3-13.

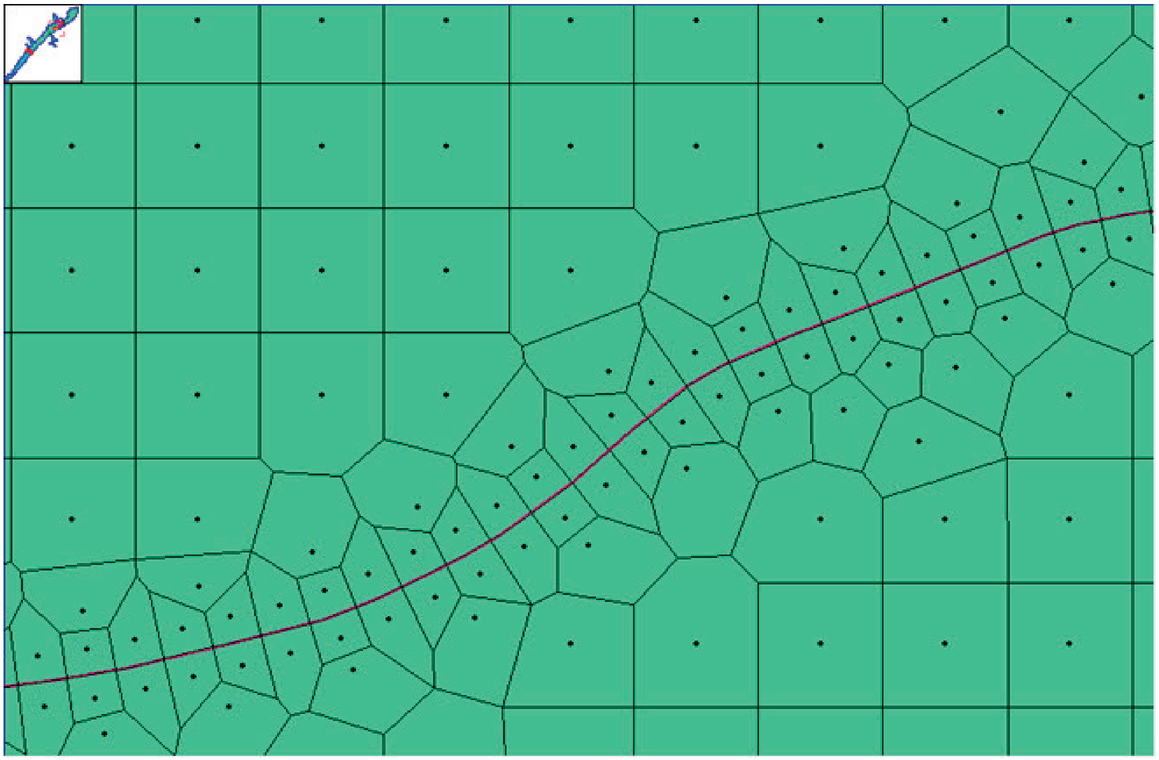


Figure 3-13. Final mesh after adding additional points to fix mesh problem.

- **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 3-14) 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 3-15.

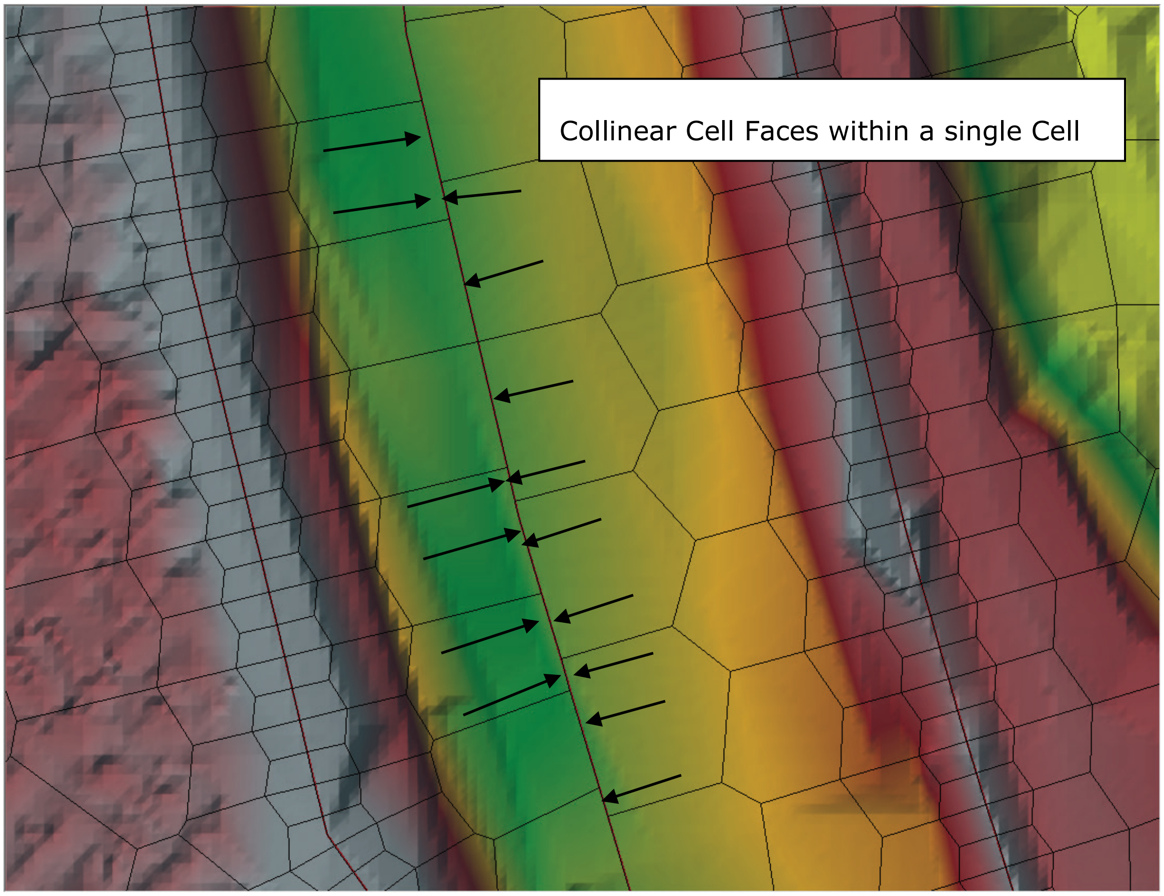


Figure 3-14. Example of bad cells with two or more faces that are collinear.

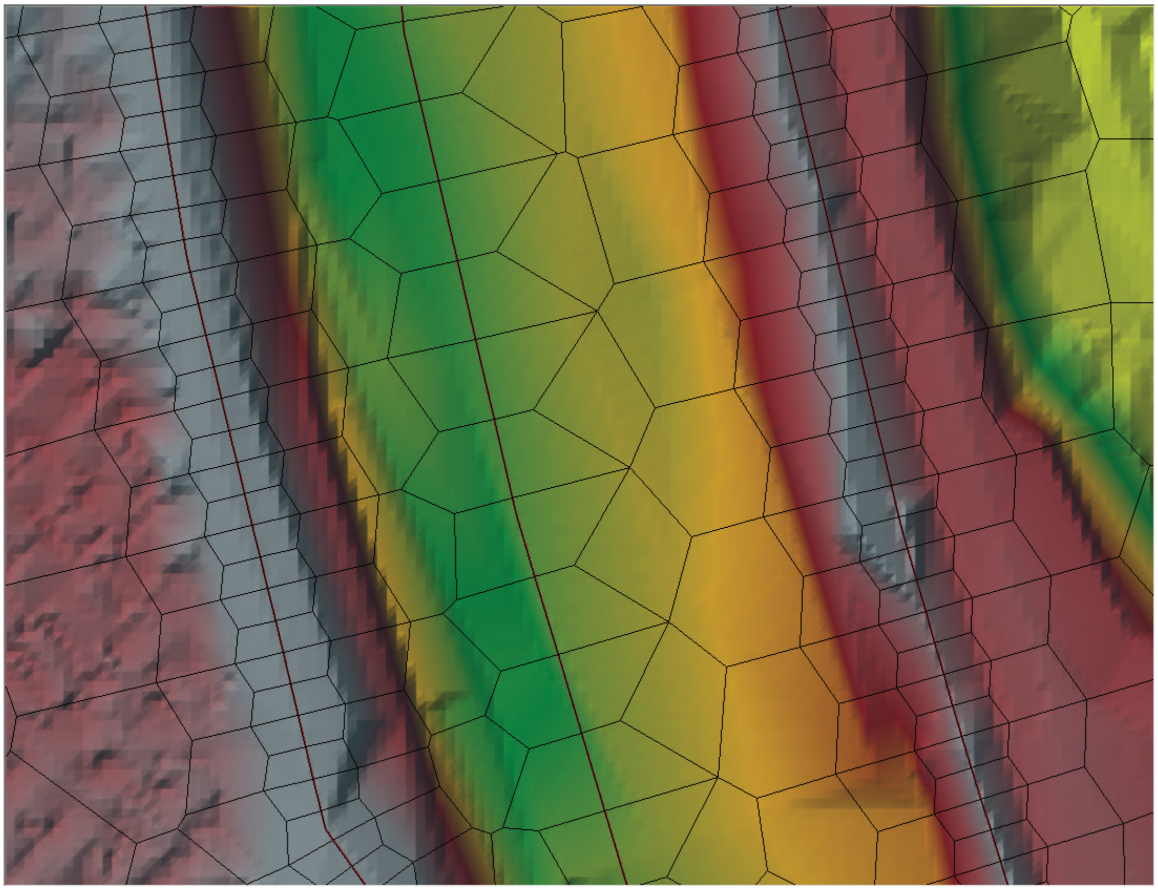


Figure 3-15. Example of properly formed cells along the centerline break line.

Creating a Spatially Varied Manning's Roughness Layer

A spatially varying land cover data set can be created in RAS Mapper, and then associated with a specific geometry data set. Once a land cover data set is associated with a specific geometry file, the user can specify Manning's n values to be used for each land cover type. Additionally, the user can create their own 2D area Manning's n value regions (user defined polygons), in which they can override the Manning's n values from the land cover data set. User defined 2D Area Manning's n value regions can also be used to assist in calibrating a model for specific regions of a 2D flow area.

***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 define Manning's n Regions.**

In the current version of HEC-RAS, users can import land use information in both polygon (shapefile) and gridded formats. Shapefile layers can be created by users outside of HEC-RAS (i.e., in ArcGIS), then imported into RAS Mapper. Gridded land used data can be obtained from USGS websites (NLCD 2011 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 2011 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, buildings, 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 2D flow 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 used coverage (Layer) and stores it as a GeoTIFF file.

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 utilize spatially varying roughness within HEC-RAS, go to RAS Mapper, then from the **Tools** menu, select **New Land Cover** or right click on **Map Layers** and select **Add New 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; Selected File Land Cover Identifiers;** 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 Selected File Land Cover Identifiers 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 Land Cover Identifiers, their numeric ID, and optionally a user entered Manning's n value for each Land Cover type.

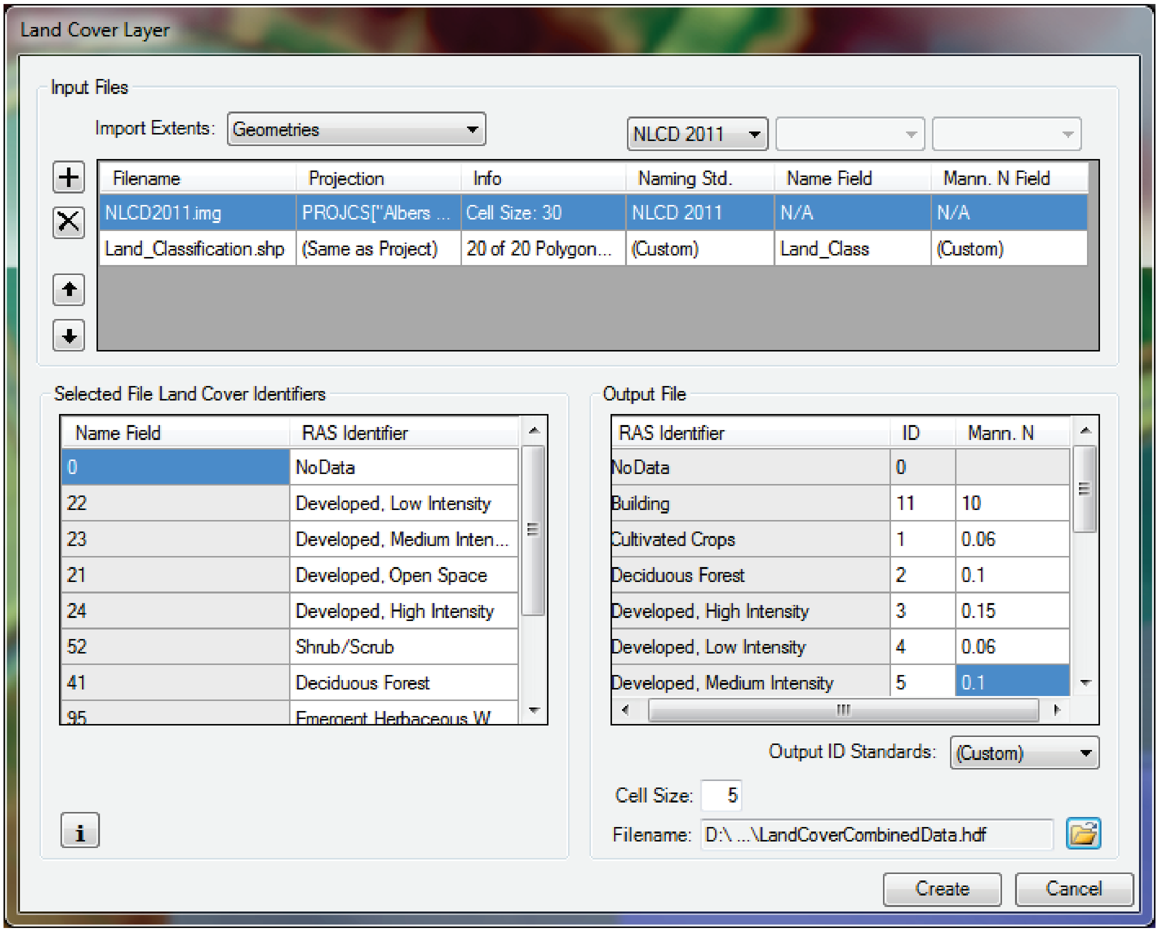


Figure 3-16. RAS Mapper New Land Classification Layer Editor.

The window shown in Figure 3-16 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 2011 (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 Custom (which is the user defined option).

From the Input File section of the editor, if the user selects (highlights) a shapefile, then the **Name Field** and the **Mann. N Field** columns will be active, and the user can select a column contained in the file 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. The options for the Naming standard are: NLCD 2011; Anderson II, and Custom.

When you select a land cover layer in the Input File section of the editor**,** the Selected File Land Cover Identifiers 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, and Manning's n values. Manning's n values could have been selected from a Shapefile, or the user can edit/enter them directly into this table. Manning's n values entered into this table will be used as the default values for a given land use type. The user can override these Manning's n values within any specific geometry file.

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 3-17 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.

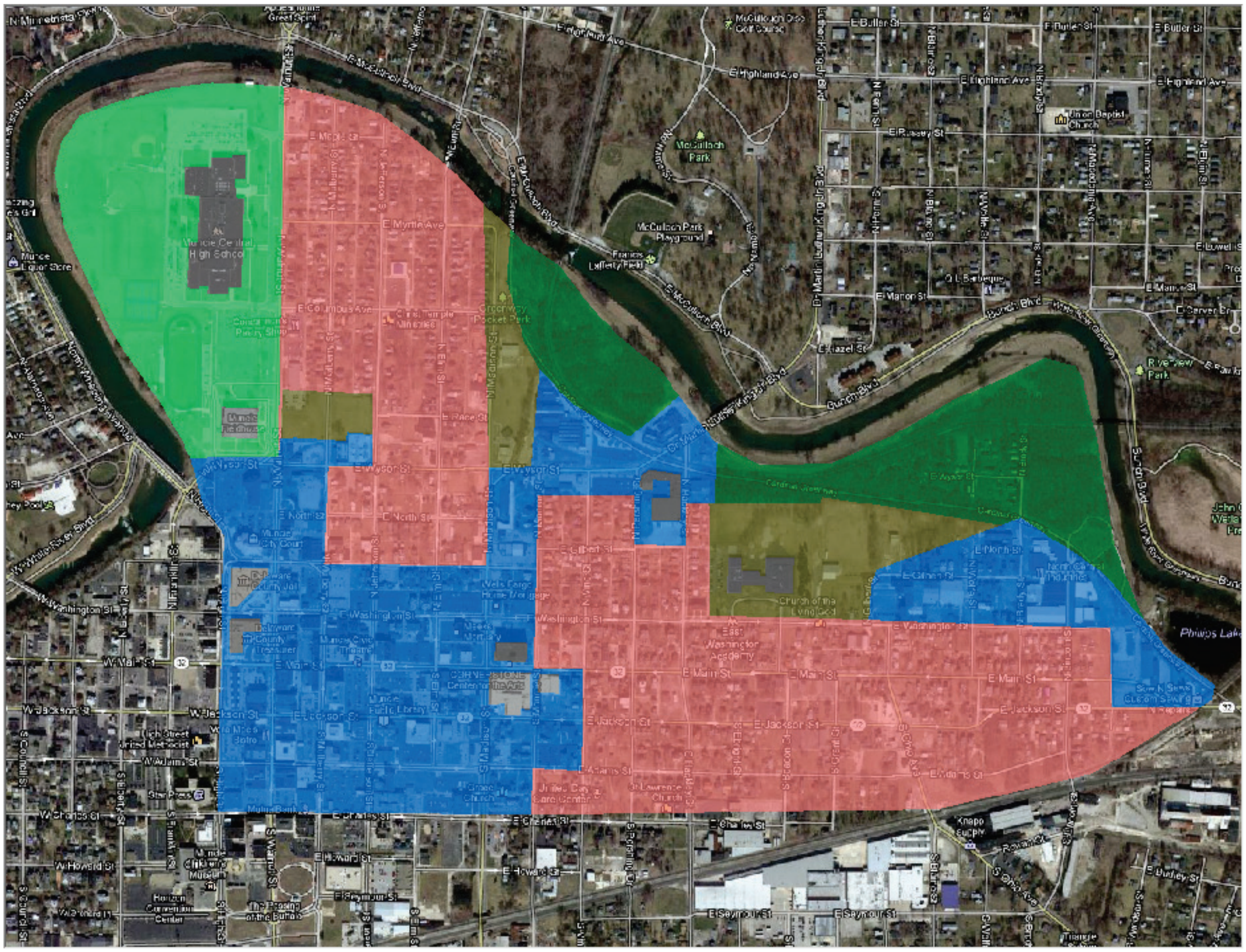


Figure 3-17. Polygon Shapefile containing Land Classification values.

Once the user has created a Land Cover layer in the *.tif file format, then they need to associate that data 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 desired geometry layer (on the left side of the RAS Mapper window) and select **RAS Geometry Properties**. This will bring up a window that will allow the user to select both the desired Terrain Layer and Land Cover to associate with this geometry file (See the Figure 3-18 below).

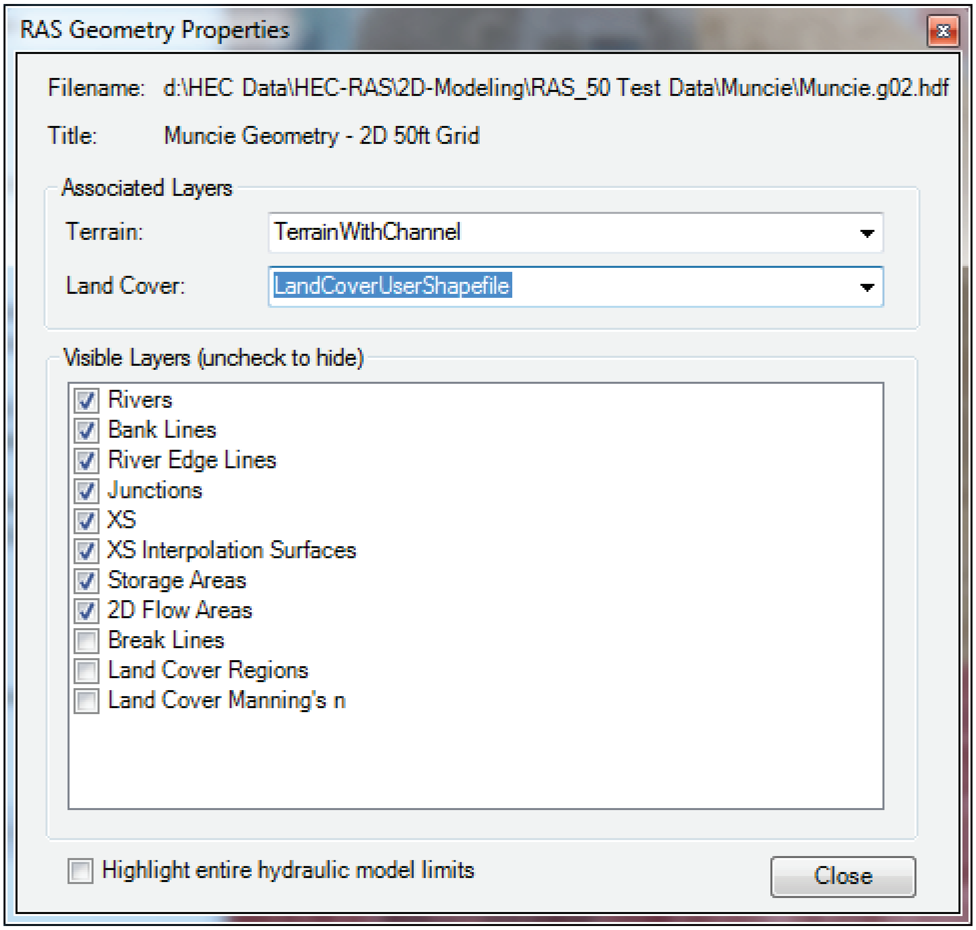
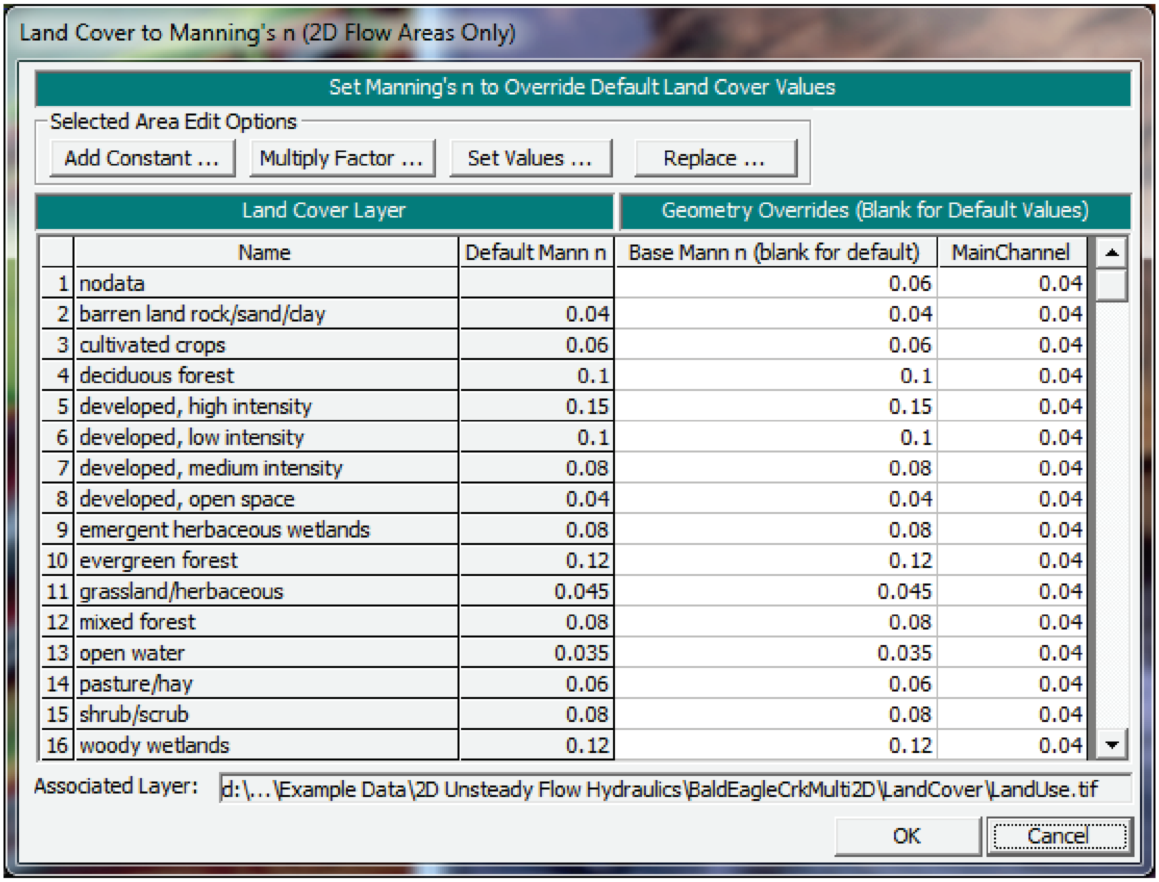


Figure 3-18. Geometry Properties editor for associating Terrain and Land Classification data.

Once a Land Cover layer is associated with a geometry file, 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. This Land Cover versus roughness table is developed from within the HEC-RAS geometric data editor for a specific geometry file. Go to the **RAS Geometry Editor** and open the geometry of interest (i.e. the geometry that has been associated with a Land Classification). From the **Tables** menu select the option labeled "**Manning's n by Land Cover**." This option will bring up a table that shows all the land cover identifiers that are contained in the associated Land Cover layer (Figure 3-19). The editor contains three or more columns. The first two columns are the Land Cover Name and the default Manning's n values based on that land cover data set. The third column is available to allow the user to override the default Manning's n values with what they would like to have as their base Manning's n values for this specific geometry file. The fourth and subsequent columns will show up if the user has defined "**2D Area Manning's n Regions**".

The Land Cover Name column contains all of the unique classification names that it found in the Land Cover layer. Each unique Land cover type is listed once in the table, even though there may be many polygons or grid cells with that land cover layer. The user must enter a Manning's n value for each land use type in the table (including the **NoData** row). Default values may already be in the table, if they were define when creating the land cover layer. The user can change any default values to what they want for this specific geometry file.



	Name	Default Mann n	Base Mann n (blank for default)	MainChannel
1	nodata		0.06	0.04
2	barren land rock/sand/clay	0.04	0.04	0.04
3	cultivated crops	0.06	0.06	0.04
4	deciduous forest	0.1	0.1	0.04
5	developed, high intensity	0.15	0.15	0.04
6	developed, low intensity	0.1	0.1	0.04
7	developed, medium intensity	0.08	0.08	0.04
8	developed, open space	0.04	0.04	0.04
9	emergent herbaceous wetlands	0.08	0.08	0.04
10	evergreen forest	0.12	0.12	0.04
11	grassland/herbaceous	0.045	0.045	0.04
12	mixed forest	0.08	0.08	0.04
13	open water	0.035	0.035	0.04
14	pasture/hay	0.06	0.06	0.04
15	shrub/scrub	0.08	0.08	0.04
16	woody wetlands	0.12	0.12	0.04

Figure 3-19. Manning's n by Land Use Classification Table from Geometric Data Editor.

In addition to defining Manning's n values by Land Cover, the user has the option to create their own 2D flow area Manning's n value regions. These regions are user defined polygons that can be used to override the base Manning's n values within that polygon. They can also be used to calibrate a model. The user defined 2D flow area Manning's n regions are defined in the Geometric data editor, and apply only to that geometry file. To create 2D flow area Manning's n regions, select the button at the top of the Geometric data editor labeled **2D Area Mann n Regions**. When this button is selected, the cursor will change to a drawing pencil and the user can draw a polygon on the schematic for the location (area) where they want to define a new 2D flow area Manning's n value region. Single click to start drawing the polygon, single click to add additional vertices, and double click to end the polygon. Once the user has finished creating the polygon, the software will ask the user to enter a name for this region. Each 2D flow area Manning's n value region must have a unique name. The user can create as many of these regions as they want. Once you have entered all of the desired 2D flow area Manning's n value regions, go to the **Tables** menu and select **Manning's n by Land Cover**. The Manning's n by Land Cover table will now contain additional columns for each of the user define 2D flow area Manning's n value regions. The user can enter new Manning's n values to be used for each land cover type. However, the Manning's n values entered in the user define regions will only be applied within the polygon for that region. The 2D flow area Manning's n value regions option can be used to uniquely define Manning's n values, or they can be used to adjust multiple Manning's n values within an area in order to calibrate the model. Manning's n values entered for any user defined region will override the base Manning's n values in that region. Shown in Figure 3-20 is a 2D flow area with a Land Cover data set for the entire area, and a user defined 2D Area Manning's n value region for the main channel.

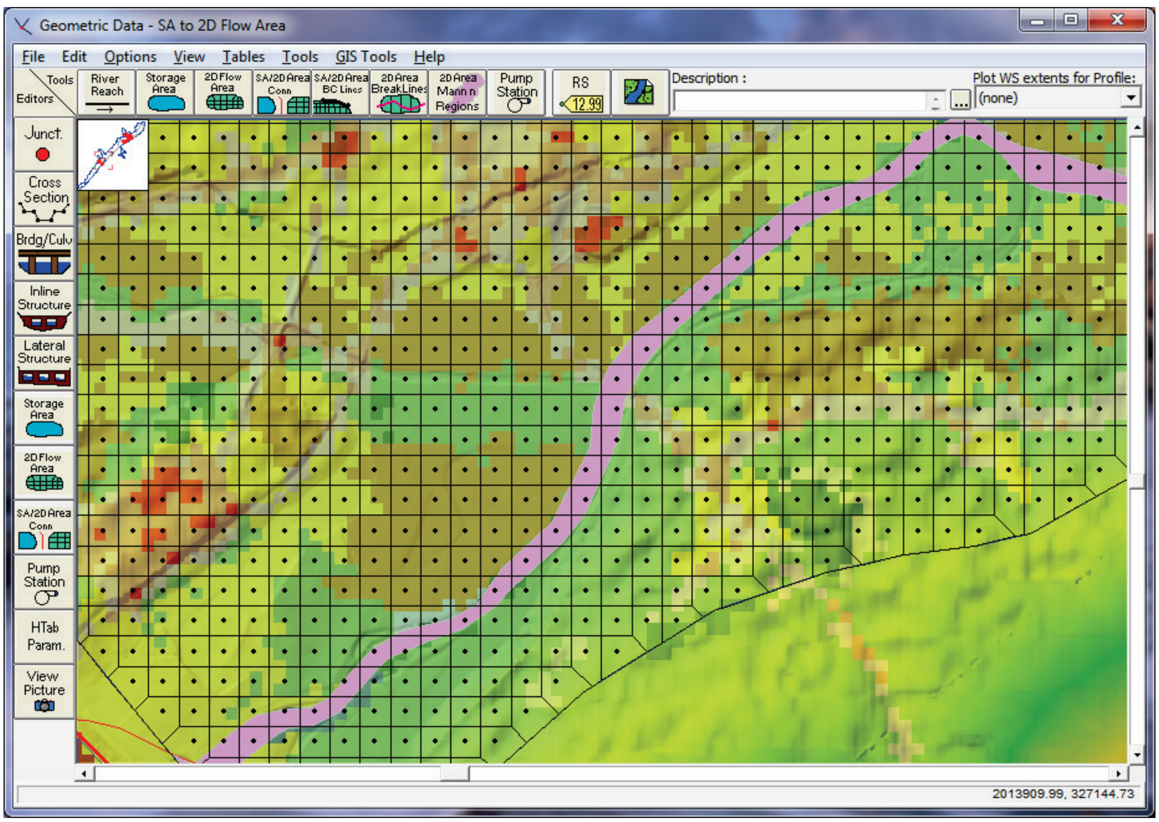


Figure 3-20. Example Land Cover data set and User Defined 2D Area Manning's n value Region.

This Manning's n by Land Cover table 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 (version 5.0), 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.

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 Layer with a Geometry File

After a new terrain layer is added, the user **must** associate the terrain layer 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 3-21, 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.

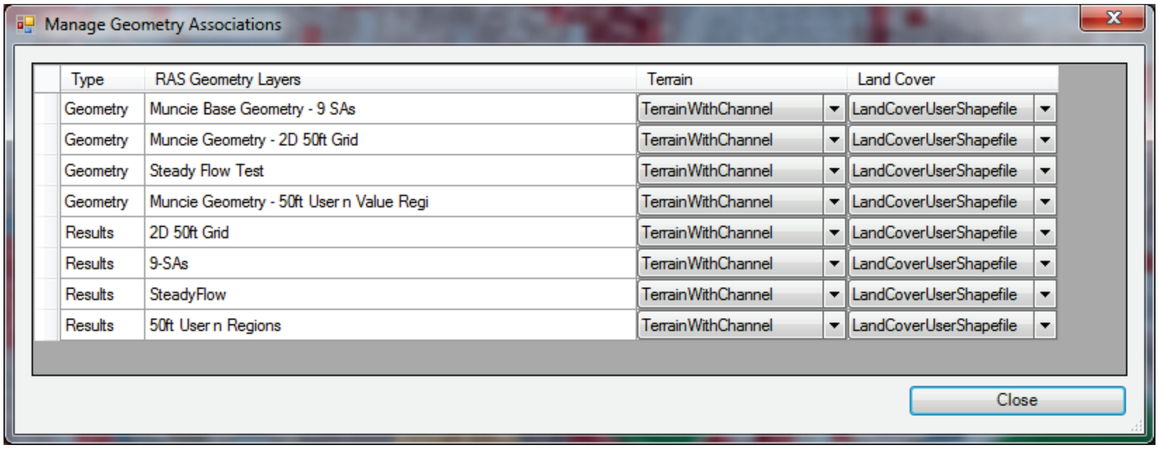


Figure 3-21. Terrain Association Editor.

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

2D Cell and Cell Face 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 3-22 through 3-25. Shown in Figure 3-23 are the details of the underlying terrain within a single computational cell.

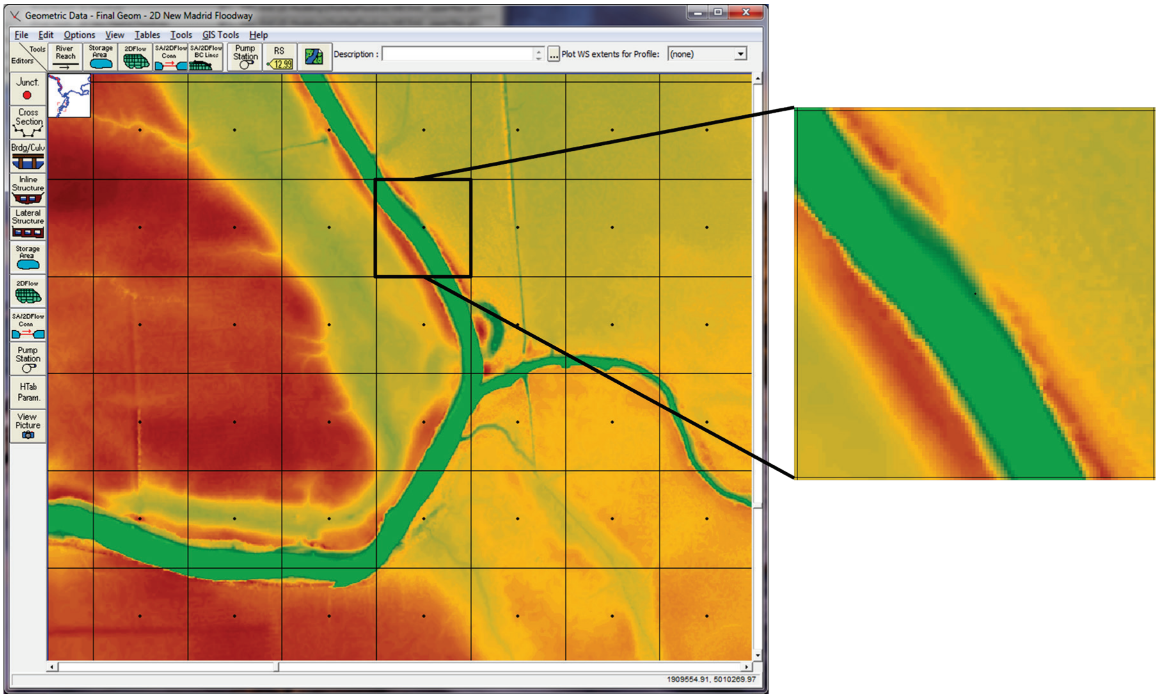


Figure 3-22. Details of Underlying Cell Terrain Data.

When the 2D Geometric Pre-processor runs a detailed elevation-volume relationship is developed for each cell. See an example of this in Figure 3-23 below.

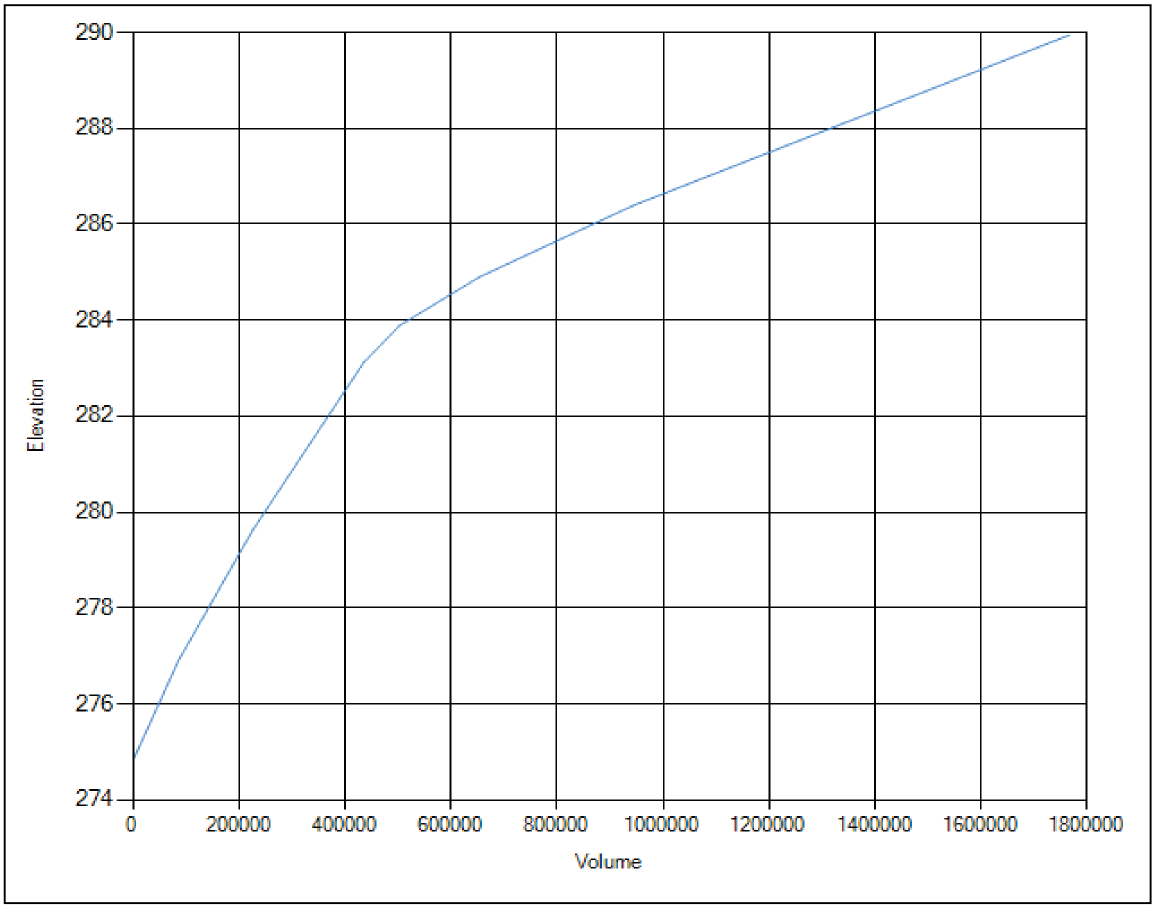


Figure 3-23. Elevation – Volume relationship for a 2D cell.

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 3-24 below:

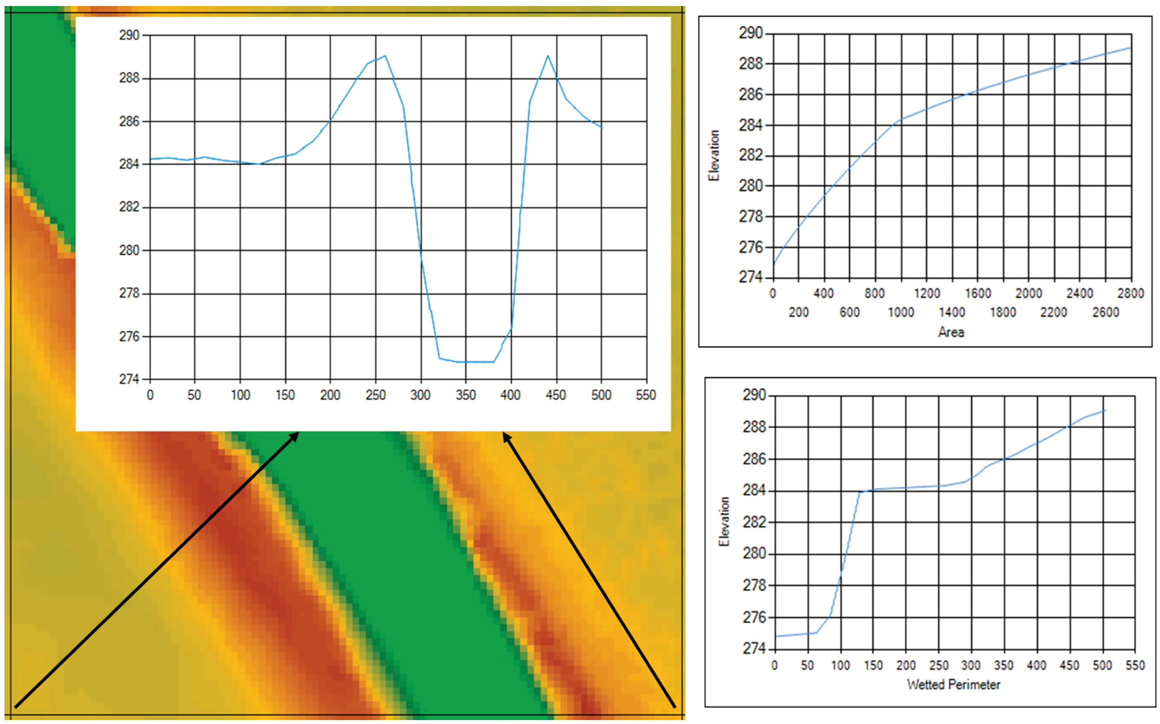


Figure 3-24. Example of how Cell Faces are processed into detailed hydraulic tables.

As shown in Figure 3-24, 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 3-25.

Figure 3-25. Example of detailed channel moving through larger cells in HEC-RAS

Shown in Figure 3-25 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 smaller than the channel width 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 3-25 is a viable option.

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 elevation 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 3-26). 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.

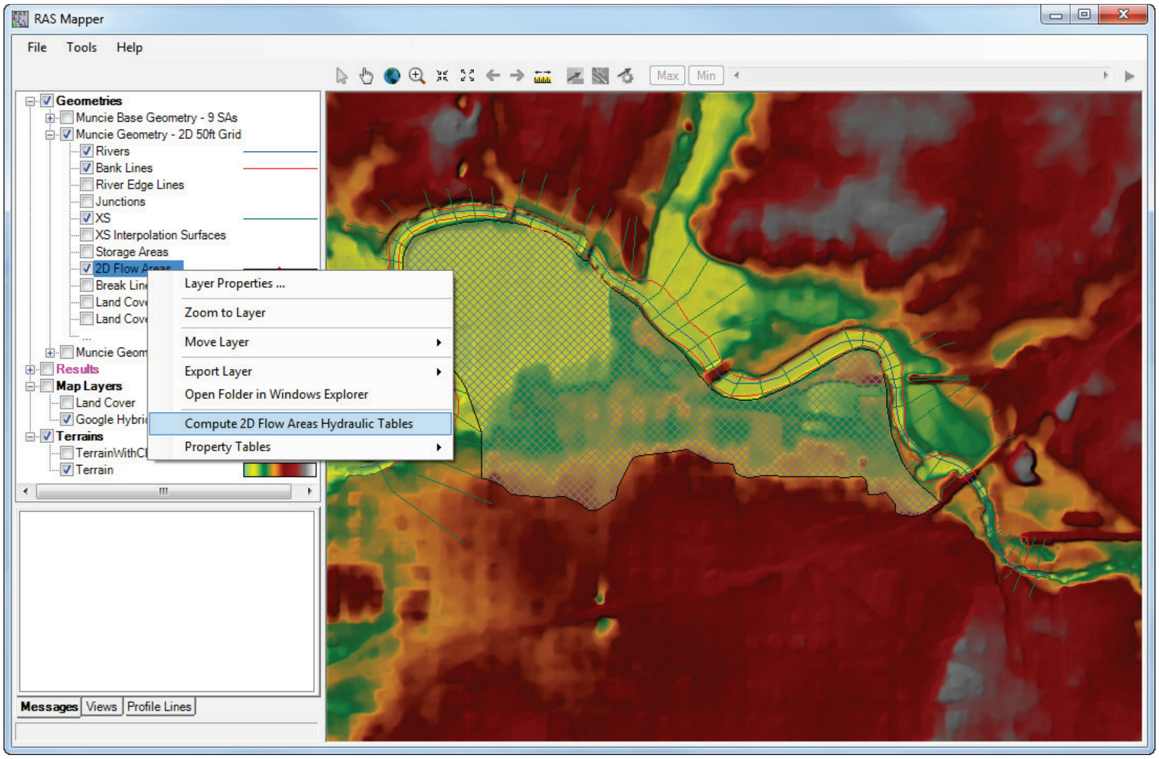


Figure 3-26. Computing 2D flow area Hydraulic Tables from RAS Mapper.

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 is accomplished in the HEC-RAS Geometric Data editor.

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 3-27).

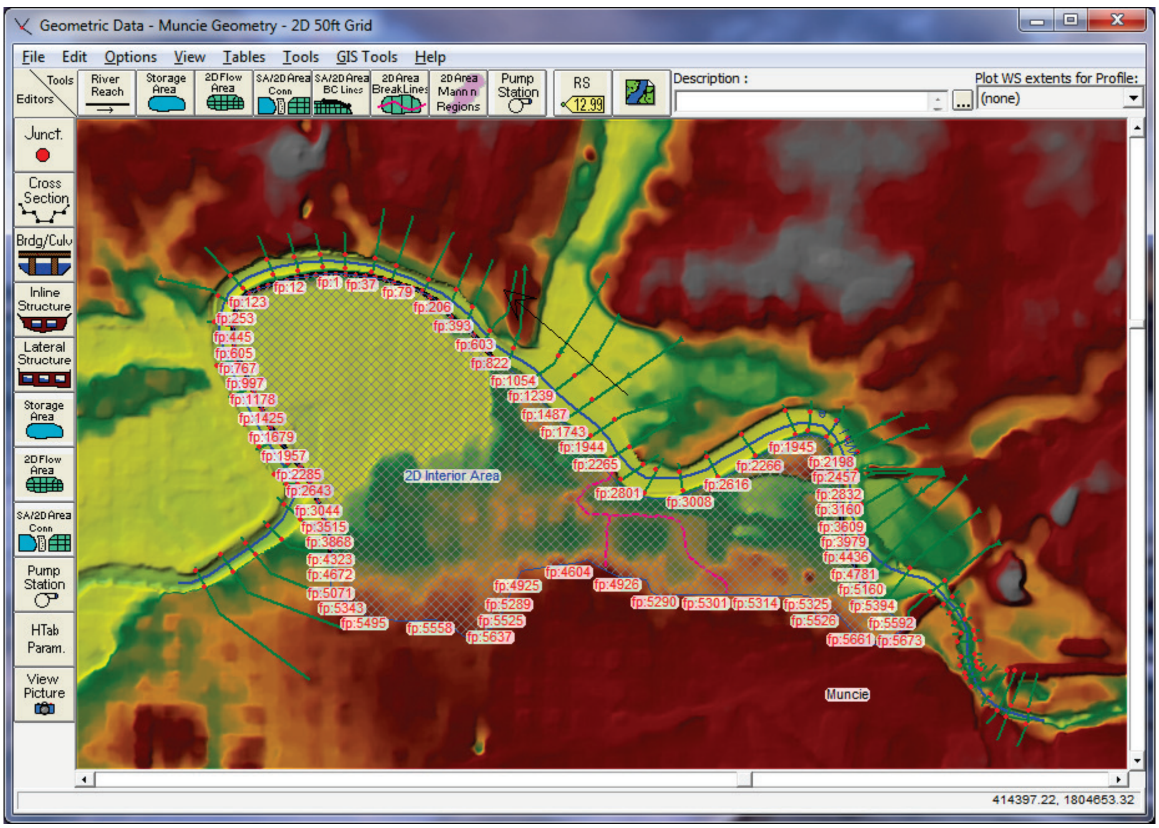


Figure 3-27. HEC-RAS Geometric Editor with the 2D flow area boundary Face Points displayed.

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 3-28.

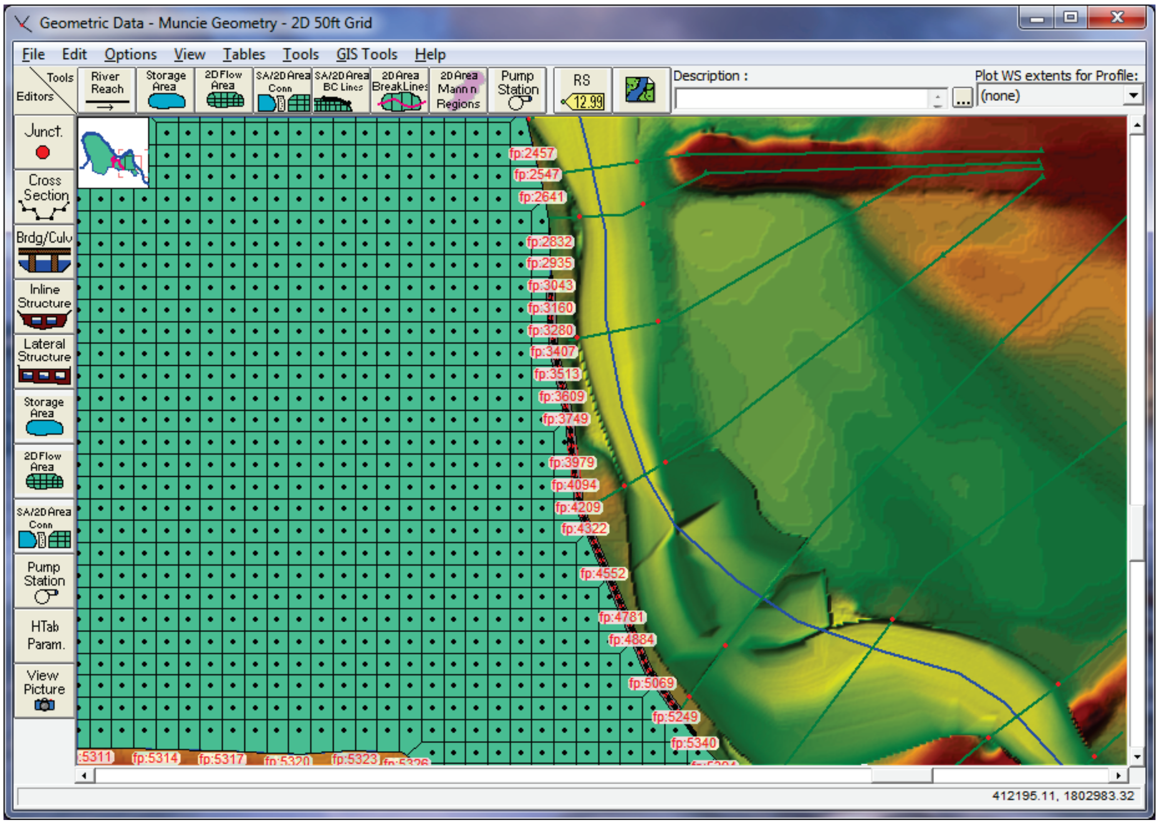


Figure 3-28. Example of a Lateral Structure (levee) hooked up to a 2D flow area.

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 now 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.

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 3-29 is the **Lateral Structure Centerlines Table**.

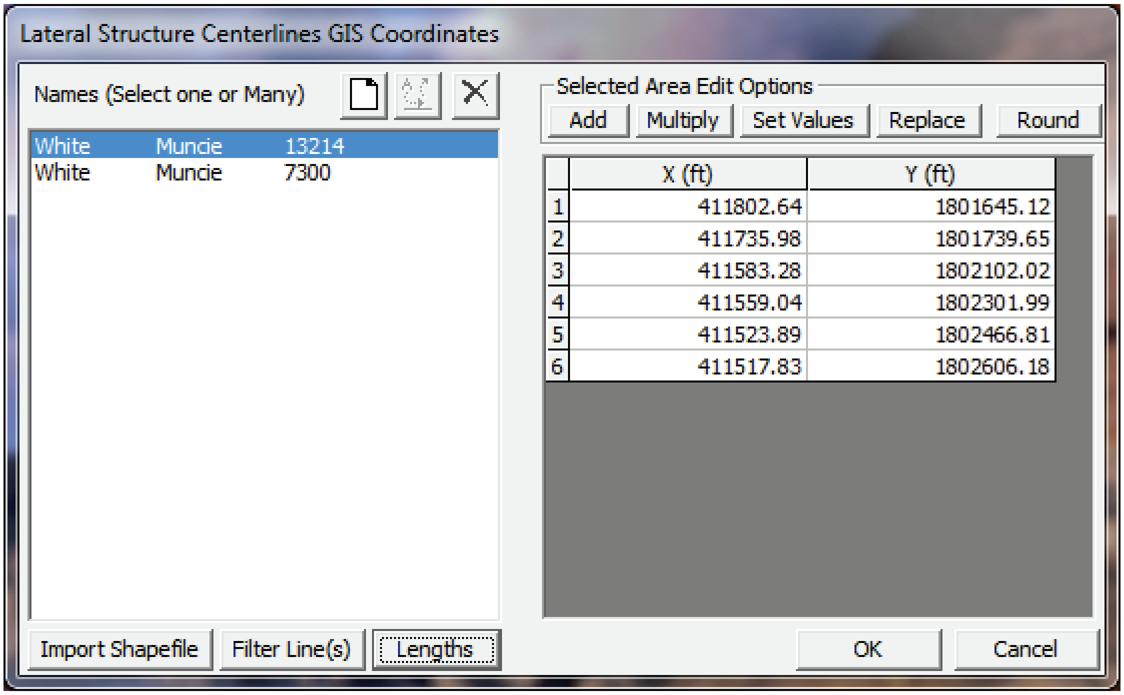


Figure 3-29. Example Lateral Structure Centerlines Table with Geospatial coordinate data for a levee.

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 3-30.

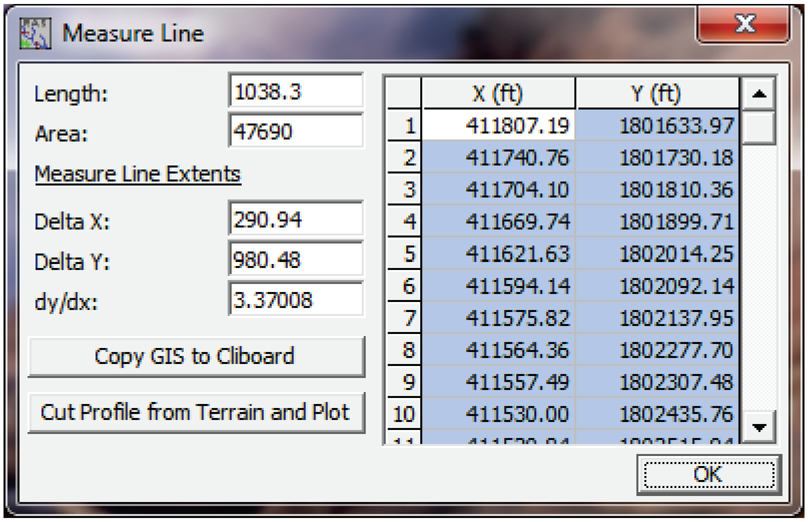


Figure 3-30. Example Measuring Tool line data window.

As shown in Figure 3-30, 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 3-31).

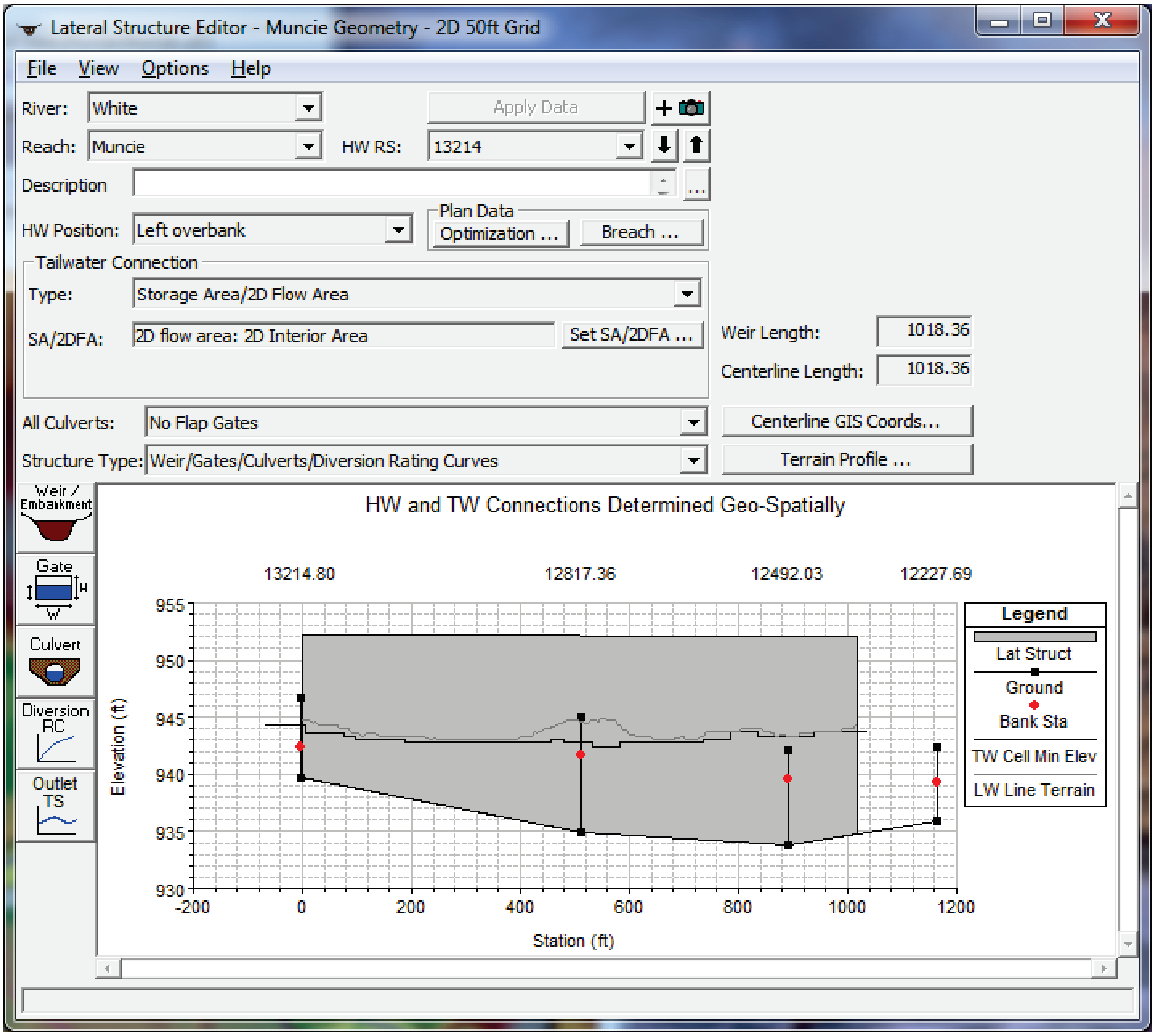


Figure 3-31. Lateral Structure Editor with tailwater connection to a 2D flow area.

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 3-32.

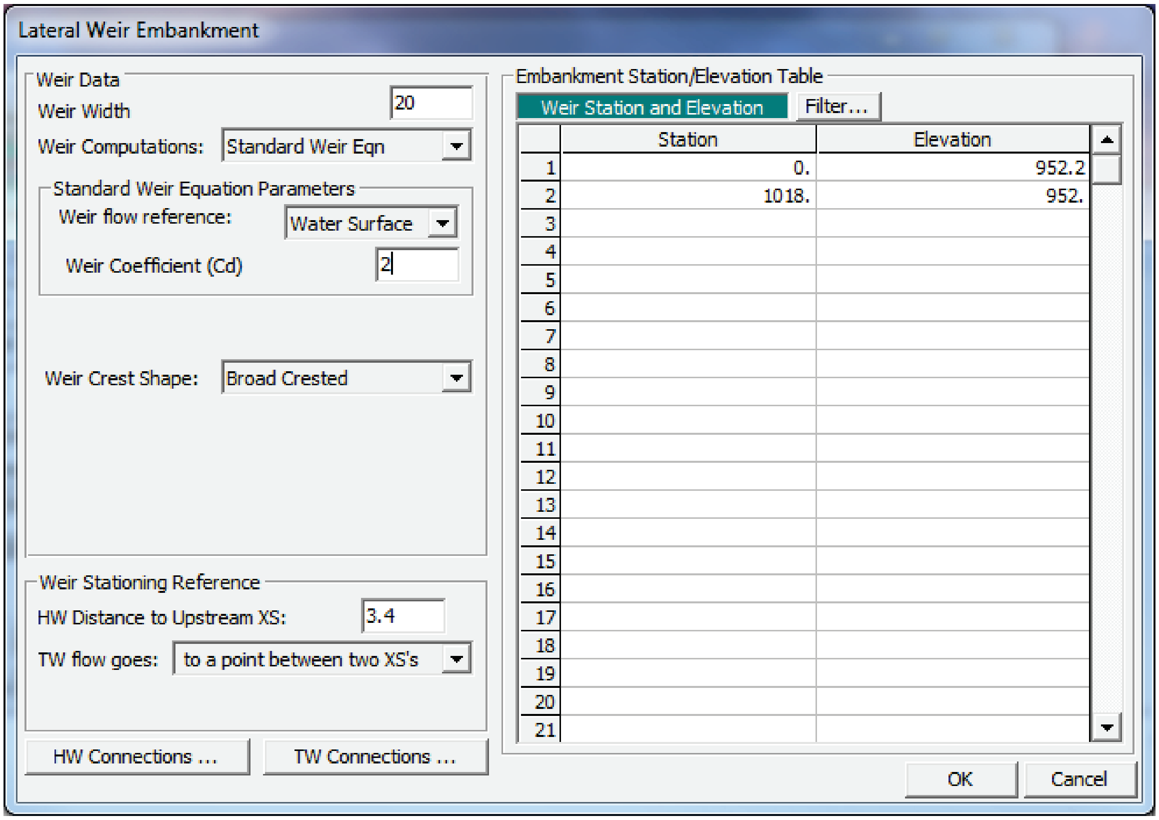


Figure 3-32. Lateral Structure Editor with structure Station/Elevation data, and 2D Face Point stationing.

As shown in Figure 3-32, 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 3-33. 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.

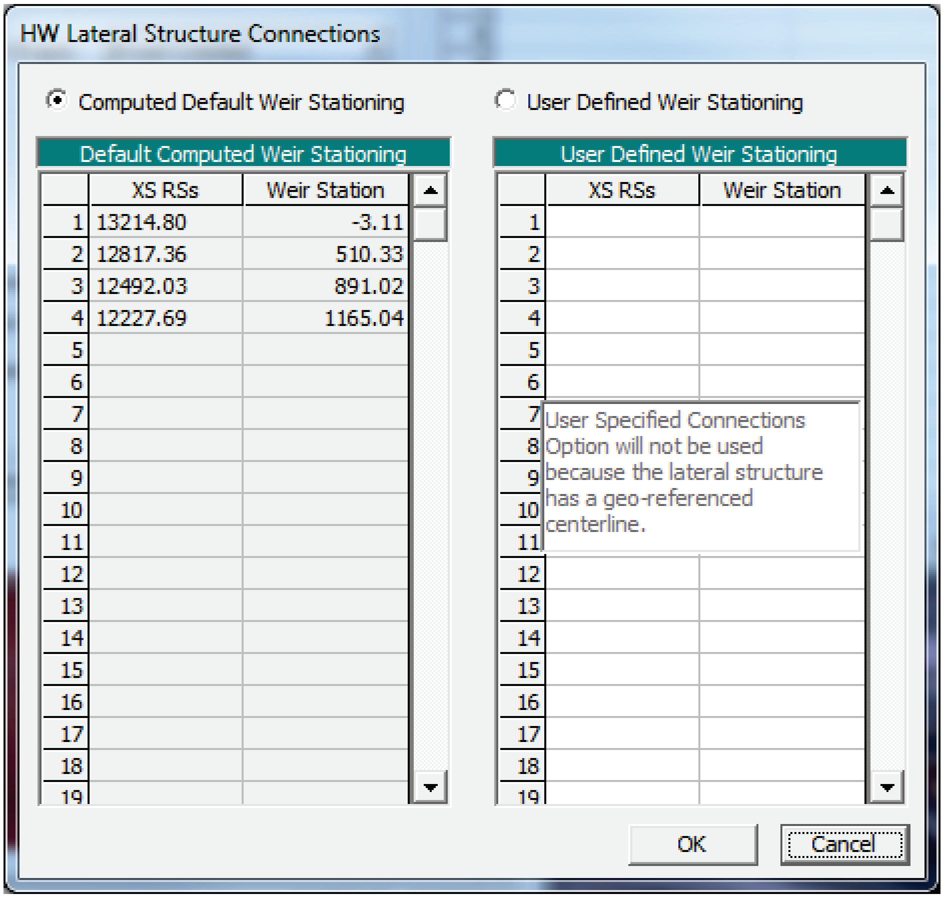


Figure 3-33. Head Water Connection Table for a Lateral Structure.

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 3-34. 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.

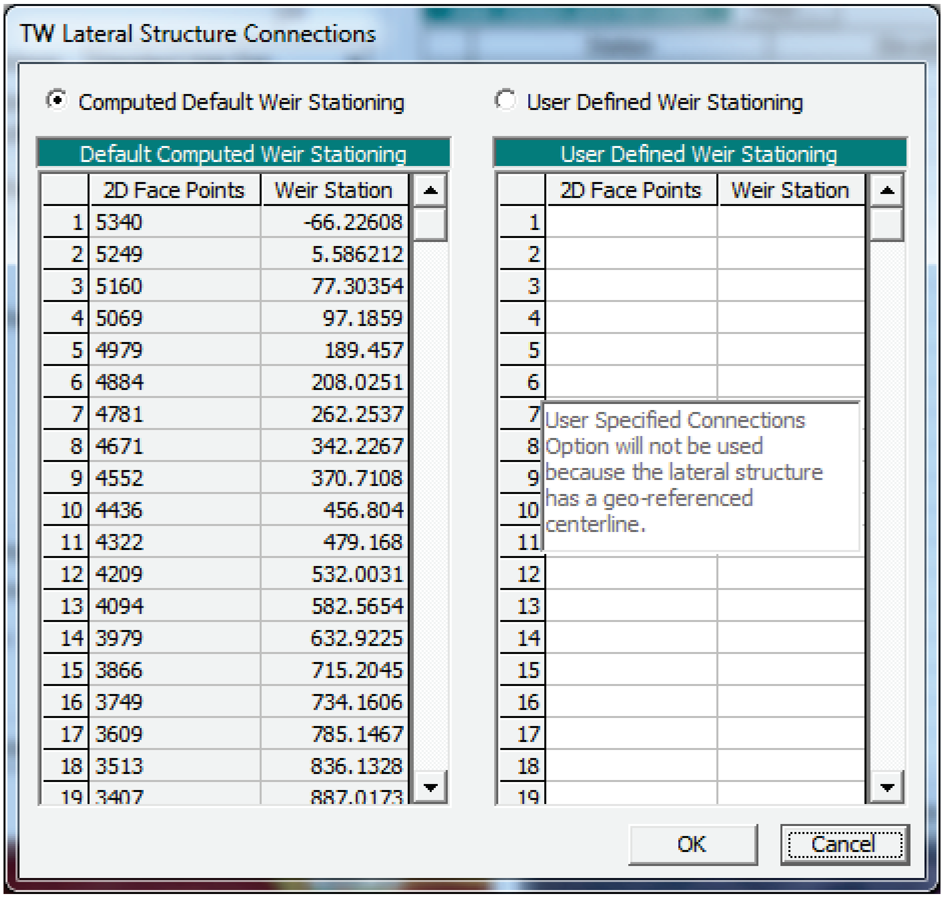


Figure 3-34. Tailwater Connection Table for a Lateral Structure.

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.

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.

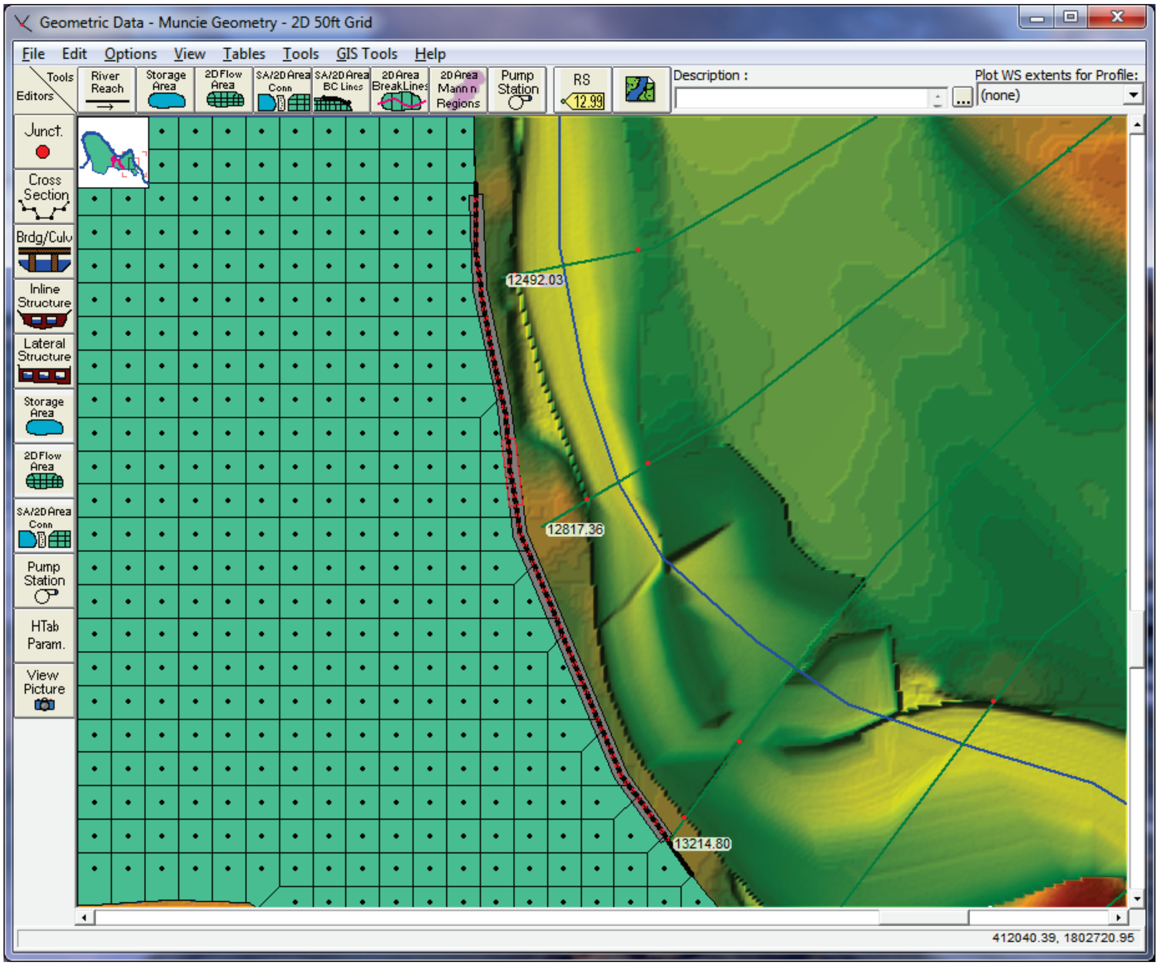


Figure 3-35. HEC-RAS Geometric Editor showing a thick black line for the connection of a Lateral Structure to the 2D flow area Face Points.

For this example a breach location/data has also been added for the analysis of this upper levee. Shown in Figure 3-35 is the Levee (Lateral Structure) Breach Data editor with the data for this levee.

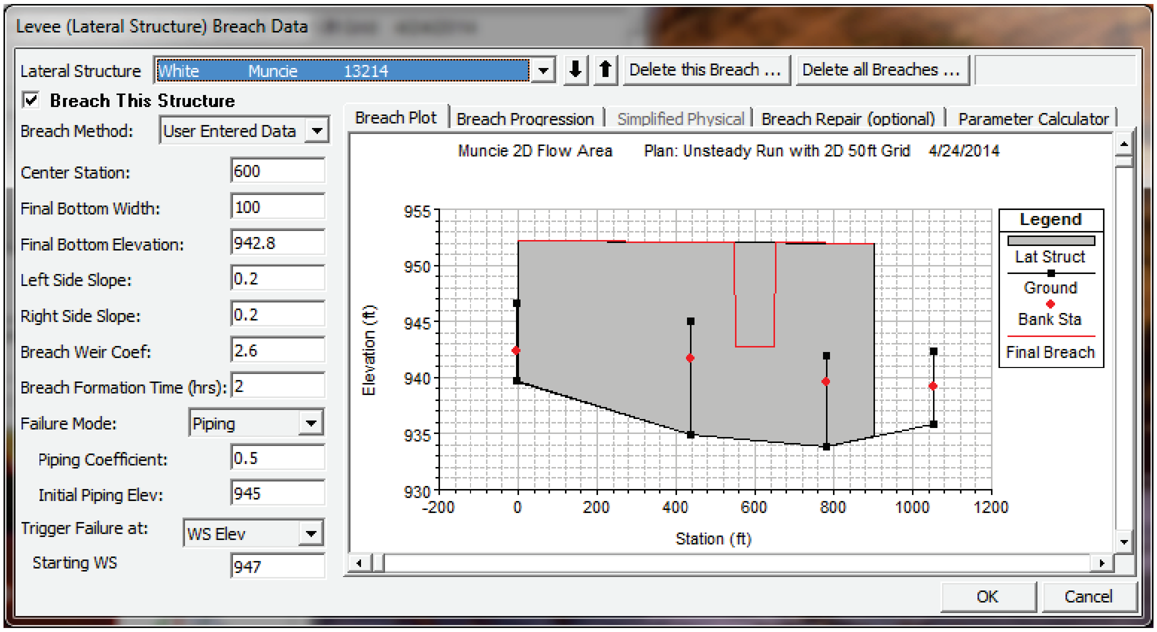


Figure 3-36. Levee (Lateral Structure) Breach Data editor, with breach data for the upstream levee.

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 3-37 for the extents of this downstream Levee. The Levee (Lateral Structure) is highlighted in red.

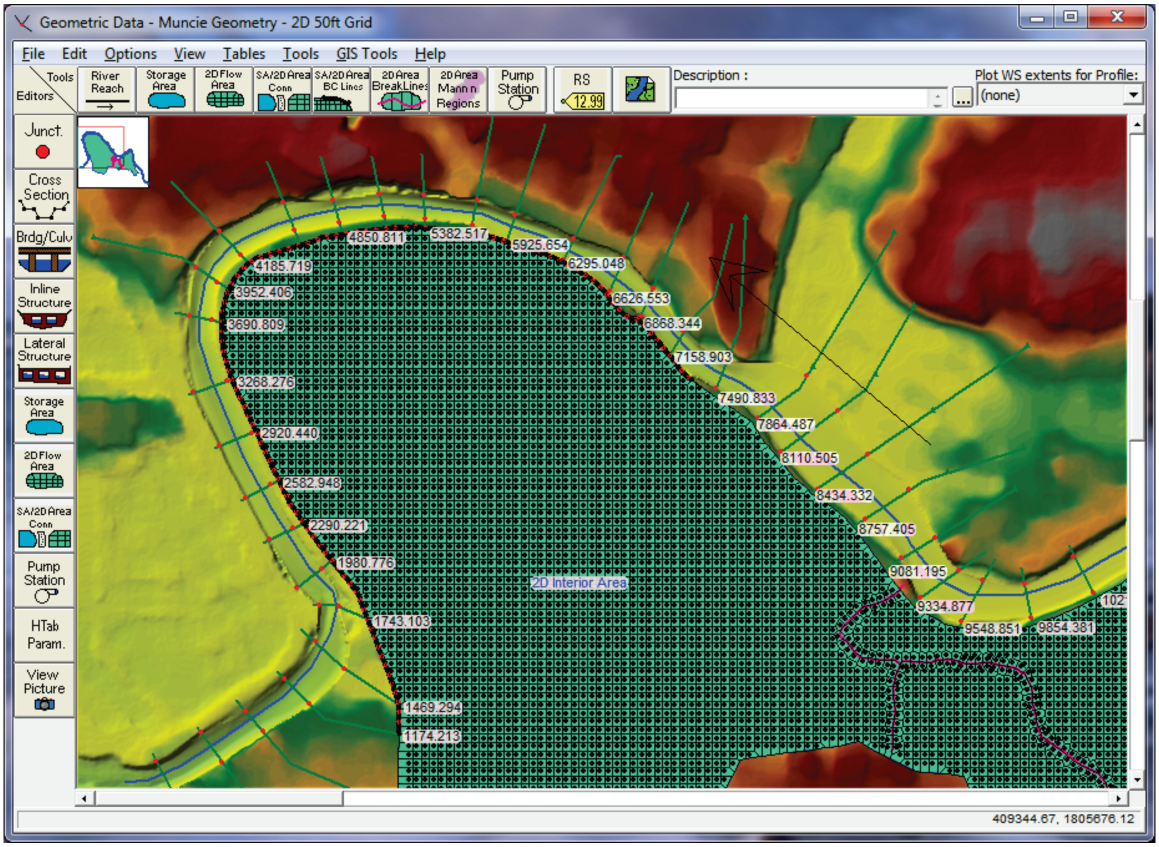


Figure 3-37. HEC-RAS Geometric Editor with downstream Levee (Lateral Structure) highlighted in red.

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 3-38.

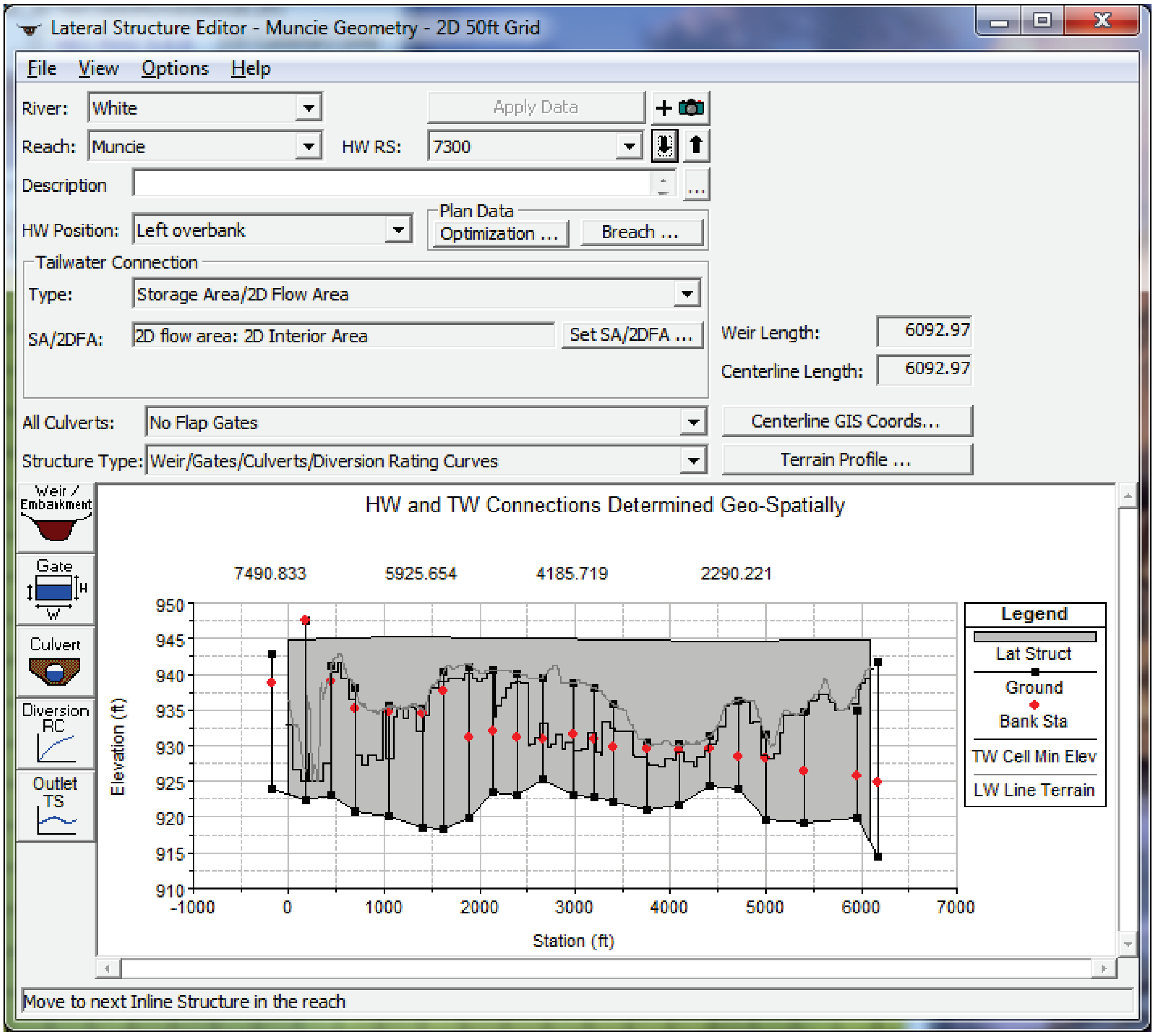


Figure 3-38. Downstream Levee (Lateral Structure) with a tailwater connection to the 2D flow area.

Shown in Figure 3-39, 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).

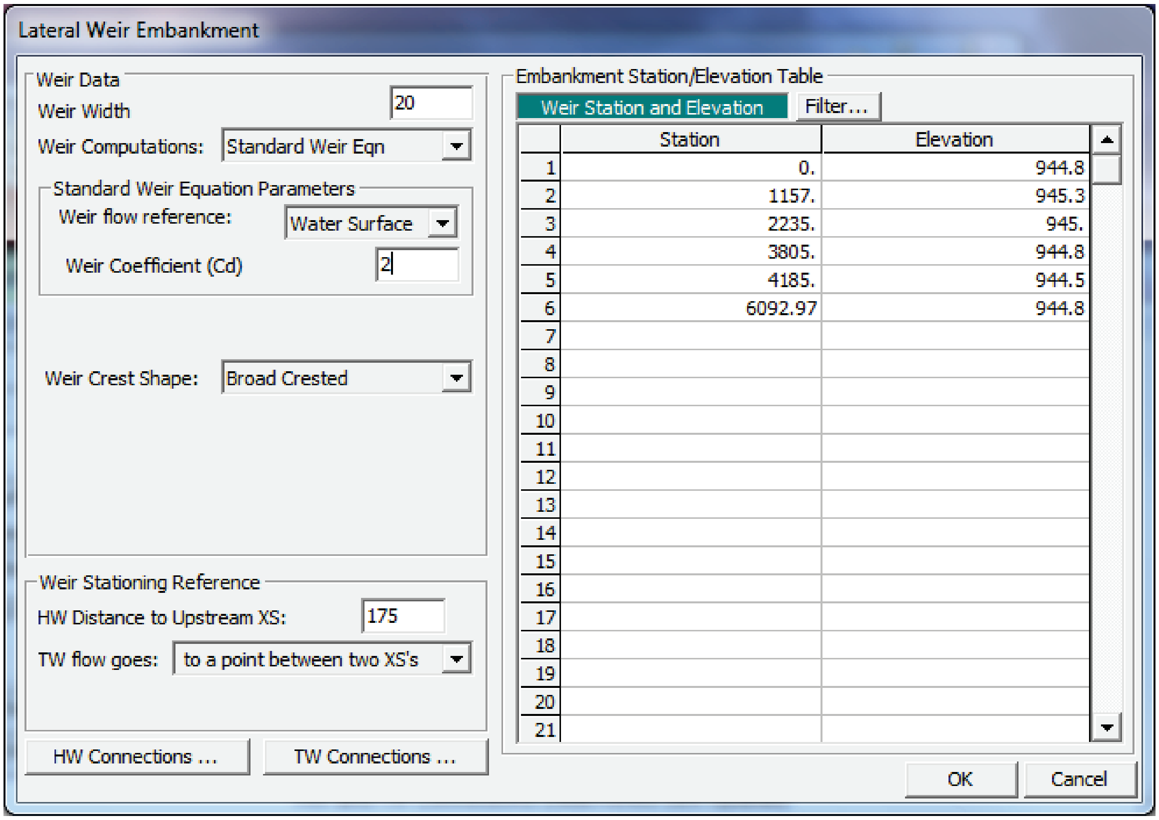


Figure 3-39. Lateral Weir Embankment editor with data for the downstream levee and linked to the 2D flow area Face Points.

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.	**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 sloping 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 3-40.

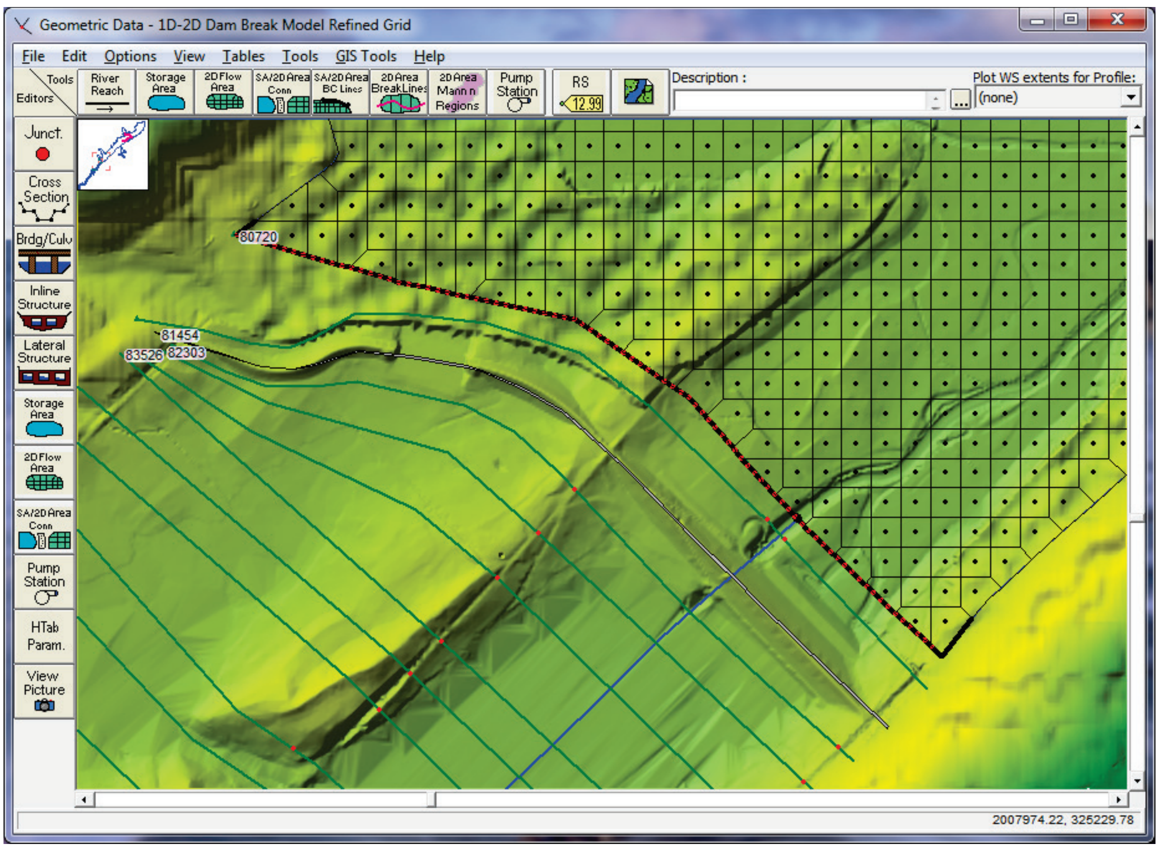


Figure 3-40. Example of an upstream 1D River Reach connected to a downstream 2D flow area.

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 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 converges within a user specified tolerance. 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 1D unsteady flow solver (at this time) 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 3-41.

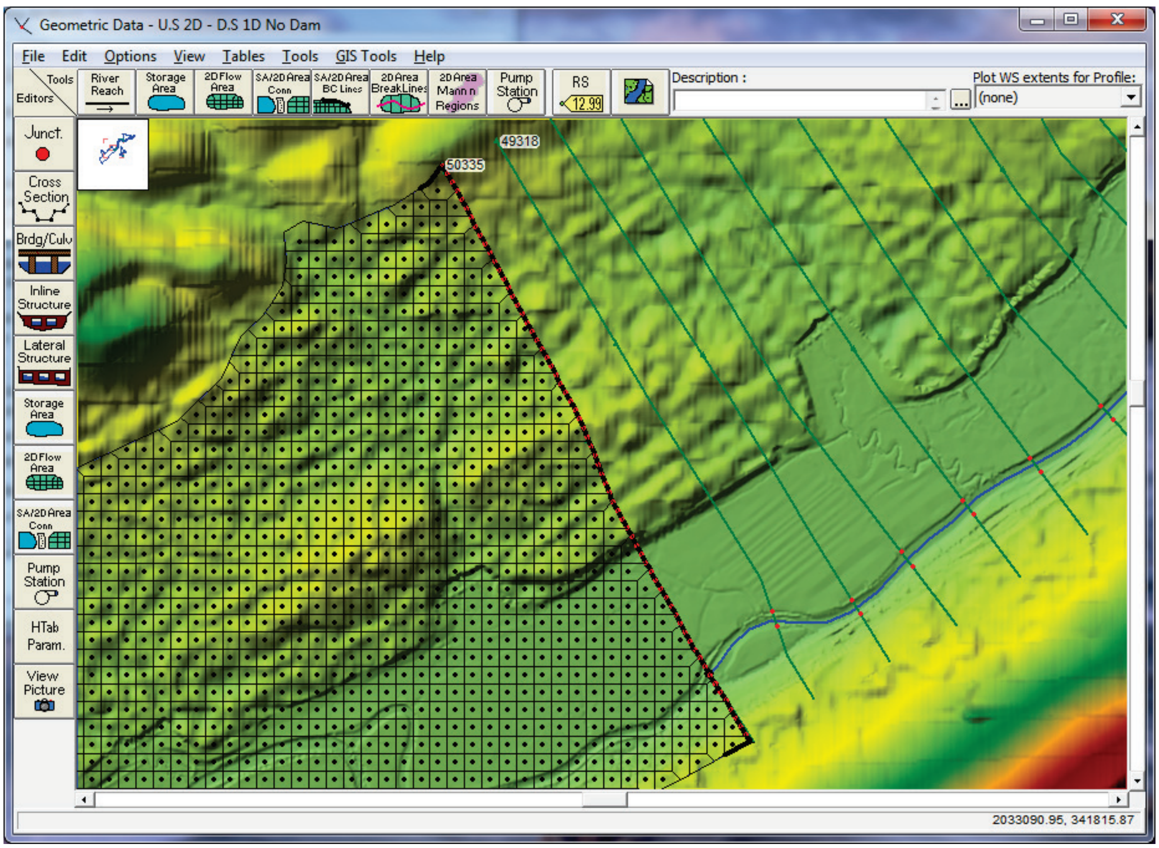


Figure 3-41. Example of an upstream 2D flow area connected to a downstream 1D River Reach.

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 3-42 below.

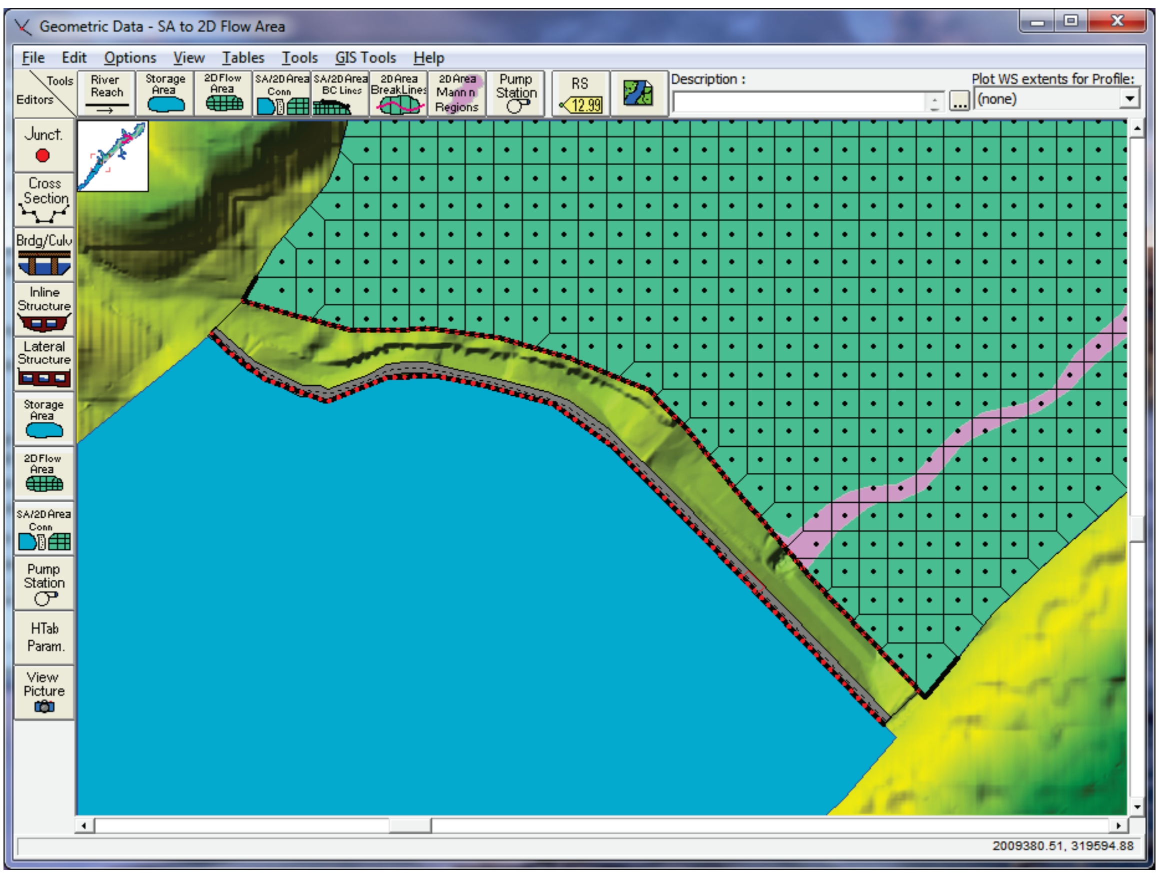


Figure 3-42. Example of a Storage Area connected to a 2D flow area.

In the example shown in Figure 3-42, 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 3-42, 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. **Note: For any culverts and/or gates, the minimum elevation of the culvert/gate must not be below the minimum elevation of the cell it is connected to. This is another reason to use cells that are large enough to span at least to the bottom of the embankment.**
- 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 3-42.
- 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 3-43.

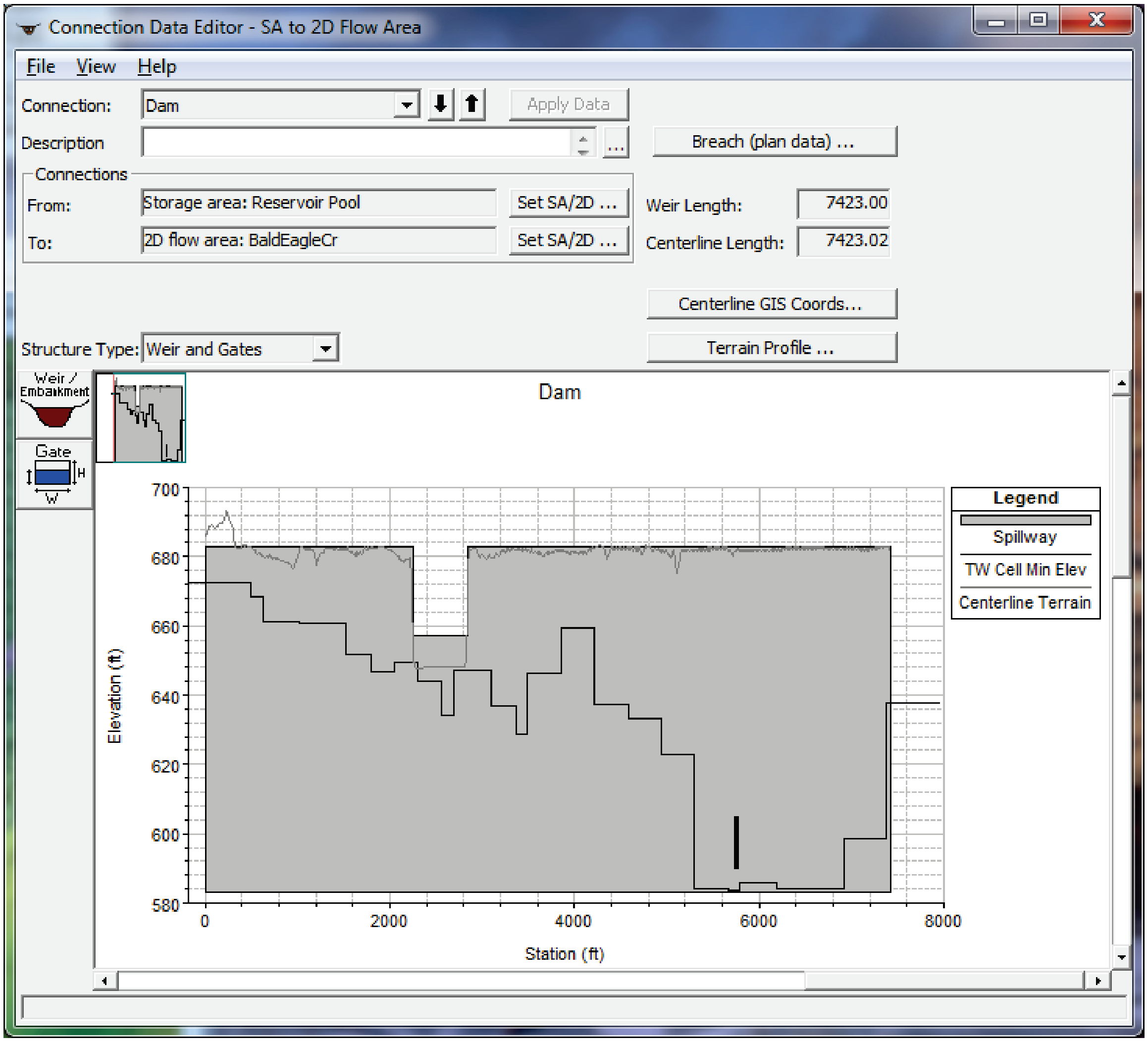


Figure 3-43. SA/2D Area Hydraulic Connection editor.

- 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 3-43, there is an embankment with an emergency spillway defined, and there are also low flow gates defined.

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 3-44.

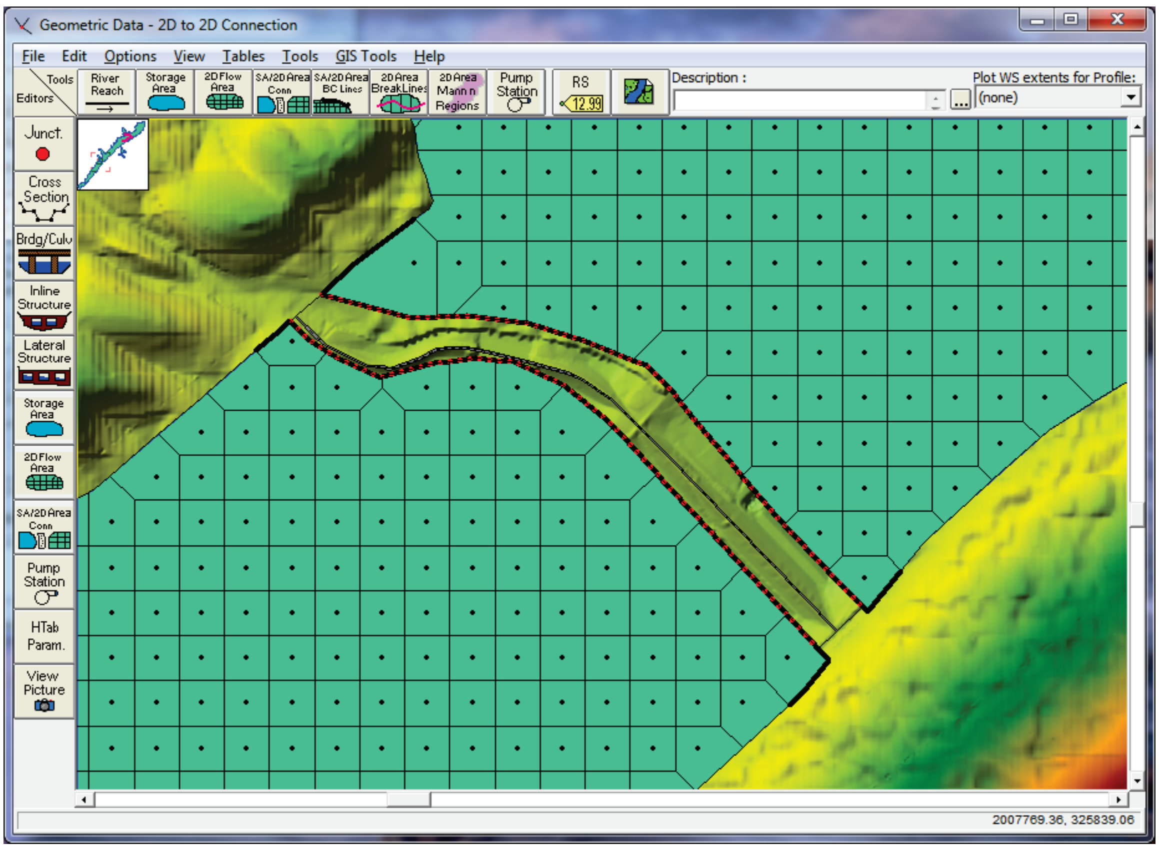


Figure 3-44. Example of connecting one 2D flow area to another 2D flow area with a Hydraulic Structure.

In the example shown in Figure 3-44, 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 3-44.
- 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 3-45.

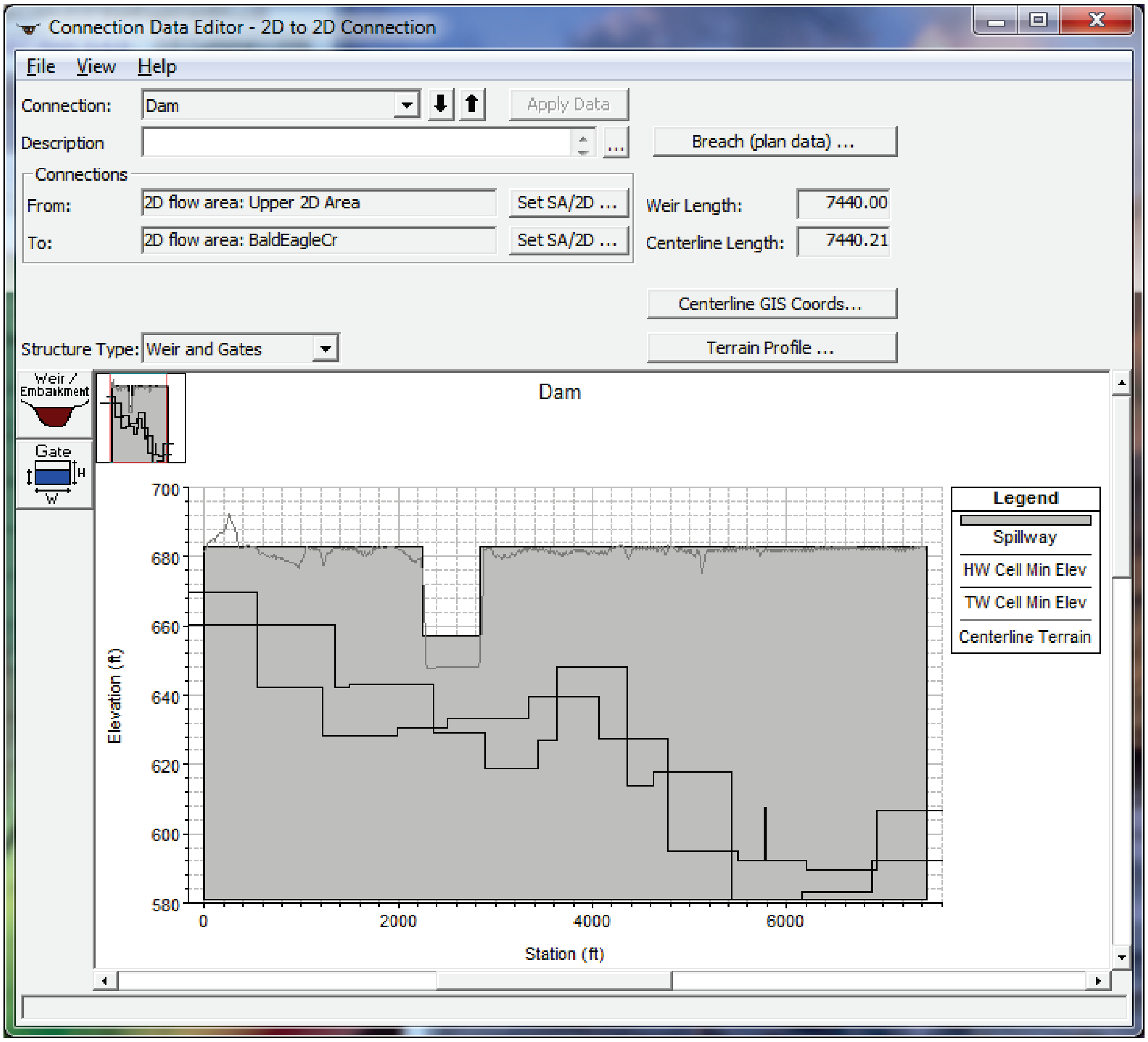


Figure 3-45. SA/2D Area Hydraulic Connection editor.

- 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 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 3-46.

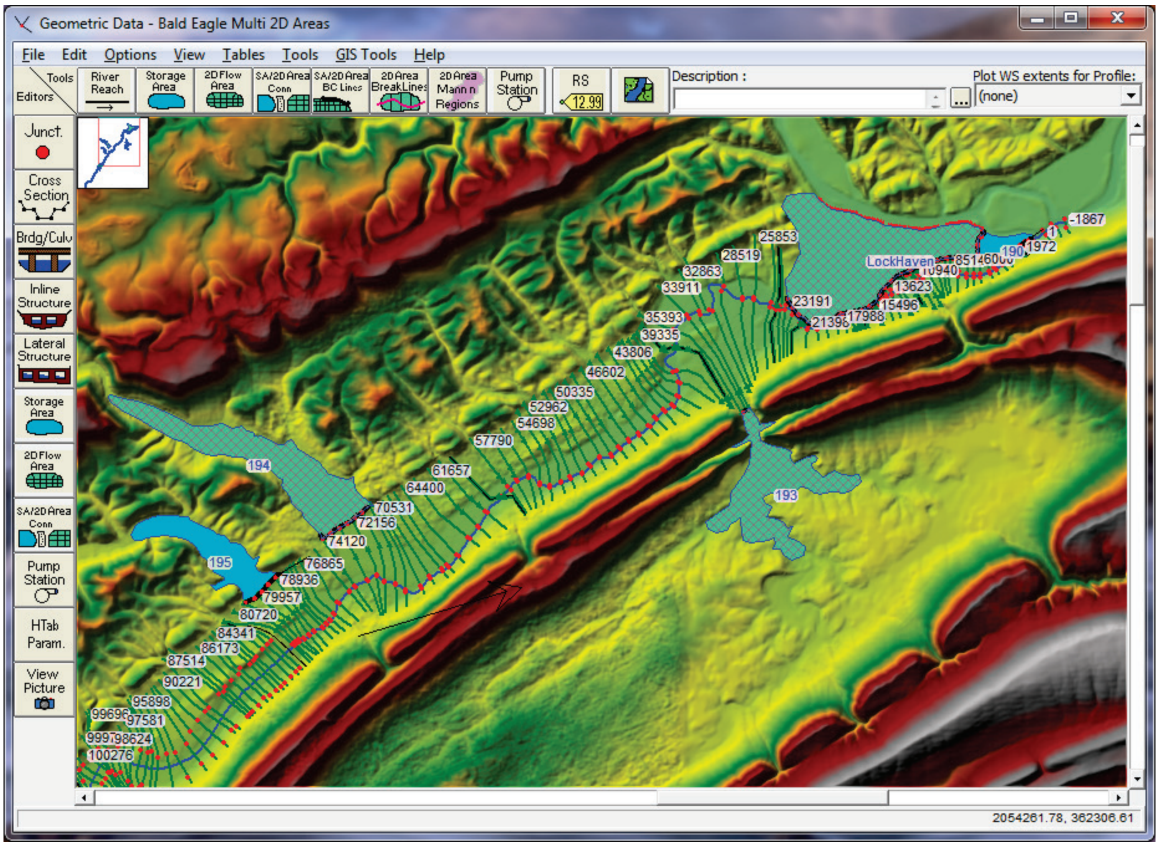


Figure 3-46. Multiple 2D flow areas in a single geometry file.

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 must be laid out along the faces of the 2D cells (The 2D cell faces control flow movement). See Figure 3-47.

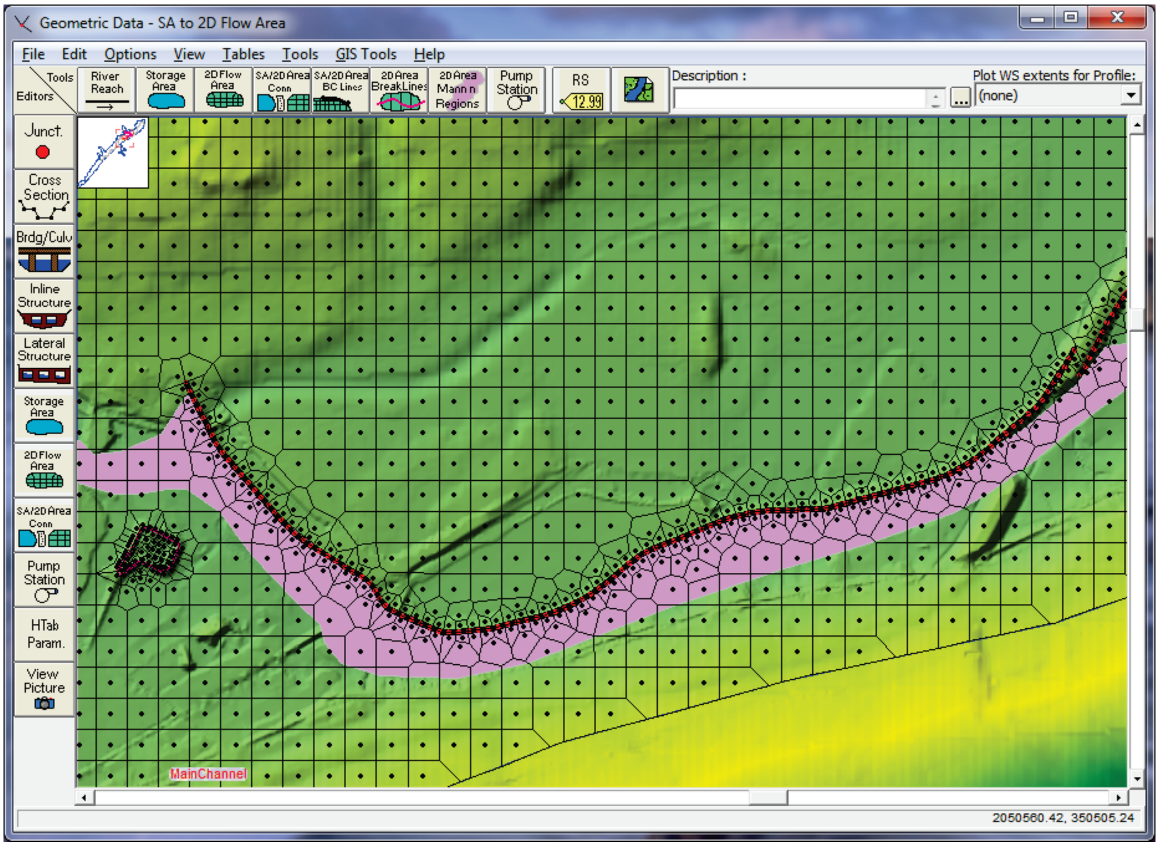


Figure 3-47. Example hydraulic structure inside of a 2D flow area.

The Internal Hydraulic structures can be used to model the following:

1. Structure Type **Weir**: Flow going over top of levees, roads, weirs, spillways, etc… The user has the option to model the overflow with the 2D Unsteady Flow equations or a Weir equation. The user also has the option to breach the embankment during the run.

2. Structure Type **Weir and Gates**: Can model everything discussed in no. 1 above. Plus you can add gates through the embankment. Gates can be controlled in many ways from the Unsteady Flow Data editor. If you put a gate into the hydraulic structure, both cells that it is connected too (head water and tail water sides), must have terrain data that is lower than the gate invert elevations. Additionally the volume of water available in the connected cells needs to be consistent with what can be computed

through the gate. This is especially true at low flow. A common problem is when a gate is placed right at the very bottom of a cell, then for very shallow water surface elevations, the flow rate computed through the gate takes the entire volume of water out of the cell in one time step, thus inducing a computational instability.

3. Structure Type **Weir and Culverts**: Can model everything discussed in no. 1 above, and you can also put culverts through the embankment. Culverts can be set for two way flow (the default), or the user can turn on flap gates and pick a flow direction (this limits flow to only going one way through the culvert. If you put a culvert into the hydraulic structure, both cells that it is connected too (head water and tail water sides), must have terrain data that is lower than the culvert invert elevations. Additionally the volume of water available in the connected cells needs to be consistent with what can be computed through the culvert. This is especially problematic at low flow. A common problem is when a culvert is placed right at the very bottom of a cell, then for very shallow water surface elevations, the flow rate computed through the culvert takes the entire volume of water out of the cell in one time step, thus inducing a computational instability.

Note: Currently the Internal Hydraulic structures are limited to connecting cells along faces. Hydraulic outlets such as gates and culverts can only be connected from the cells on one side of the structure to the cells on the other side of the structure. In future versions we will allow culverts, gates, etc… to be connected to cells upstream and downstream of the structure that are some distance away from the structure (i.e. you will be able to specify the X and Y coordinates of the culvert or gate entrance and exit).

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.
- 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 3-47, 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 (break line) 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 3-48.

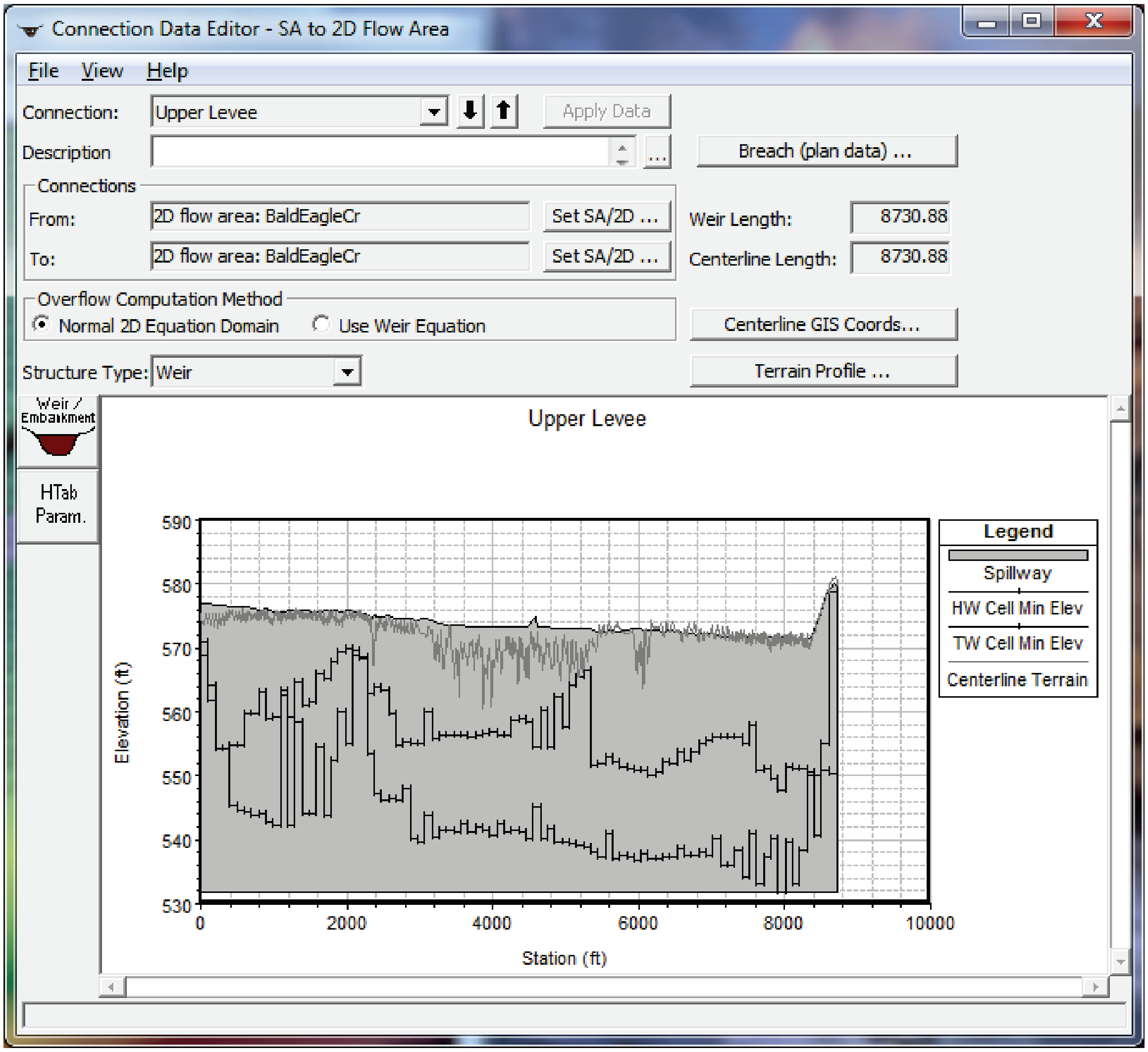


Figure 3-48. Example of using SA/2D Area Conn to put a hydraulic structure inside a 2D flow area.

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 and gated openings can be added into the hydraulic structure (depending on what "Structure Type" was selected). 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.

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.

External 2D flow area Boundary Conditions

Overview

In addition to connecting a 2D flow area to 1D River Reaches and Storage Areas, there are five types of external boundary conditions that can be linked directly to the 2D flow areas. These boundary condition types are:

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

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. The **Precipitation** boundary condition can be applied directly to any 2D flow area as a time series of rainfall excesses (right now we do not have interception/infiltration capabilities, these will be in future versions).

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 3-49). 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 3-49, 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 "DSNormalDepth" and "DS2NormalD", 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.

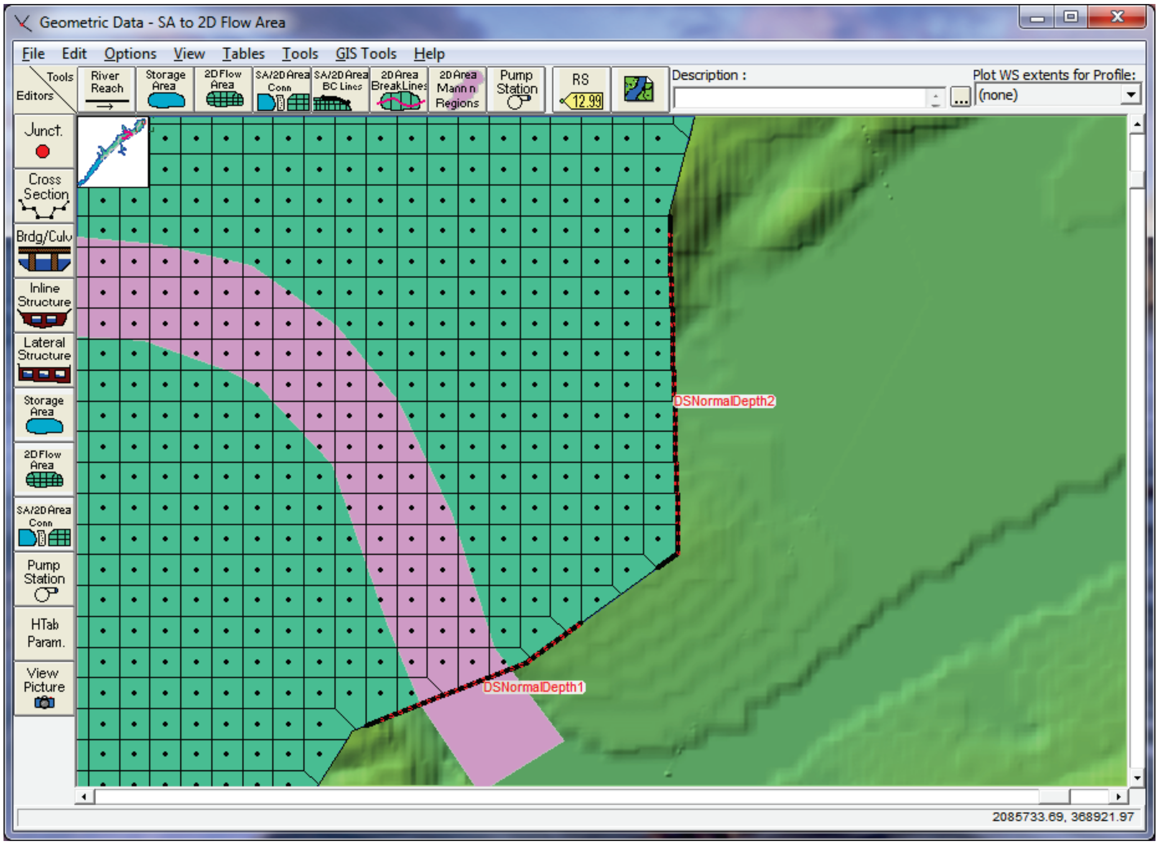


Figure 3-49. Example of adding an External 2D flow area boundary condition location.

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 3-50).

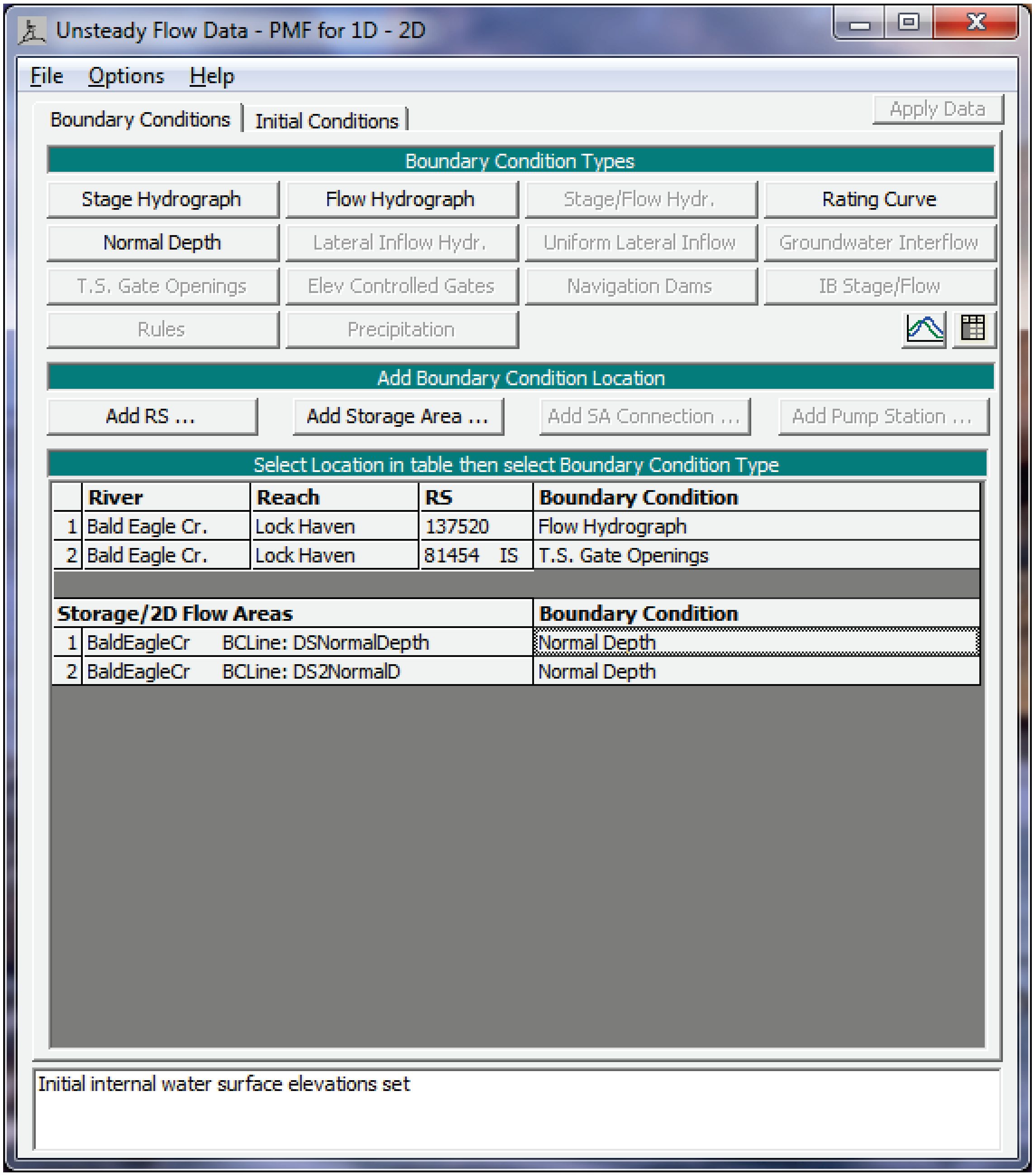


Figure 3-50. Example of adding external boundary conditions directly to a 2D flow area.

As shown in Figure 3-50, 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 open field 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 3-49, 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.

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** 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.

Precipitation

The **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.

2D Flow Area 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; 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 3-51). 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 3-51). More sophisticated starting water surfaces is another item planned for future versions.

Restart File Option for Initial Conditions

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.

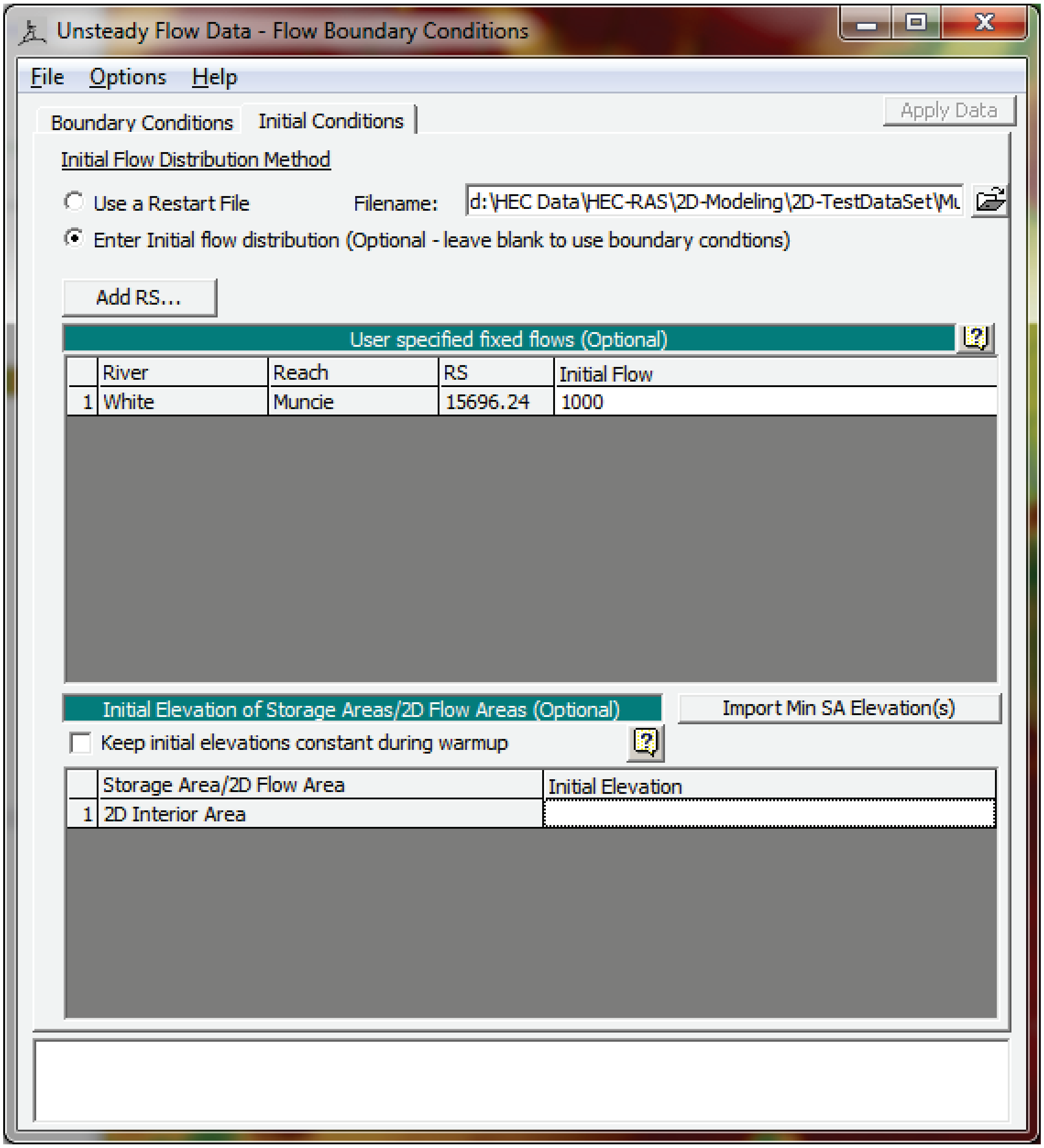


Figure 3-51. Unsteady Flow Data Editor with 2D flow area initial conditions.

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 1D 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 3-52 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 0.5 (50 % of the ramp up time).

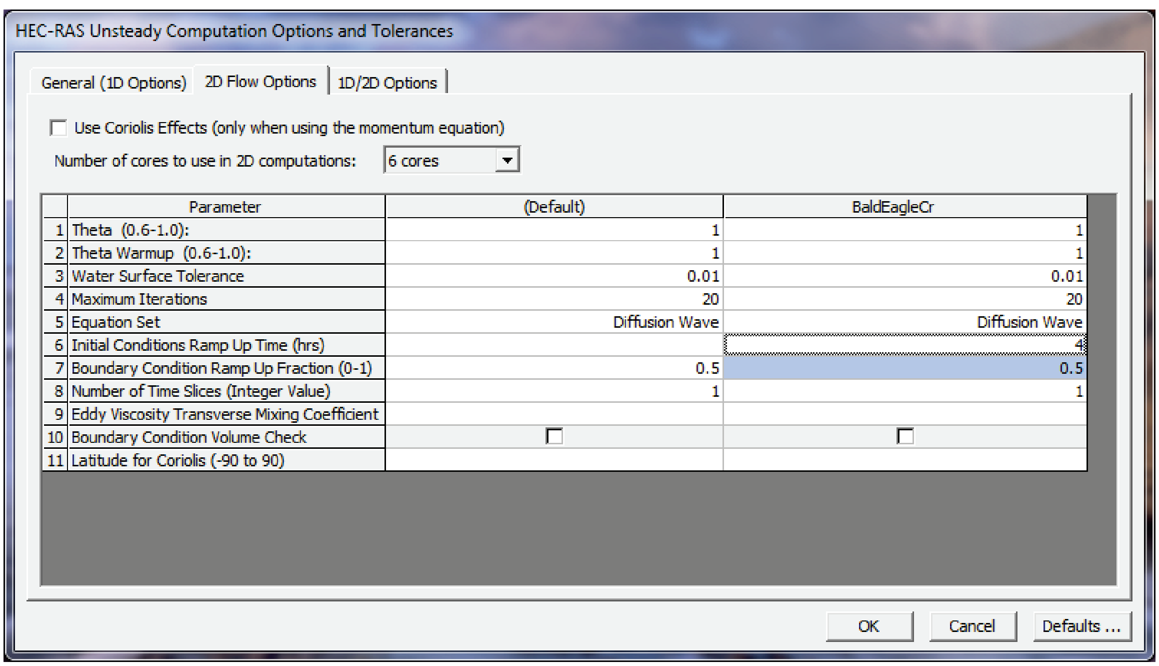


Figure 3-52. 2D flow area Computational Options.

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 4

Running the Combined 1D/2D Unsteady Flow Model

Running a 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. So 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), and they are solved together iteratively. 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.

Full Saint Venant or Diffusion Wave Equations

As mentioned previously, HEC-RAS has the ability to perform two-dimensional unsteady flow routing with either the Full Saint Venant equations (with added terms for turbulence modeling and Coriolis effects) or the Diffusion wave equations. 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 full Saint Venant 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 computational method to the **Full Momentum** equation option (Full Momentum 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 Full Momentum (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 Full Momentum equation set should always be used. The following is a list of examples of situations in which the user should generally use the Full Momentum 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. **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.

4. **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.

5. **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.

6. **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.

7. **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.

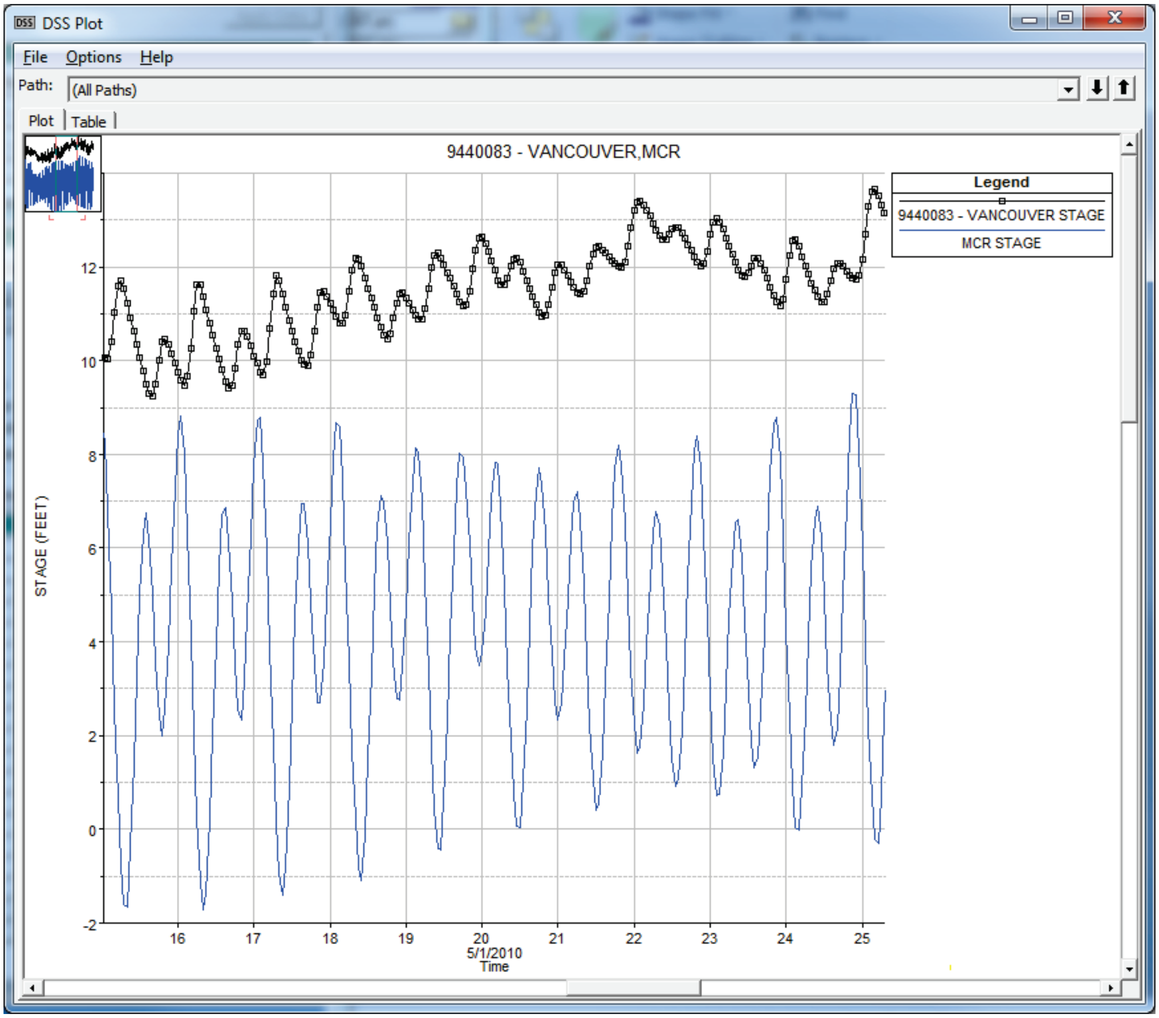


Figure 4-1. Stage Hydrograph at Vancouver Washington and Mouth of Columbia River.

Selecting an Appropriate Grid Size and Computational 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 surface 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 simple plane, 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. 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 the water surface slope. 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 will vary rapidly, smaller cell sizes must be used in that area to capture the changing water surface and its slope. 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 slope 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, 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 breaklines along the banks of the main channel. 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.

2. The cell size must be adequate to describe the water surface slope and changes in the water surface slope. If the water surface slope does not change rapidly, larger cell sizes can be used to accurately compute the water surface elevation and slope. If the water surface slope changes rapidly, then smaller cell sizes need to be used to have enough computation points to describe the changing water surface, 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 two equation sets that can be used to solve for the flow moving over the computational mesh, the Diffusion Wave equations and the Saint Venant equations (often referred to as the shallow water equations, called "Full Momentum" in the HEC-RAS interface). In general, the Diffusion Wave equations are more forgiving numerically than the Saint Venant equations. This means that larger time steps can be used with the Diffusion Wave equations (than can be with the Saint Venant equations), and still get numerically stable and accurate solutions. The following are guidelines for picking a computation interval for the Saint Venant equations and the Diffusion Wave equations:

<u>Saint Venant Equations (full momentum):</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)

<u>Diffusion Wave Equations:</u>

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

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. Select a ΔT, such that the Courant Number (C) is equal to the suggested value (i.e. 1.0 for Saint Venant Equations). However, you may be able to get away with a Courant number as high as 3.0 for the Saint Venant equations 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.

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 4-2). 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**.

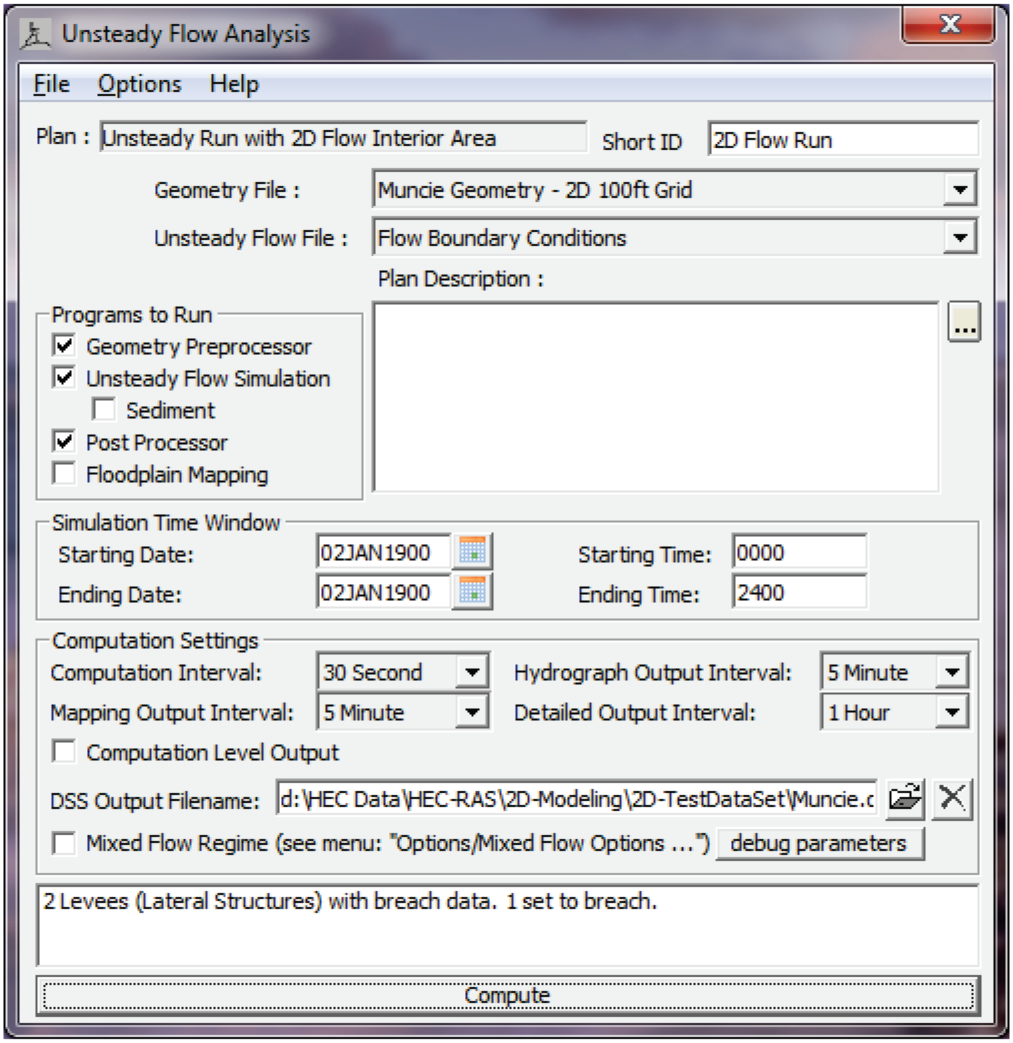


Figure 4-2. Unsteady flow Analysis Window with the new Floodplain Mapping feature.

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 the water surface and numerical error, 2D cells now also print out a Convergence flag. The following is the definition of each of these convergence flags:

Flag Value	**Definition**
Blank	CONVERGED
1	Went to max iterations but was converging
2	Went to max iterations but was diverging
3	Went to max ITER, diverging with positive and negative WSEL
4	went to max ITER,converging with positive and negative WSEL

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 4-3.

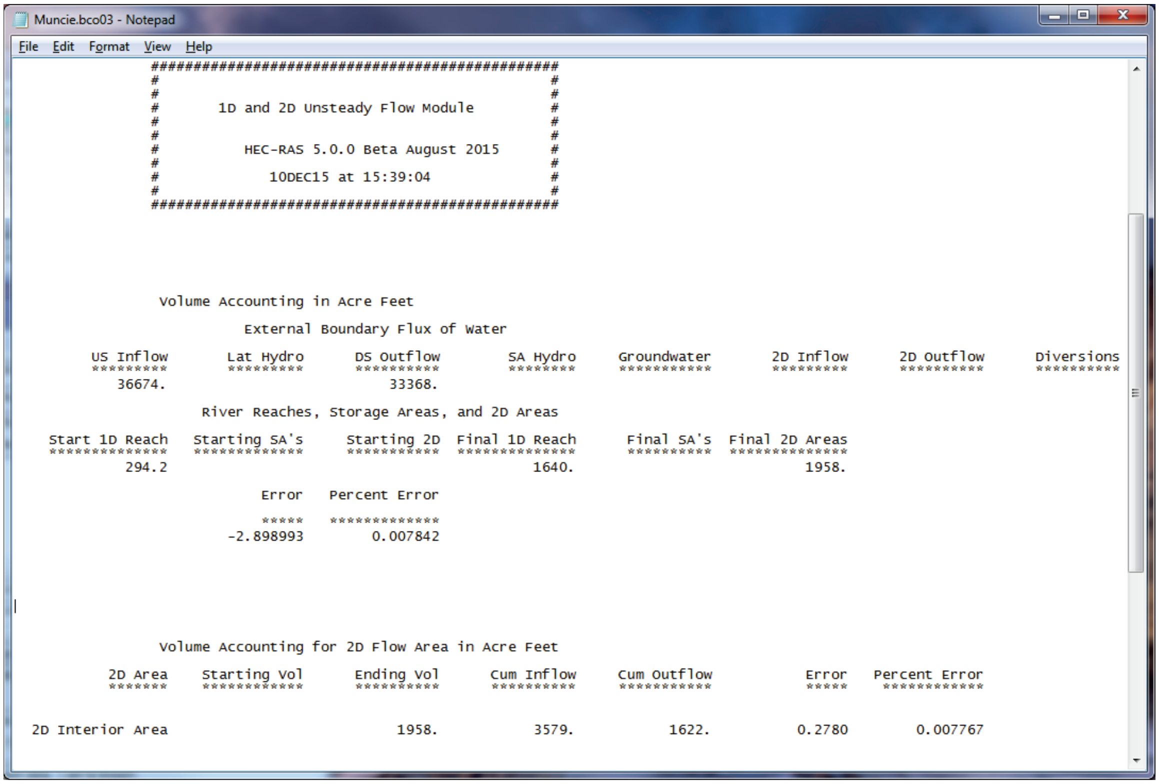


Figure 4-3. Example Computational Log File with Volume Accounting Output.

As shown in Figure 4-3, 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 4-3 shows that the overall model lost 4.13 acre-feet of water, which equated to a 0.011 % volume error (very low). The separate 2D Flow Area had a flow gain of 0.26 acre-feet, which equated to a 0.0075 % volume error (even lower).

2D Computation Options and Tolerances

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 4-4. This editor now has three tabs. The first Tab, labeled "**General (1D Options)",** 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.

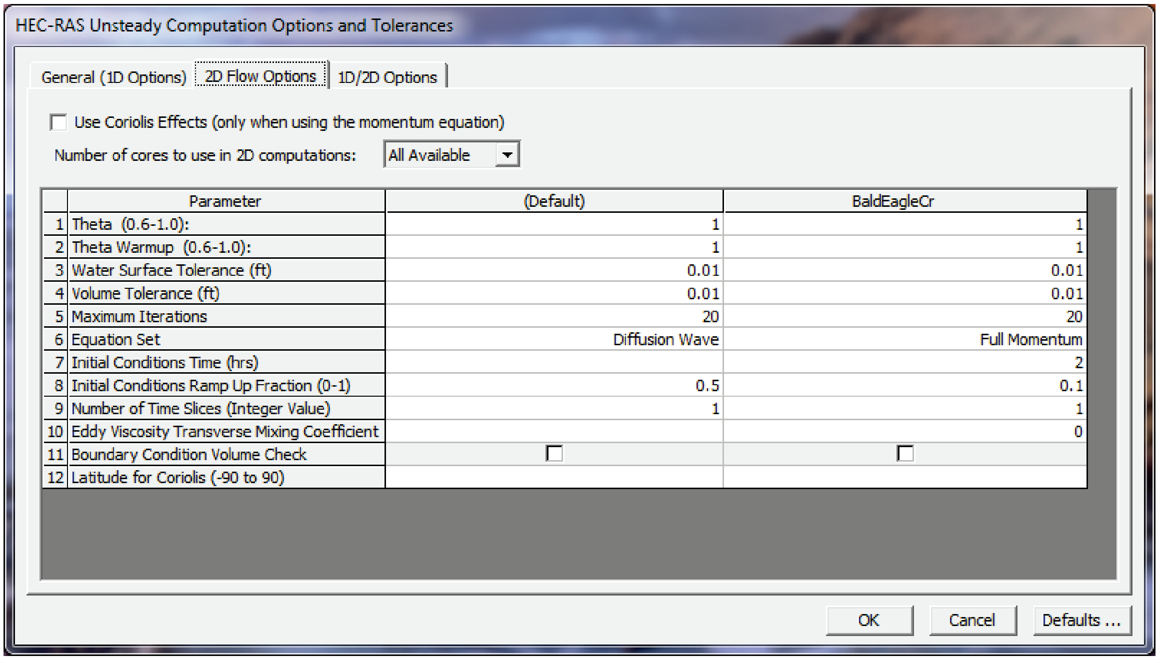


Figure 4-4. 2D Flow Area Calculation Options and Tolerances.

As shown in Figure 4-4, 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 Saint Venant Equations (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.

Number of Cores to use in computations: 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 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 4-1 and Figure 4-5 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 4-1.

Table 4-1. 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	16 days	30 s	2,251	6	48 s
EU Test No 5	1 day 6 hrs	10 s	7,460	8	54 s
Ohio/Miss	49 days	5 min	23,087	8	7 min 04 s
EU Test No 4	5 hrs	20 s	80,000	10	1 min 27 s
400 sq mi Watershed	2 days	2 min	250,000	16	54 min 42 s

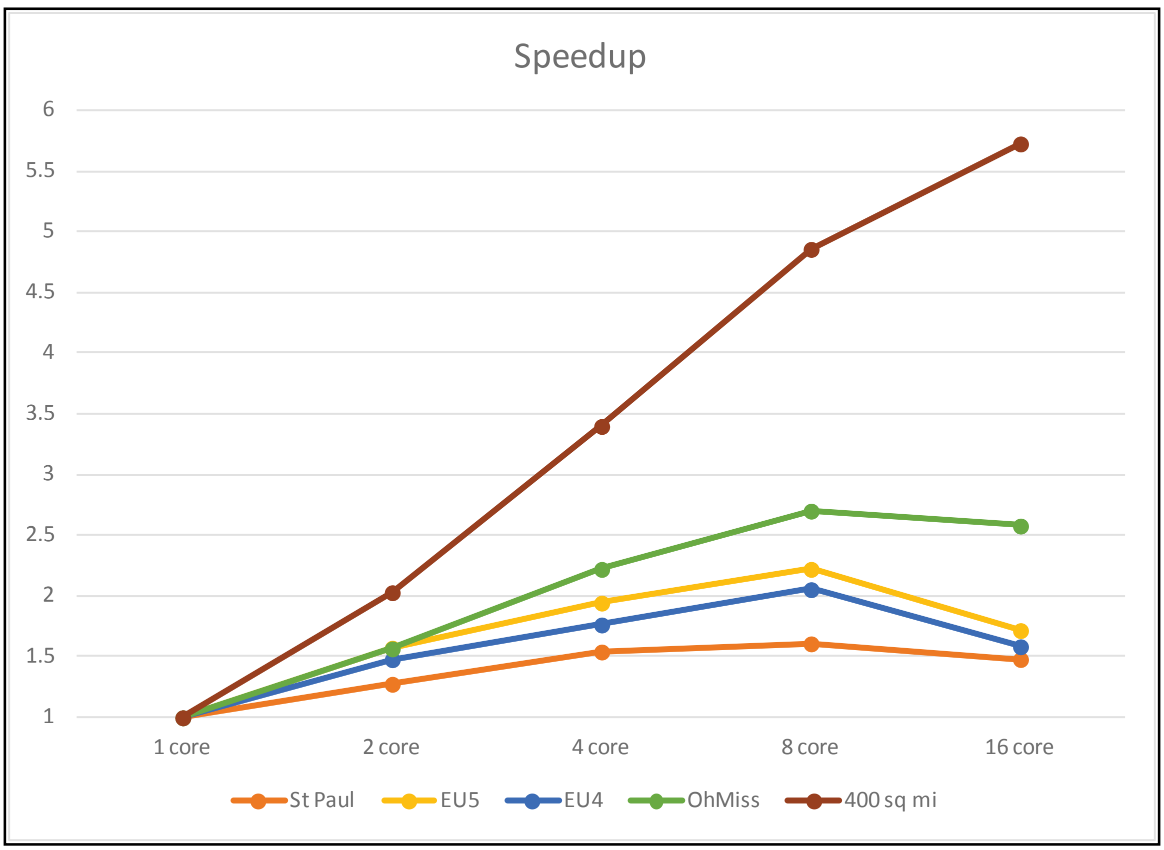


Figure 4-5. Number of processor cores vs. computational speed.

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) or Saint Venant Equations (Full Momentum)

The HEC-RAS two-dimensional computational module has the option of either running the **2D Diffusion Wave** equations, or the **2D Saint Venant** (Full Momentum) equations (sometimes referred to as the 2D shallow water equations). The default is the 2D Diffusion Wave equation set. In general, most 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 Saint Venant equations 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.

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 0.5 means that 50% 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): 0.5 (50%) 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).

Eddy Viscosity Mixing Coefficient: Default is Blank (not used)

The modeler has the option to include the effects of turbulence in the two dimensional flow field. Turbulence is the transfer of momentum due to the chaotic motion of the fluid particles as water contracts and expands as it moves over the surface and around objects. Turbulence within HEC-RAS is modeled as a gradient diffusion process. In this approach, the diffusion is cast as an Eddy Viscosity coefficient. To turn turbulence modeling on in HEC-RAS, enter a value for the Eddy Viscosity Mixing Coefficient for that specific 2D Flow Area. This coefficient requires calibration in order to get at an appropriate value for a given situation. The default in HEC-RAS is zero for this coefficient, meaning it is not used. The numerical scheme in HEC-RAS provides some numerical diffusion automatically. Additional diffusion using the Eddy Viscosity formulation can be obtained by entering a value greater than zero in this field. Below are some values for the Mixing Coefficient (D_T) that have been found to be appropriate under certain conditions (Table 4-2).

Table 4-2. Eddy Viscosity Mixing Coefficients

D_T	**Mixing Intensity**	**Geometry and surface**
$0.11\ to\ 0.26$	Little transversal mixing	Straight channel Smooth surface
$0.3\ to\ 0.77$	Moderate transversal mixing	Gentle meanders Moderate surface irregularities
$2.0\ to\ 5.0$	Strong transversal mixing	Strong meanders Rough surface

1D/2D Iteration Options. Default is zero (meaning this is not turned on)

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 so. 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

Two new 1D only computational options have been added, and one existing option has been removed in HEC-RAS 5.0. If the user selects the Tab labeled **General (1D Options)** on the **Computational Options and Tolerances** window, they will see the new format for these options, as well as the two new options. The two new 1D options are:

1. **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.

2. 1D Equation 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.

32-bit and 64-bit Computational Engines

HEC-RAS now has both 32-bit and 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. The software now automatically comes with both sets of computational engines. Users can control which engines to use from the main HEC-RAS window, by selecting **Options,** then **Program Setup**, then **Use 64-bit computational engines (when available)**. By default, the software will come with the 64-bit computational engines selected as the default. Uncheck the "Use 64-bit computational engines" option if you want the program to use the 32 bit computational engines. Our testing has not shown any differences between the results from the 64-bit and the 32-bit versions. However, there remains a very small possibility (however remote) that there could be a difference for some data set, especially a data set that is having stability problems (or is on the edge of having a stability problem).

CHAPTER 5

Viewing Combined 1D/2D Output using 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 5-1 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 Chapter 20 in the main HEC-RAS User's manual.

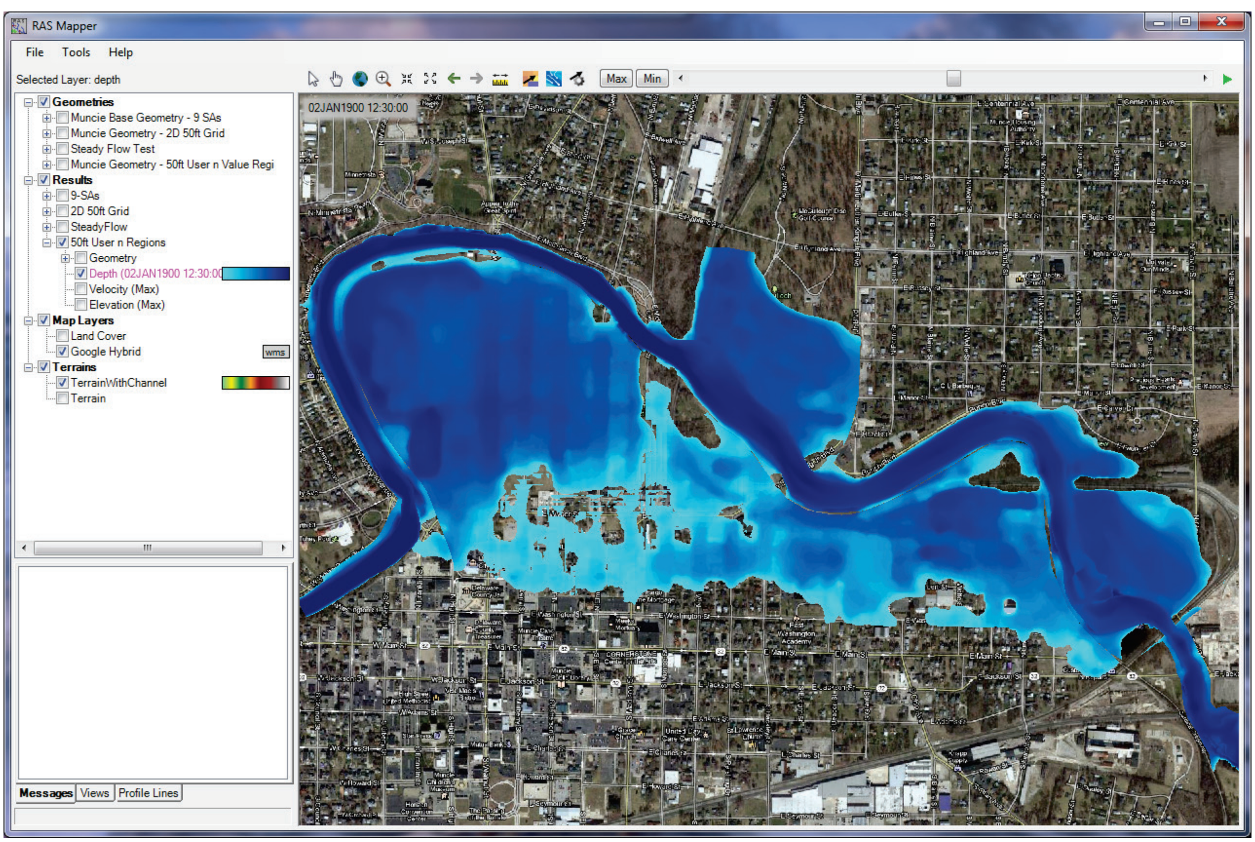


Figure 5-1. RAS Mapper window with combined 1D/2D flood inundation output displayed.

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. 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 for 2D flow areas.
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; arrival time; flood duration; and percent time inundated.
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.
5. Computed model results can be animated (dynamic mapping) or shown for a specific instance in time.
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).
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.
9. The user can now 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 now create User Defined Profile Lines, then request various types of output along those profile lines (i.e. WSE and terrain, velocity, depth, 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 5-2 below). Beneath the **Results | Plan Short ID** Layer, will be a tree of related results. By default there will be 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; Bank Line; River Edge Line; Junction; XS (cross sections); Storage Area; 2D flow area; XS Interpolation Surface, etc… Any or all of these Geometry layers can be turned on for visualization of model elements.

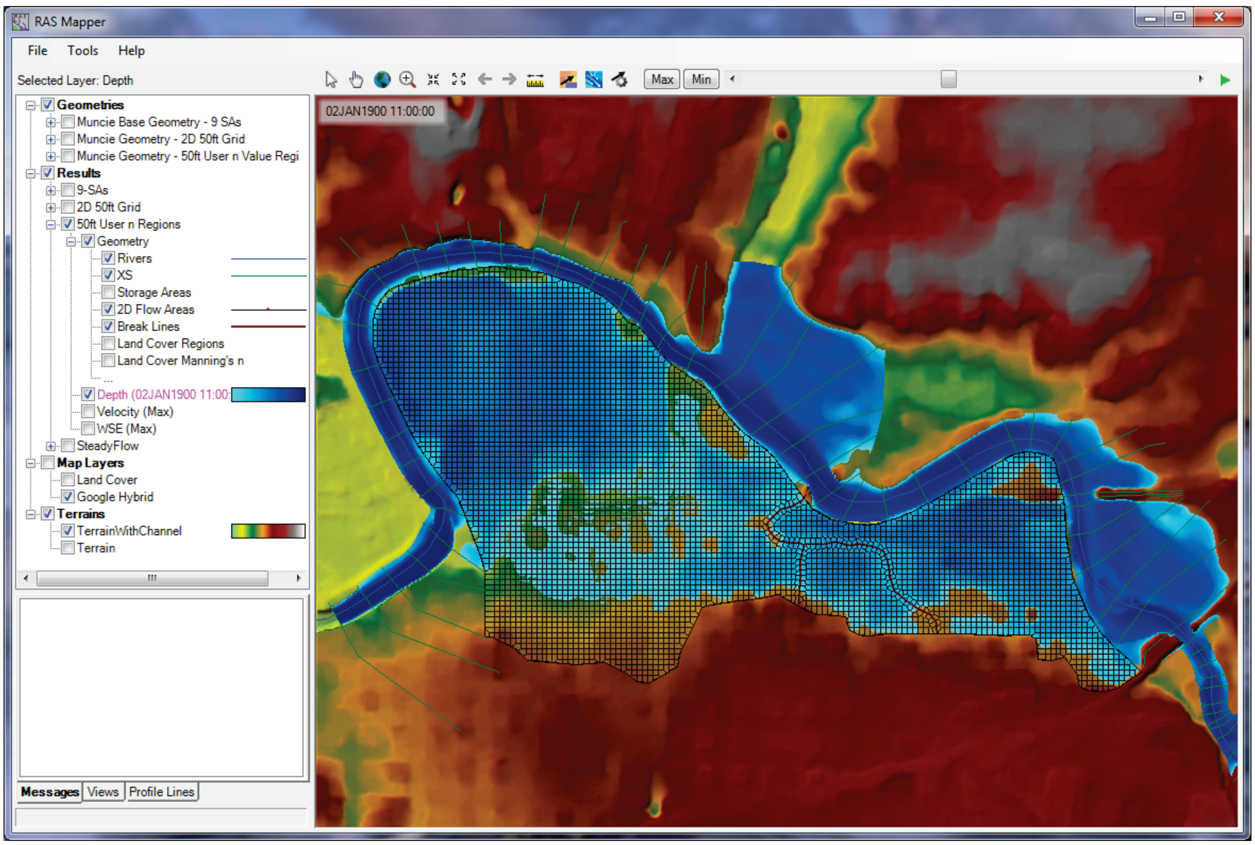


Figure 5-2. RAS Mapper with Default Results Layers shown.

By default, after a successful HEC-RAS model run, there will be three results layers called **Depth, Velocity, and 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, 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 depth 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 5-3). 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** from any of the Plan names listed in that window to create a new results map layer.

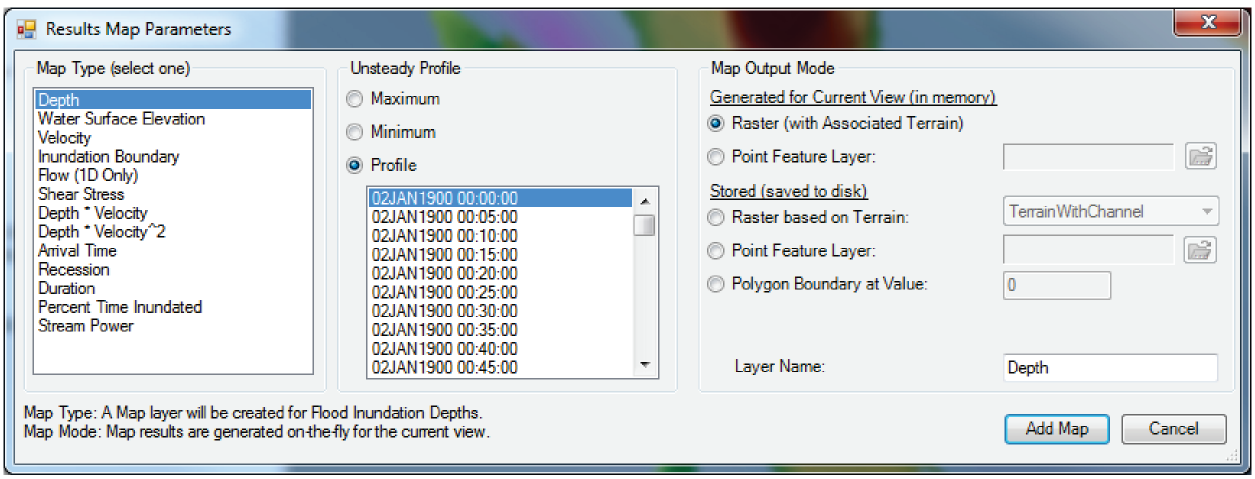


Figure 5-3. Example of the Results Map window used to create new results map layers.

As shown in Figure 5-3, 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 10 different Map Types (Table 5-1).

After a parameter 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 5-1. 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.
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.

Map Type	Description
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.

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 5-2). 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 5-2. 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.
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.

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 5-4.

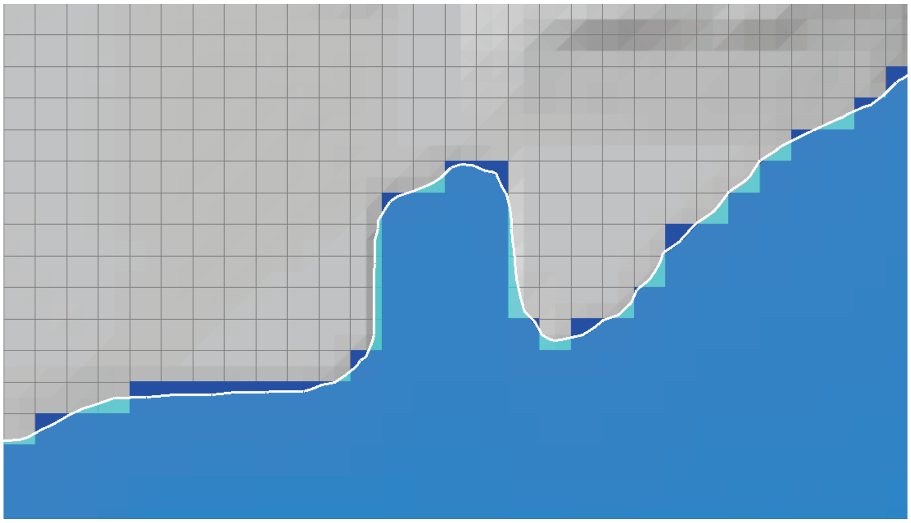


Figure 5-4. Differences in dynamic (smooth boundary) and stored (gridded) map results.

2D Mapping Options

RAS Mapper has two 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 **Sloping** (default) or as **Horizontal** within a 2D cell. The render mode selected will affect both the dynamic map and the stored map results. To select the Render Mode, right-click on the **Results** node and choose **Render Mode Options**, or select **Render Mode Options** from the **Tools** menu. An illustration of the two rendering modes is shown in Figure 5-5 while Figure 5-6 shows the spatial differences due to the rendering mode.

Sloping Water Surface (Default) – 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.

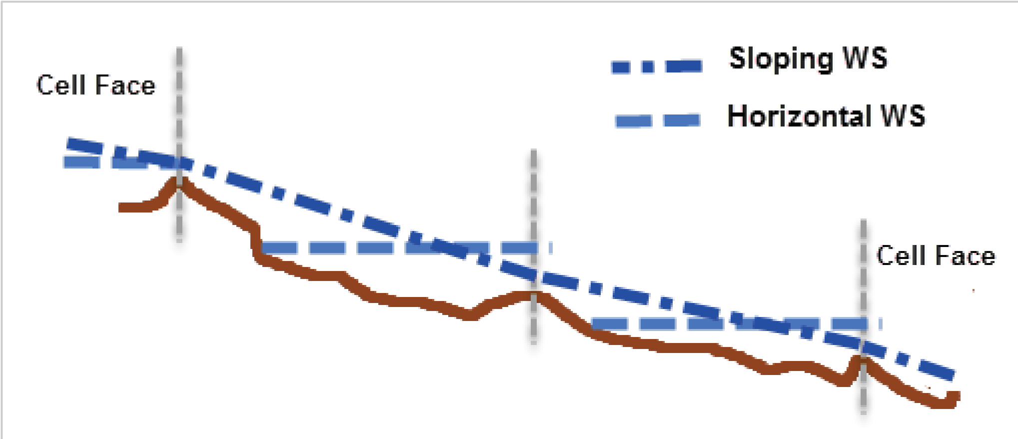


Figure 5-5. Comparison of Sloping and Horizontal water surface rendering for steep terrain.

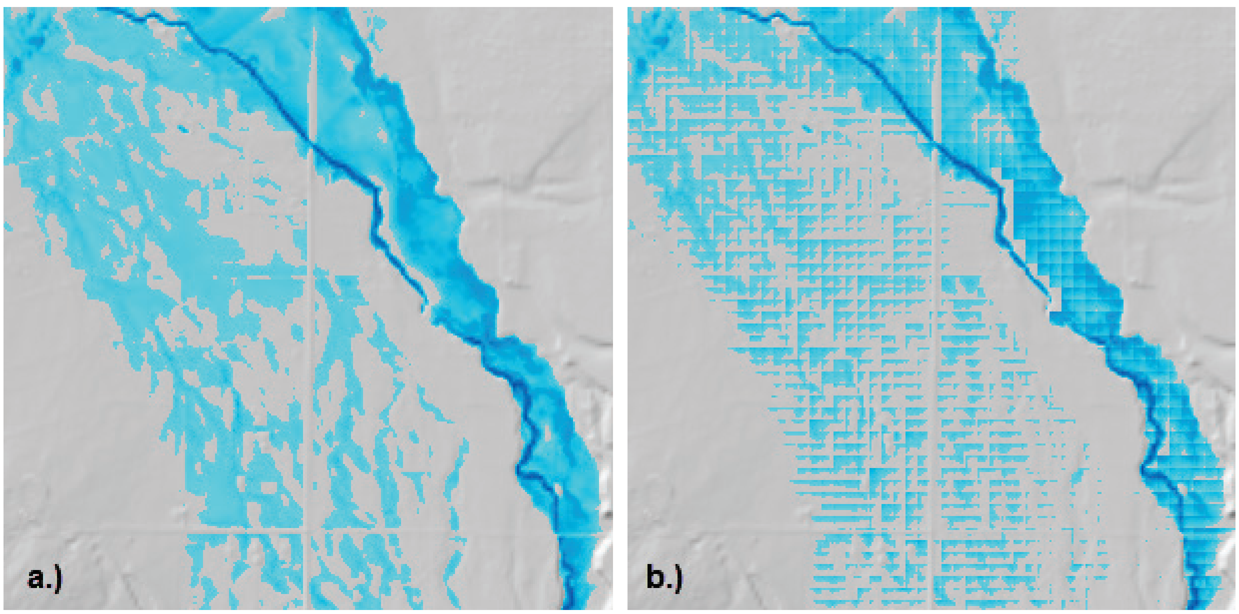


Figure 5-6. Comparison of (a) sloping and (b) horizontal water surface mapping option for inundation depth.

Dynamic Mapping

As shown in Figure 5-7, 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 four sub layers: **Geometry**, **Depth**, **Velocity**, and **WSE (water surface elevation)**. The **Geometry Layer** represents the geometry data used in the run, and written to the output file. The other layers (**Depth**, **Velocity**, and **Elevation**) 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; Zoom to Layer; Remove Layer; Move Layer; and Export Map for use in Google Map/Earth (Figure 5-7).

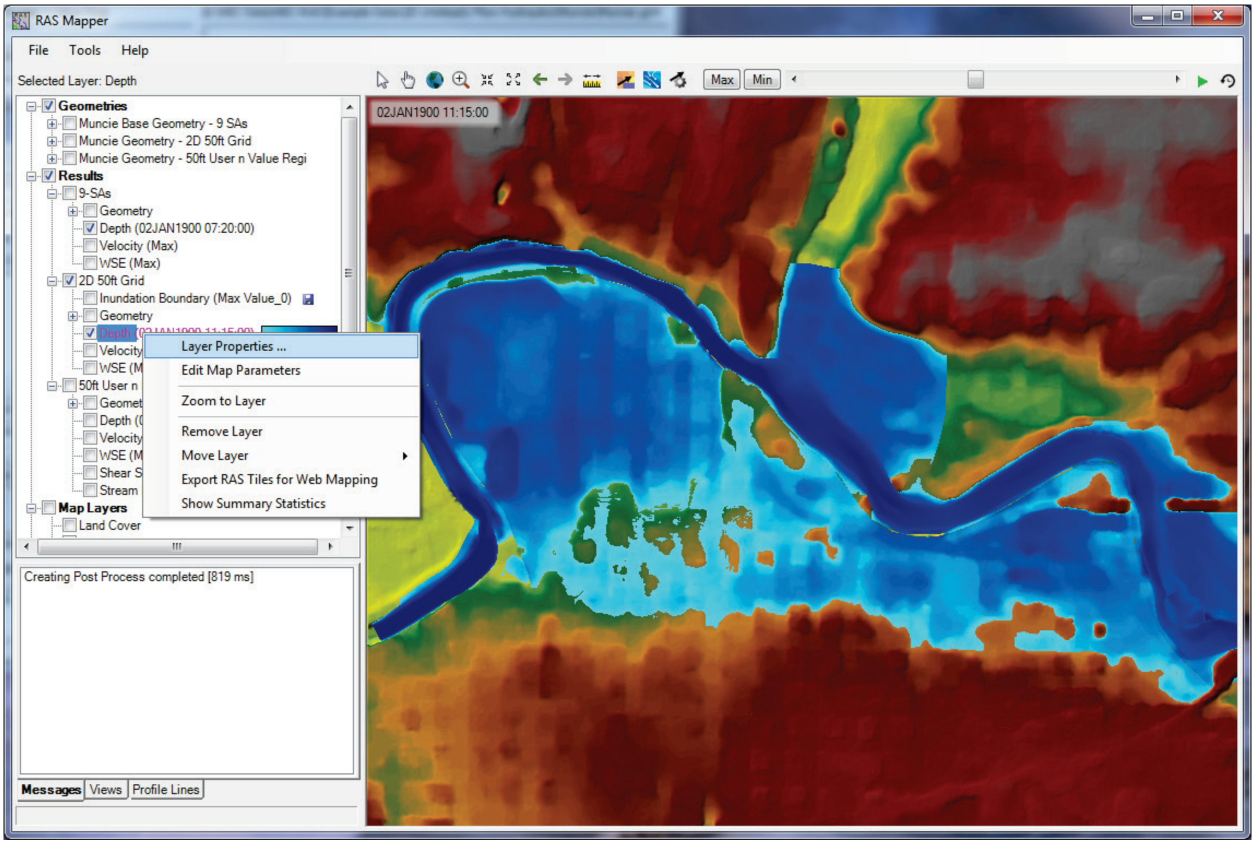


Figure 5-7. RAS Mapper with the Options menu displayed for a depth output layer.

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.

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

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 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 5-8 will appear.

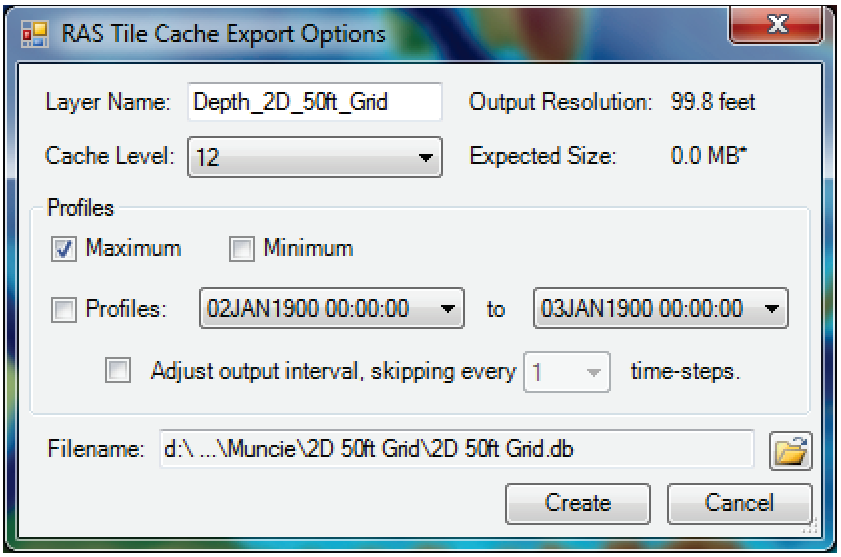


Figure 5-8. Tile Cache Options Editor.

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.

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 (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 5-9. 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.

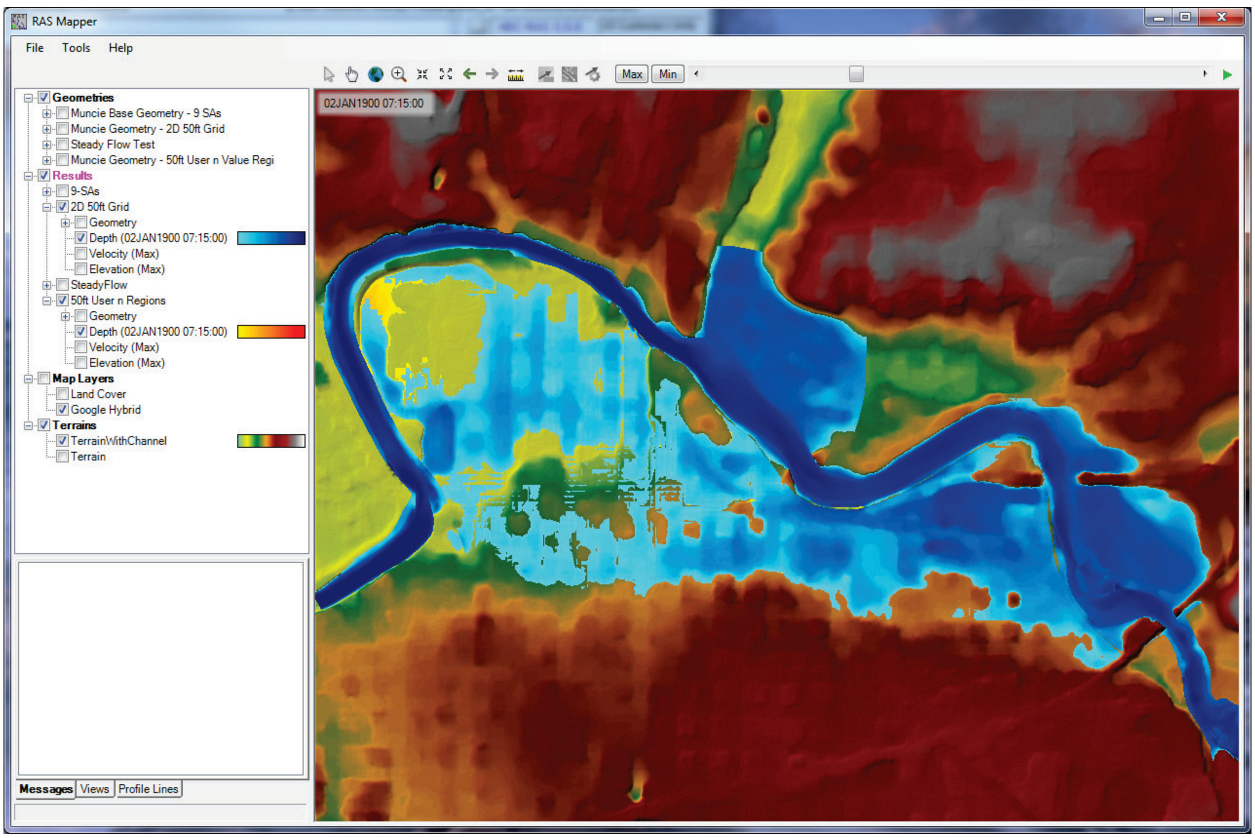


Figure 5-9. Dynamic Mapping/Animation Tool Bar.

As shown in Figure 5-9, 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.

Future versions of the **Animation Tool** will have a Record button, so the user can play the animation and record it to a video file. For now if the user wants to create a movie file use the Snagit® software (or similar screen capture utility package) to capture the screen while animating the inundation results.

Creating Static (Stored) Maps

The user can create a static map (map stored to the disk) at any time from RAS Mapper by selecting the **Tools | Manage Results Map** menu item. When this option is selected the window shown in Figure 5-10 will appear.

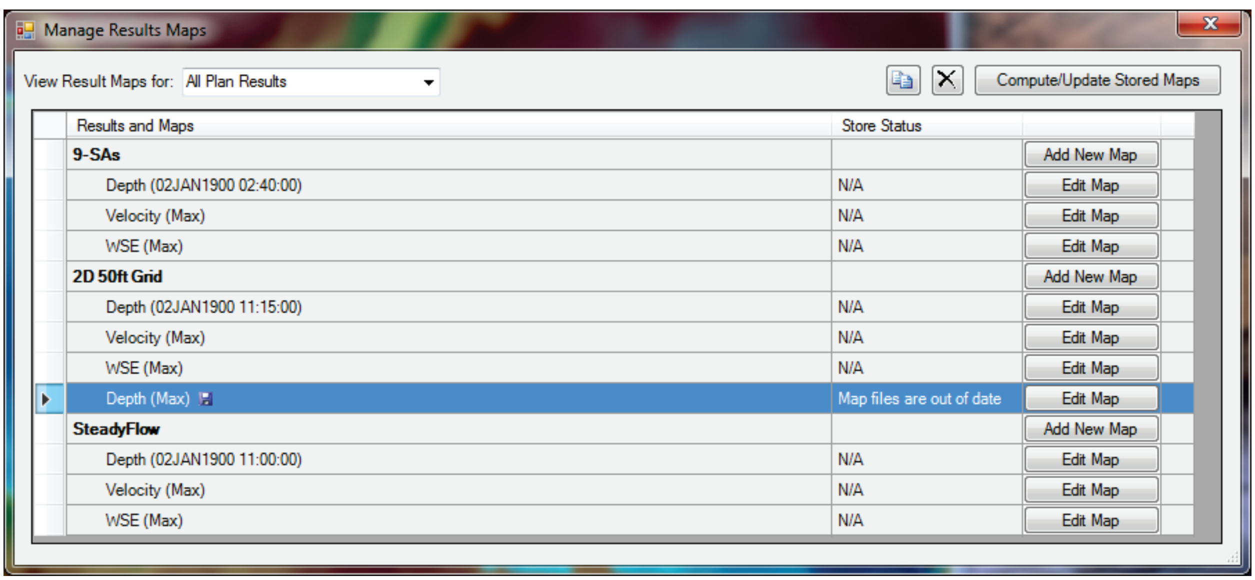


Figure 5-10. Results Mapping Window.

As shown in Figure 5-10, 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 5-11.

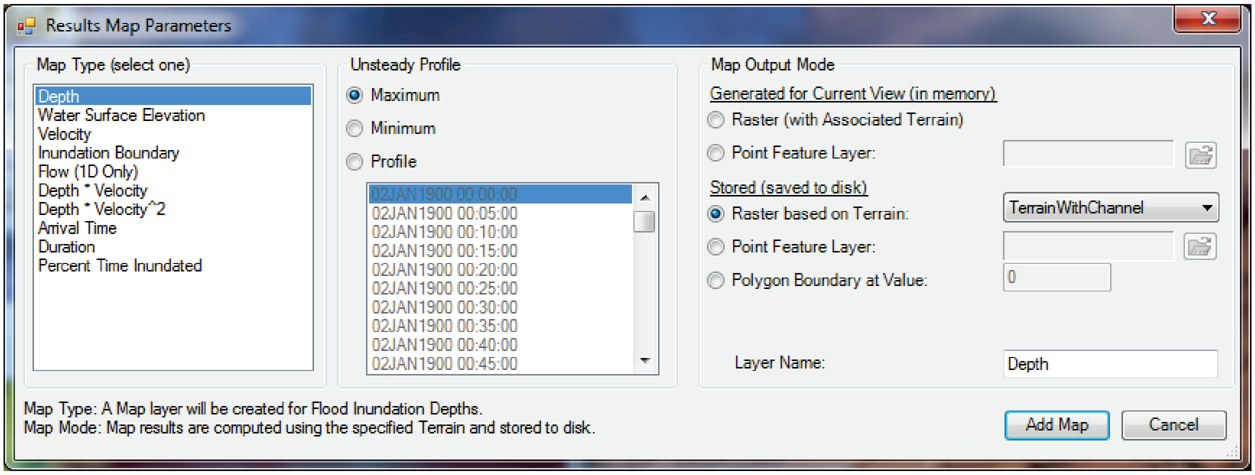


Figure 5-11. Add New Results Map Layer Window.

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 5-11). 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 5-10). 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 now 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 5-12 below.

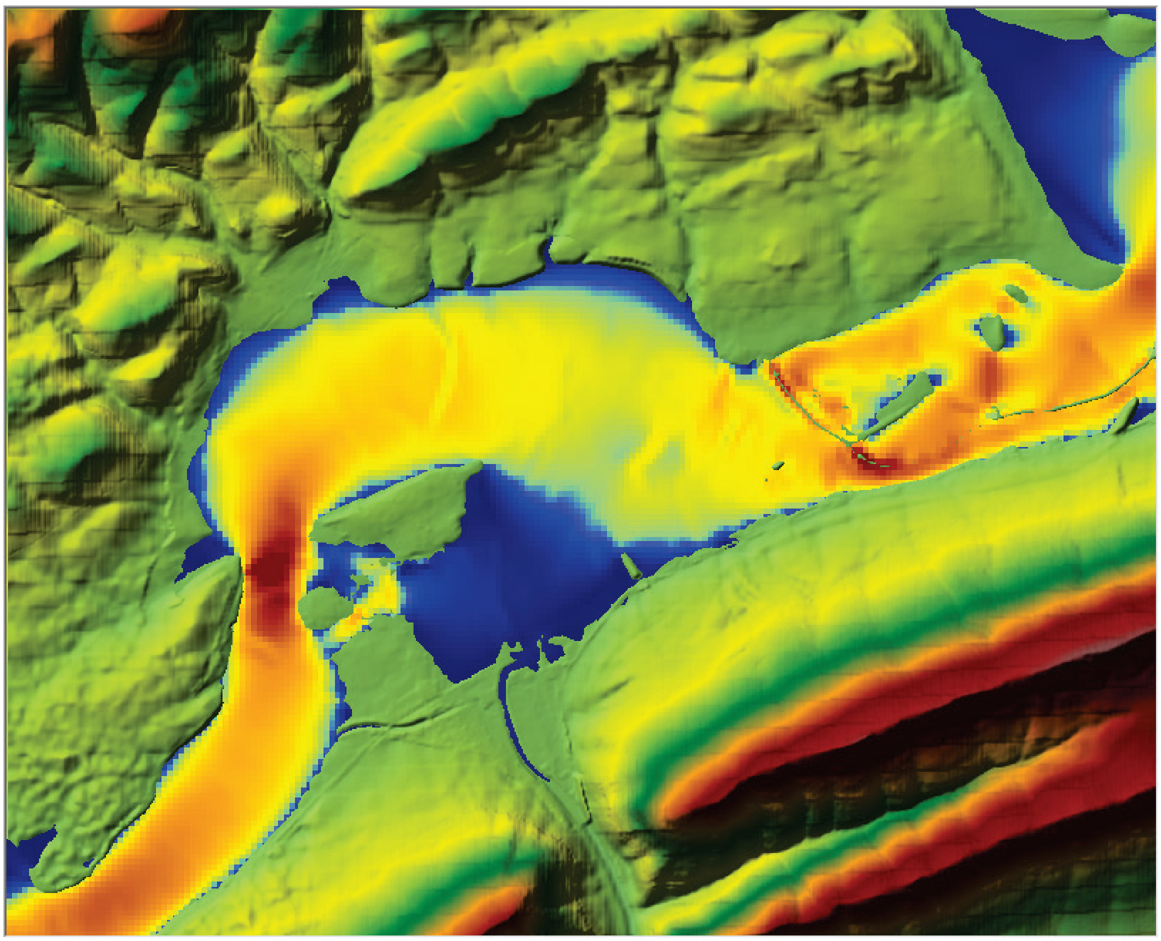
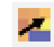


Figure 5-12. Example Color based Velocity Plot

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 5-13 below:

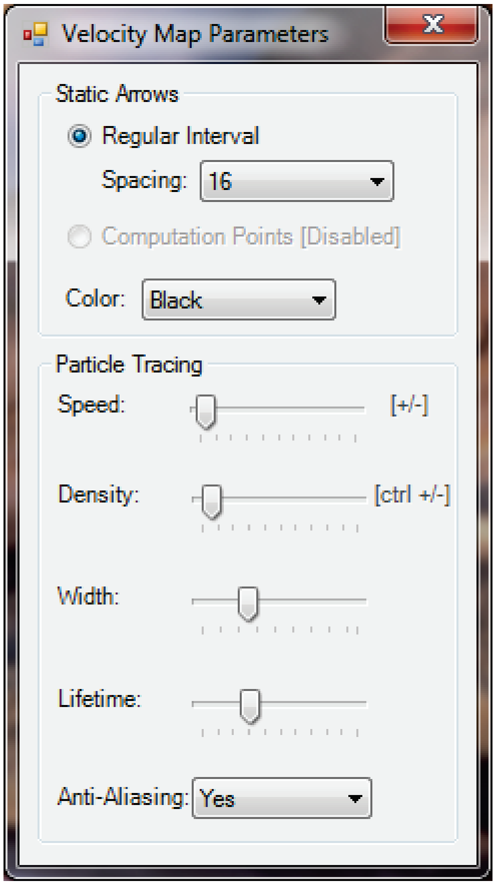


Figure 5-13. Velocity Map Parameters.

The **Velocity Map Parameters** settings window (see Figure 5-13) 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 5-14 is a velocity plot with magnitude/direction arrows turned on.

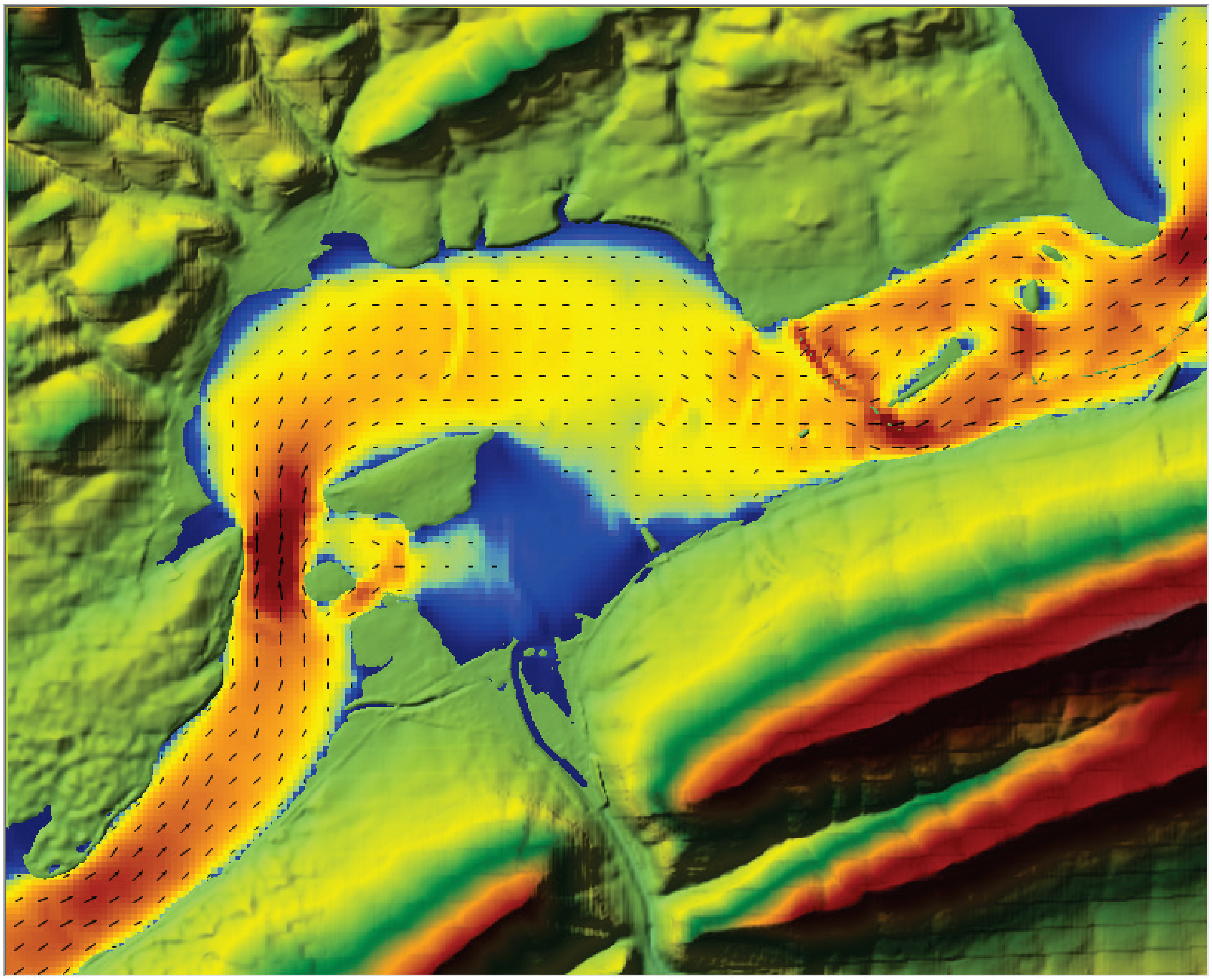


Figure 5-14. Example Velocity Plot with Color and Direction/Magnitude Arrows.

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 5-13) 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); 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 5-15 is an example plot with the velocity particle tracking option turned on, and being displayed on top of a depth layer plot.

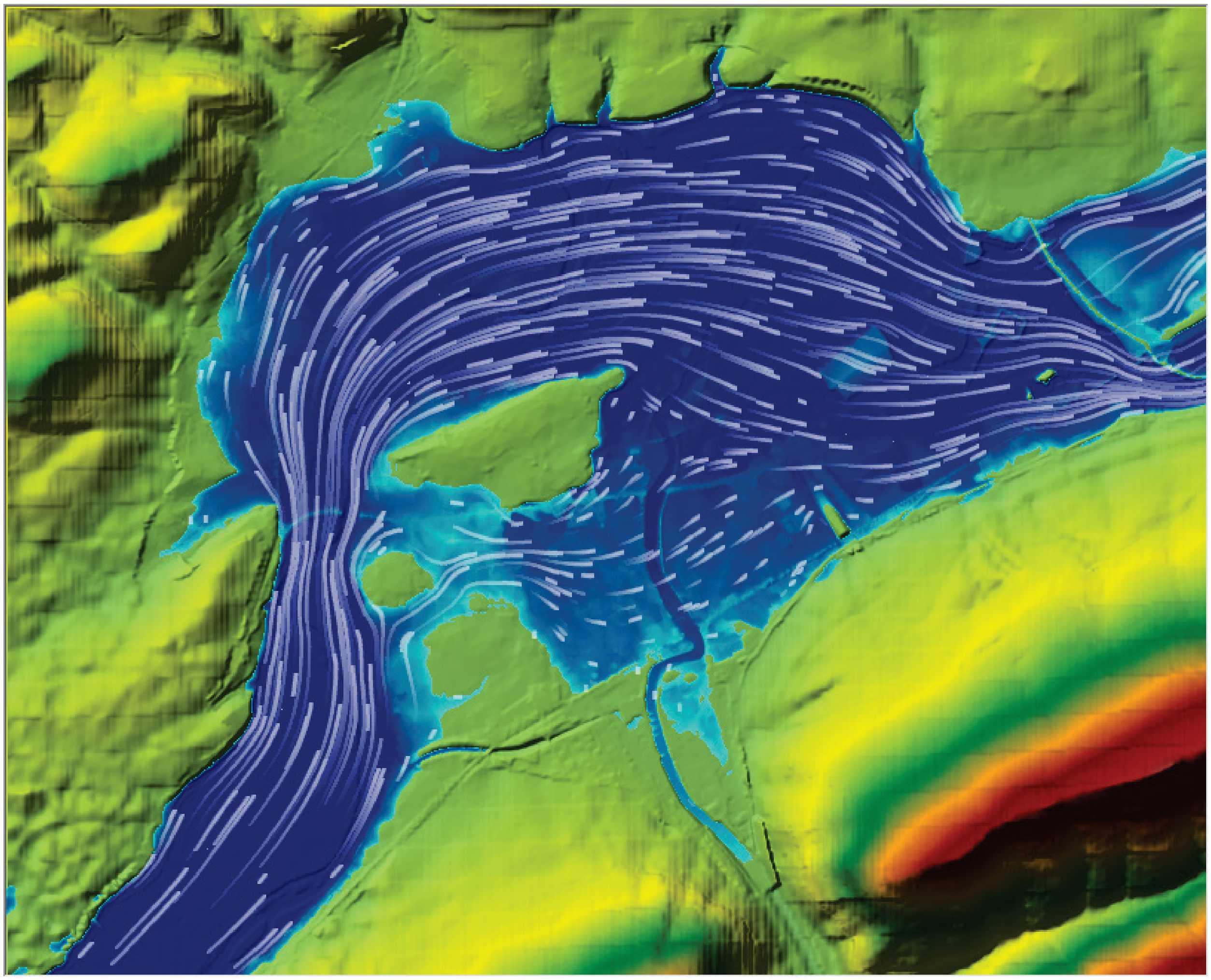


Figure 5-15. Example of the Particle tracing visualization option on top of a depth layer.

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 5-16.

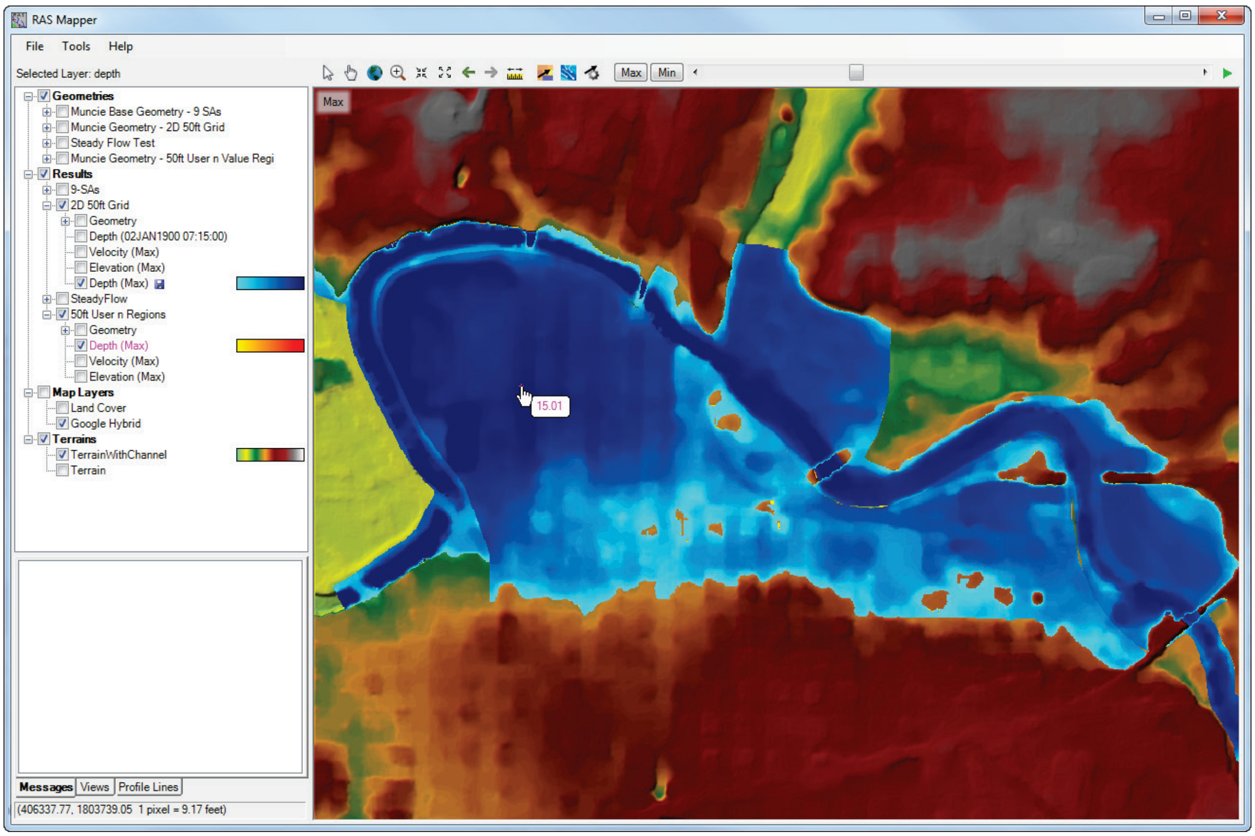


Figure 5-16. Example of Querying a value from the active map layer.

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 5-17).

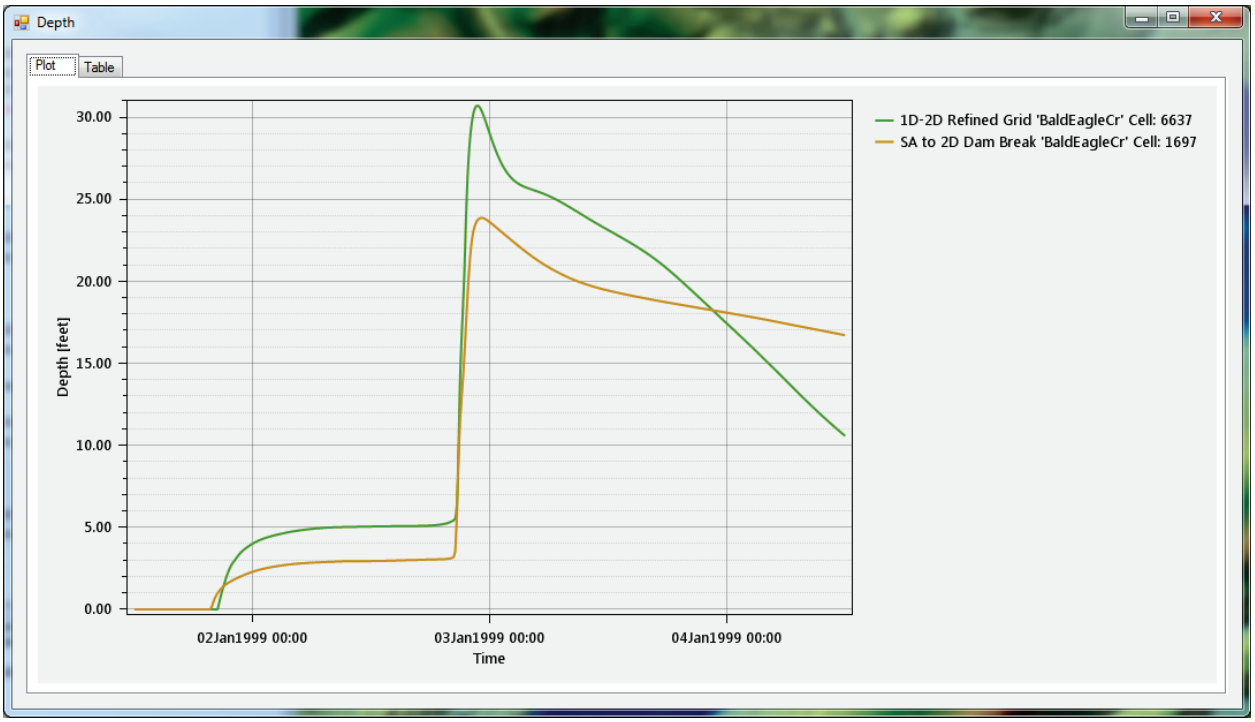


Figure 5-17. Example Time Series Plot of Depth from two different Plans.

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 5-18) the following time series: 2D Cell Water Surface Time Series; 2D Cell Water Depth Time Series; 2D Face Point Velocity Time series (this is a point velocity of the closest Cell Face Point when selected); 2D Face Perpendicular Velocity Time Series (the component of the velocity that is perpendicular to this face); 2D Face Shear Time Series (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 5-19.

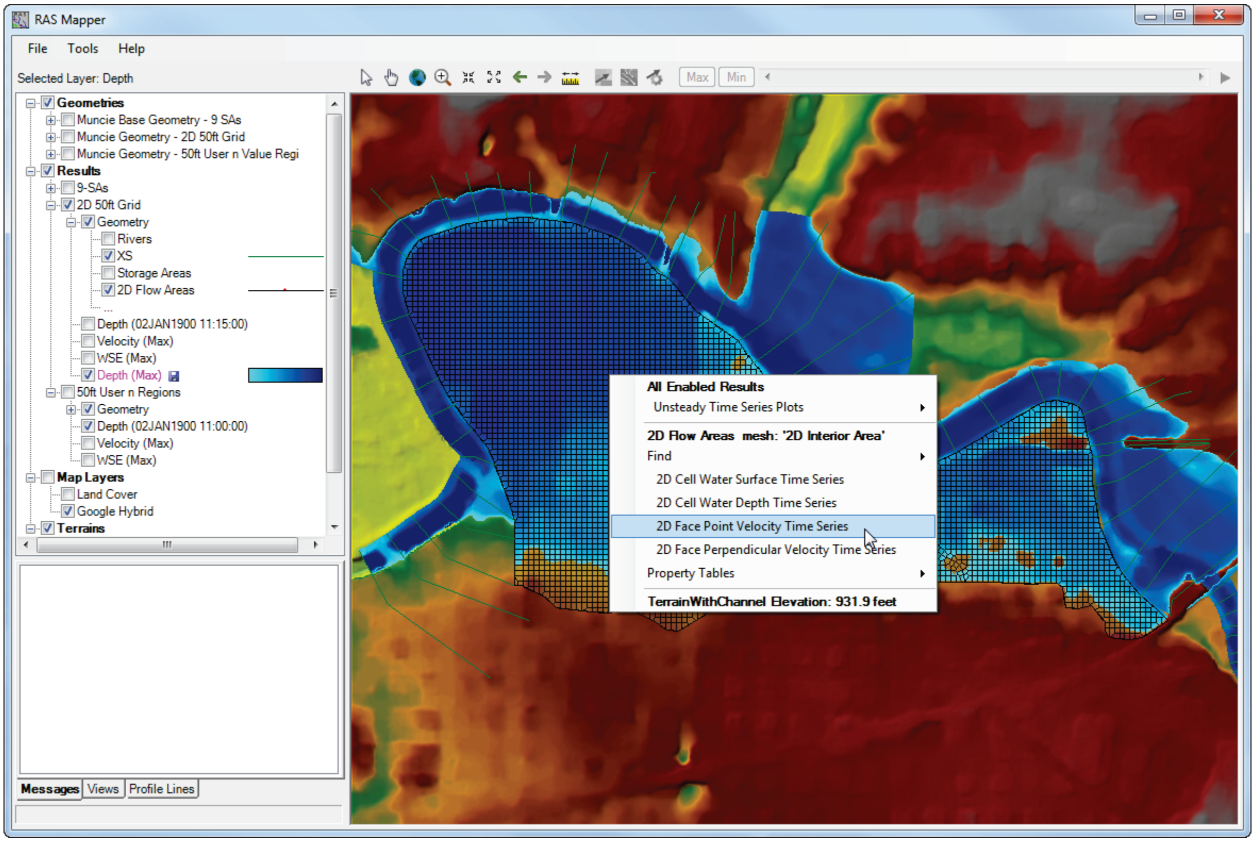


Figure 5-18. Example showing options for displaying 2D Model Output Time Series Results.

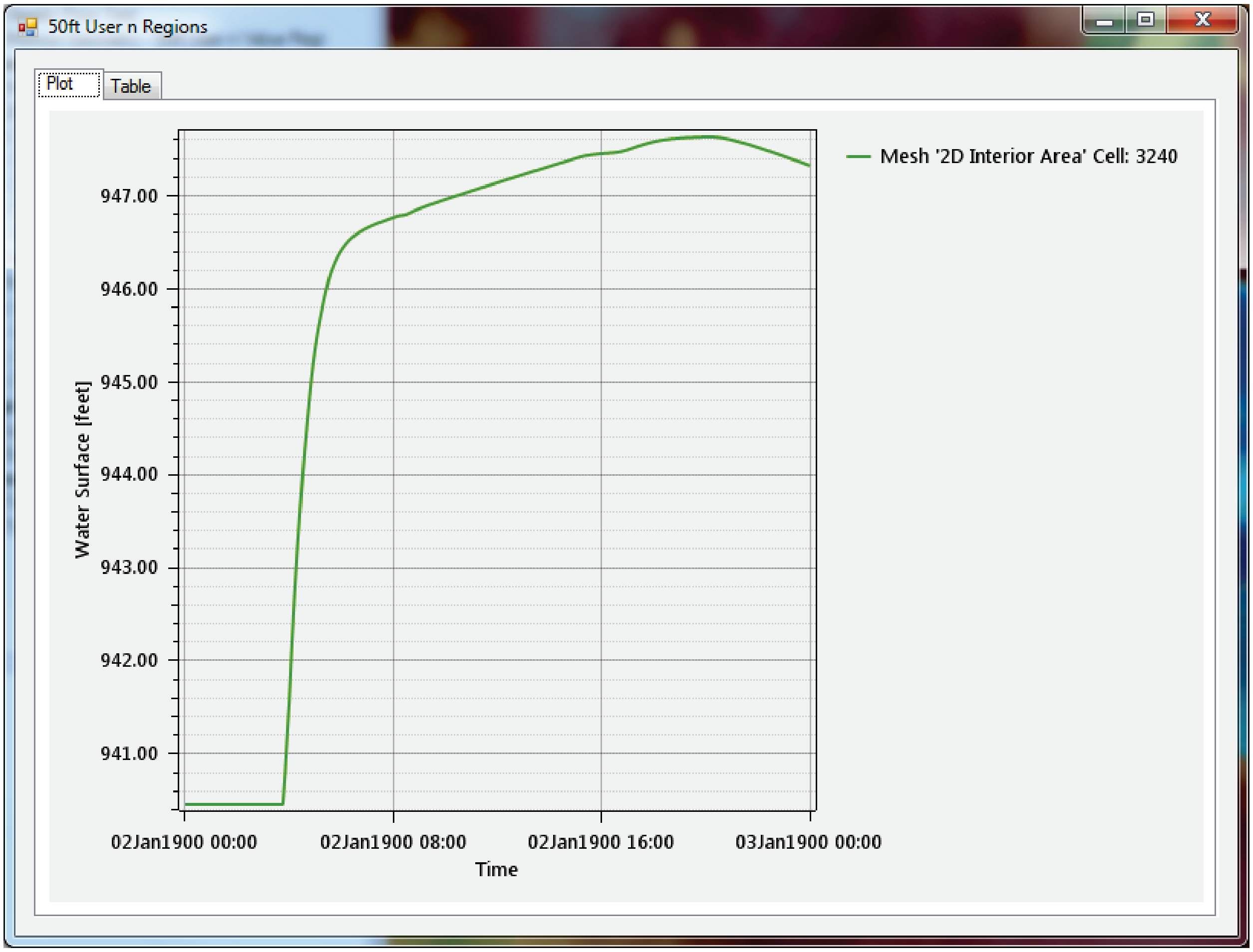


Figure 5-19. Example Time Series Hydrograph Plot of a 2D cell.

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 is turned on over top of the line. To use the profile line option, the user must select 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, a + (add a profile line) button, and **X** (delete profile line) button. Select the + button to add a new line. 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.

To plot computed results above 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 5-20). This will activate that line. Next, right click on the line, this will bring up a popup menu, of which once 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 5-21).

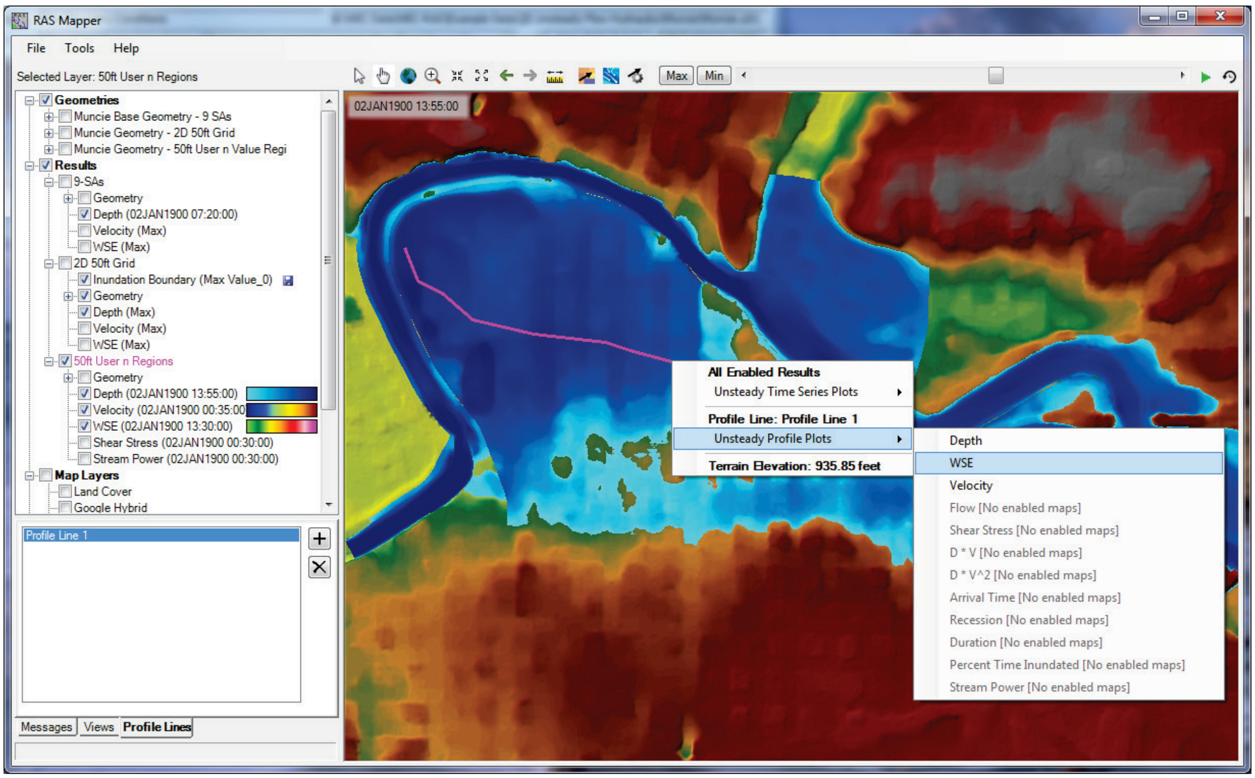


Figure 5-20. Profile Line turned on and selected for plotting options.

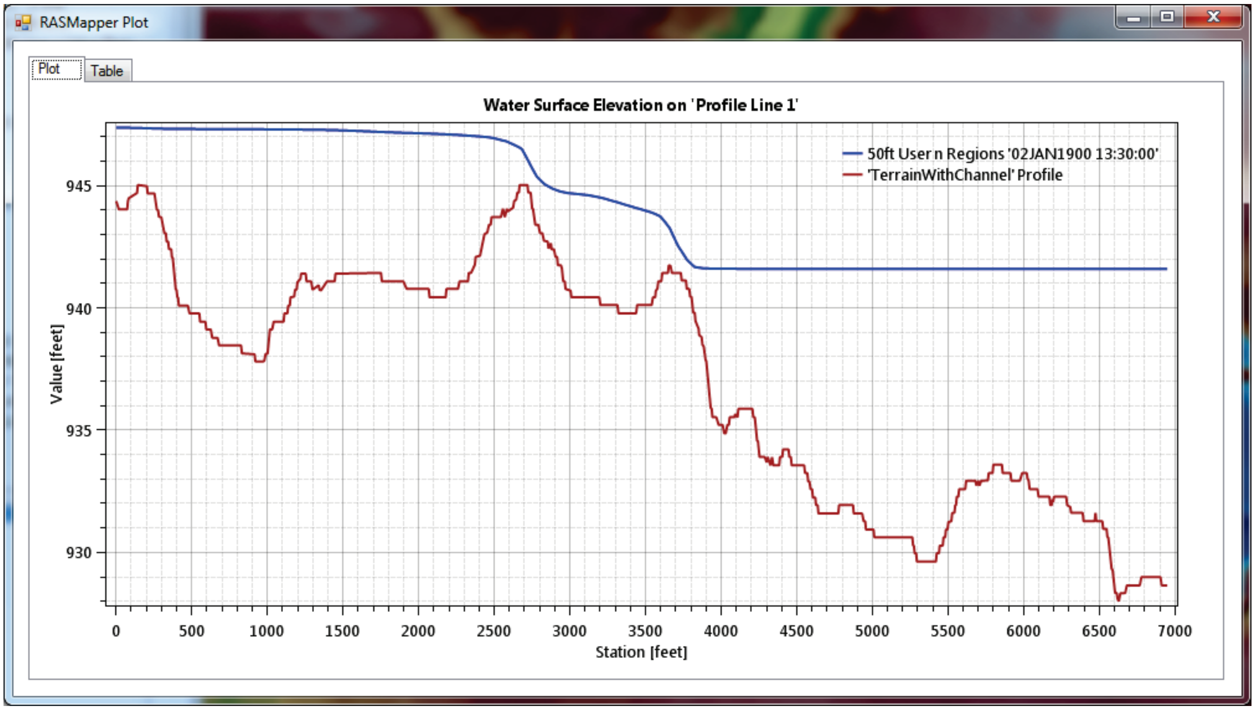


Figure 5-21. Example Profile line Plot of Water Surface Elevation (WSE)

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 **Tools | Set Projection For Project** menu item from the RAS Mapper menu bar. When this option is selected the window shown in Figure 5-22 will appear.

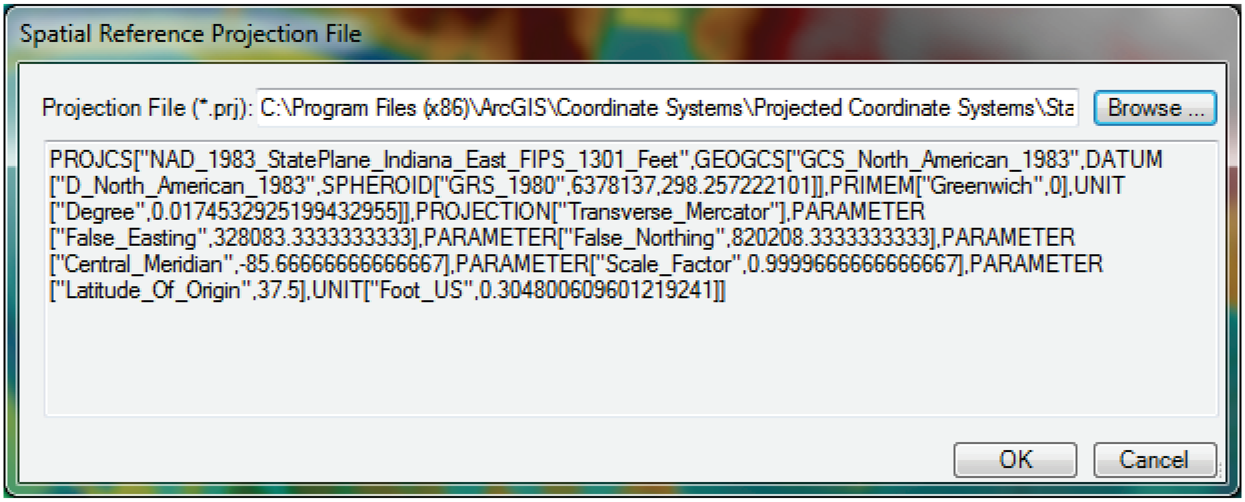


Figure 5-22. Editor to set the RAS project's spatial reference system.

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 5-22. 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 5-23). 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 5-24.

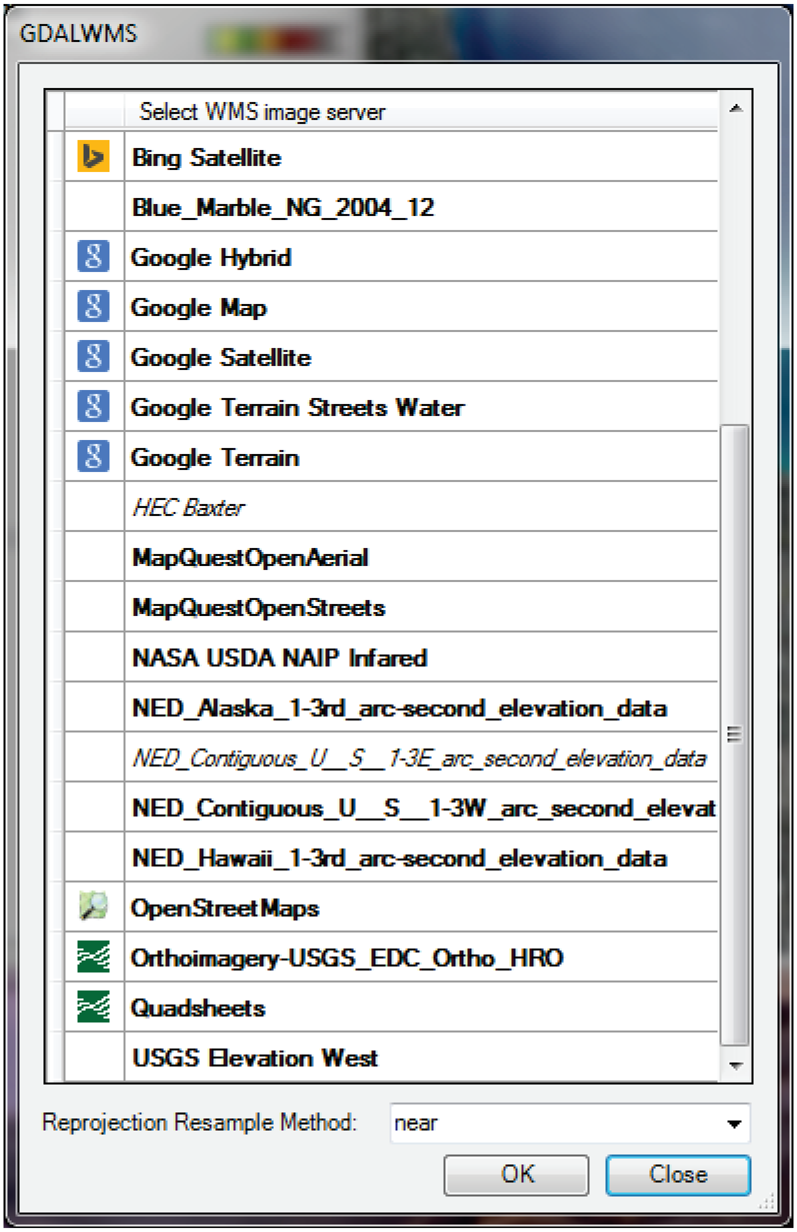


Figure 5-23. Web mapping services available in RAS Mapper.

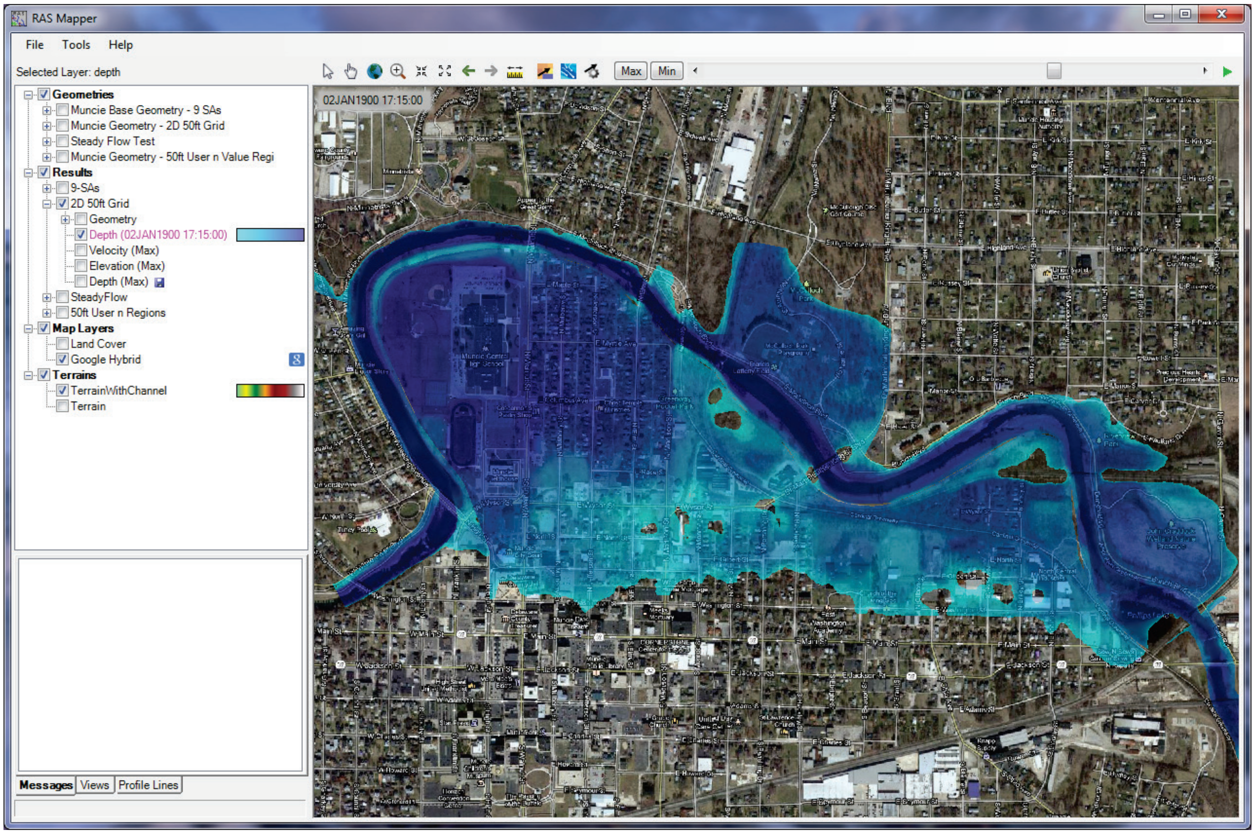


Figure 5-24. RAS Mapper with background Web imagery loaded with an inundation depth grid overlayed.

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 **Add Map Data Layers** option. The file browser window will appear, allowing the user to navigate to the desired file and select it. See Figure 5-25 below:

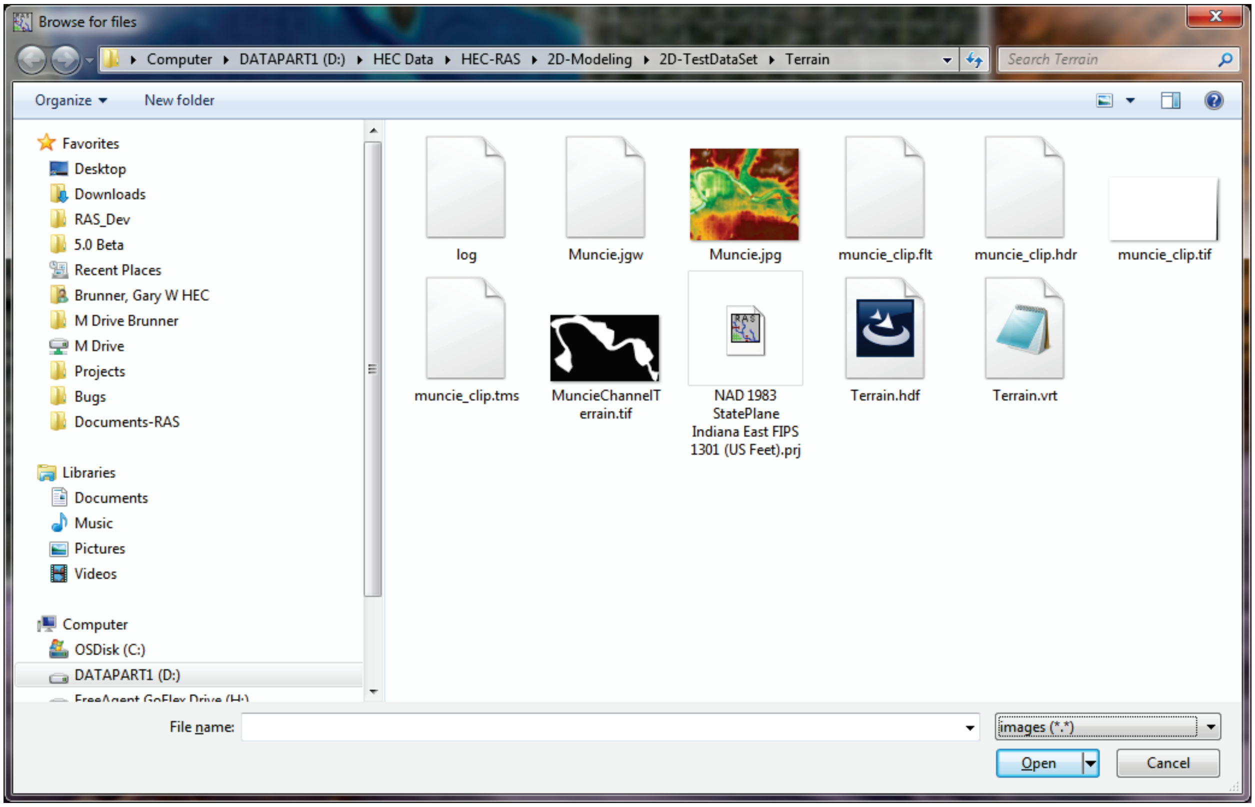


Figure 5-25. Example File chooser for bringing in Map Layers to be used for background display.

National Levee Database

The last tool to discuss is the link to the National Levee Database (NLD). If the user selects **Import NLD** from the RAS Mapper **Tools** menu, then a window will appear as shown in Figure 5-21. 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 5-26).

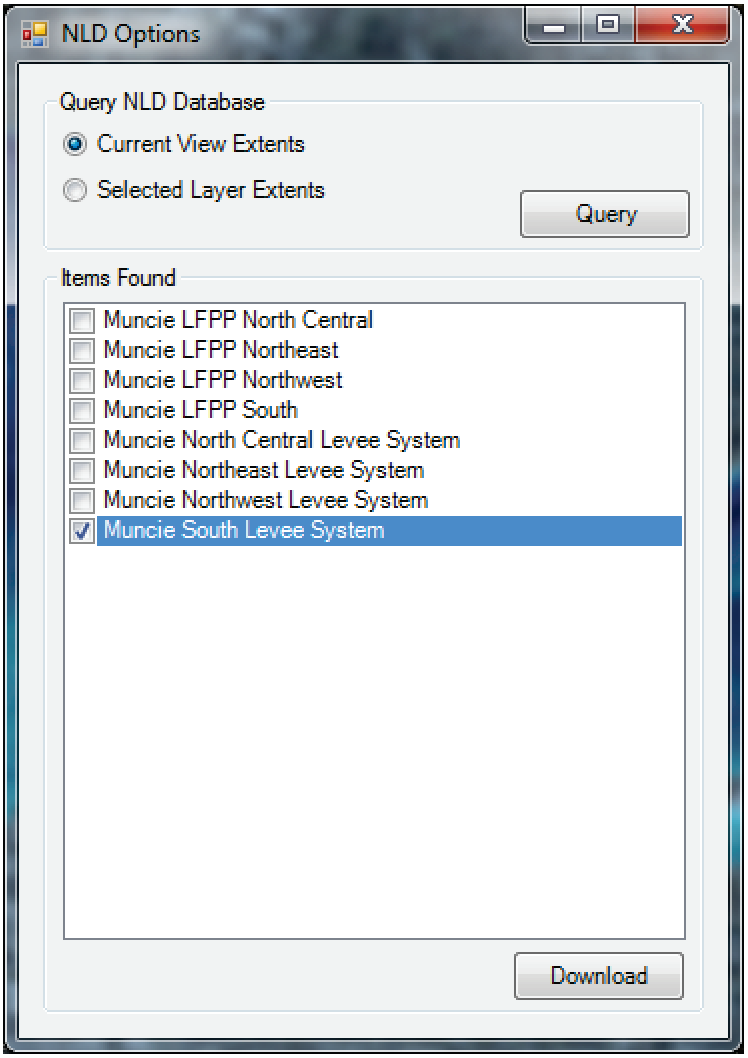


Figure 5-26. List of levees and floodwalls sent to RAS from the NLD.

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 5-26. 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 version 5.0.

2D Output File (HDF5 binary file)

The Output for the 2D flow area computations, as well as some 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:

http://www.hdfgroup.org/products/java/release/download.html

Download and Install the Windows 64-bit version if you have a 64-bit operating system. The 64-bit version can read both 32 and 64-bit files.

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 5-27 is an example HDF file output from an HEC-RAS 1D/2D model run. As shown in Figure 5-27, 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. Depth: Depth of water in each of the cells (Ft or m)
2. Face Velocity: Normal face velocity (the component of the velocity perpendicular to that face) (ft/s or m/s)
3. Node X Vel: The X component of the velocity vector at a Face Point (ft/s or m/s)
4. Node Y Vel: The Y component of the velocity vector at a Face Point (ft/s or m/s)
5. Face Shear Stress: Average shear stress over the Face (lb/ft^2 or N/m^2)
6. Water Surface: Water surface elevation for each cell (ft or m)

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. Water Surface Min: 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 "Write Velocity data at the face node locations in 2D Meshes".

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).

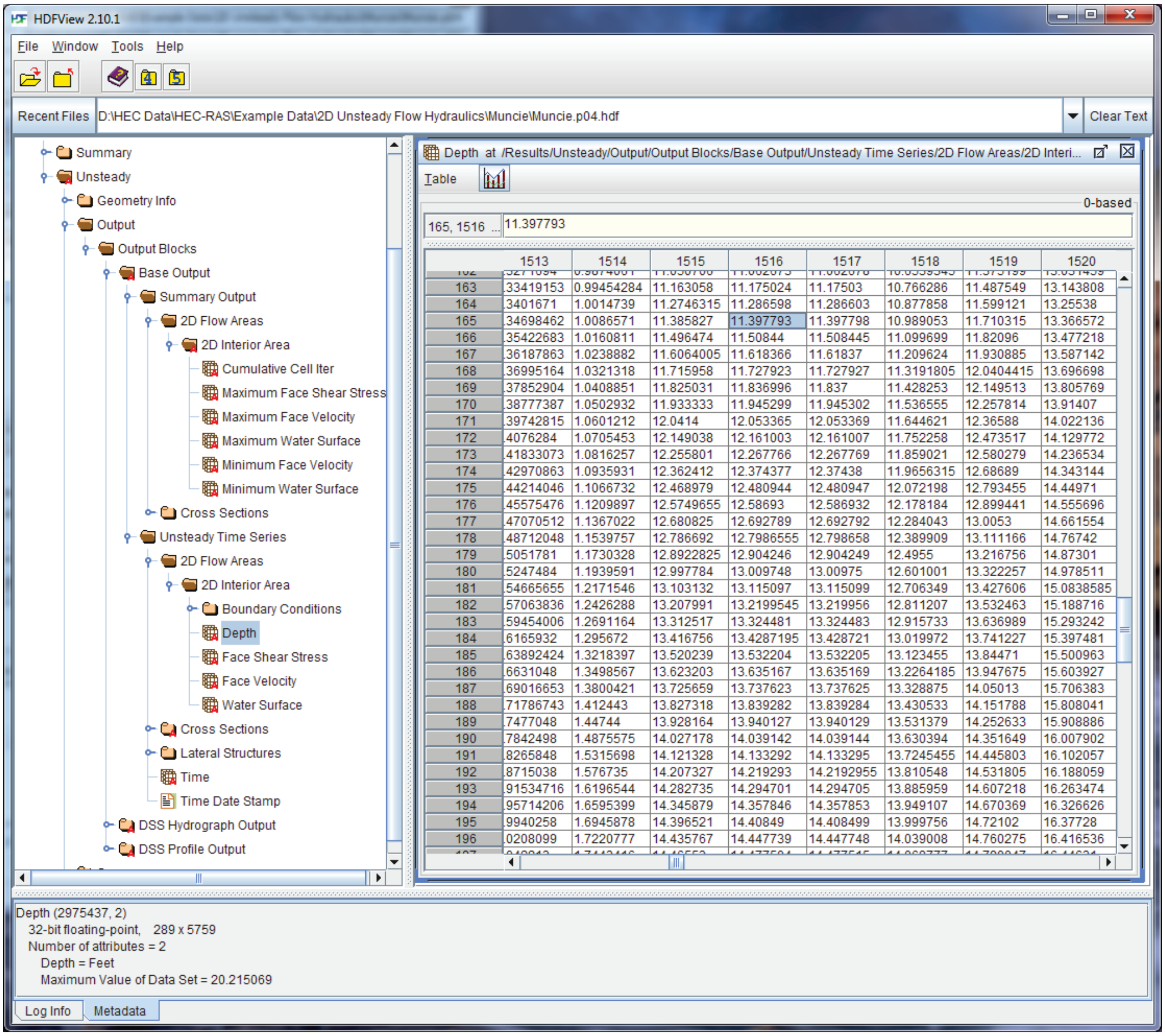


Figure 5-27. Example HDF File Output from HEC-RAS 1D/2D Model Run

CHAPTER 6

Steady vs. Unsteady Flow and 1D vs. 2D Modeling

"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. 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…?

9. 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…)?
10. What is the model purpose and expected level of accuracy required?
11. How much time and money do you have to get it done?
12. 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.

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 stage for the area of interest.
- The events being modeled are very dynamic with respect to time (i.e. Dambreak 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).
- Flow reversals occur during the event.
- Dynamic events such as levee overtopping and breaching occur during the event.
- Extremely flat river systems, where gravity is not necessarily the only significant driving force of the flow.

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 real-time river forecasting and mapping, with reasonable accuracy.

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.
- 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 I think 1D modeling will produce as good as or better results than 2D modeling for real time flood forecasting applications, with less effort (both 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 is more often than not a much greater contributor to forecast/modeling error than any differences arising from 1D versus 2D model choices.

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. Sami-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/

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.

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.

Sehili, A., L.G. Gunther, C. Lippert 2014. High-resolution subgrid models: background, grid generation, and implementation. Ocean Dynamics, Vol. 64, 519-535.

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.

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)