Combined 1D and 2D Modeling with HEC-RAS

Gary W. Brunner, HEC

May, 2014

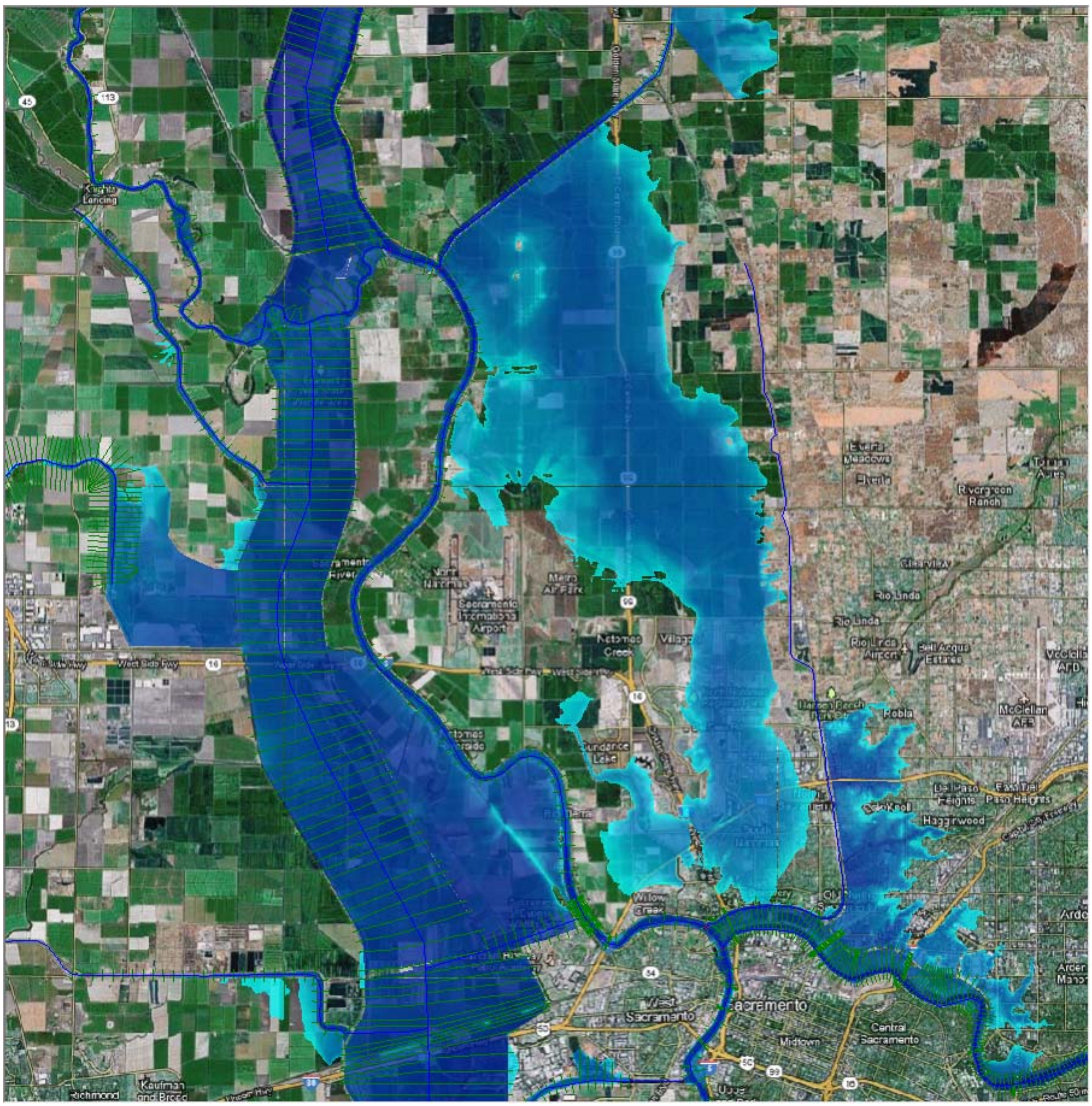


Table of Contents

Combined 1D and 2D Modeling with HEC-RAS

I. Overview

HEC has added the ability to perform two-dimensional (2D) hydrodynamic flow 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 (Full Saint Venant equations or Diffusion Wave equations), as well as combined one-dimensional and two-dimensional (1D/2D) unsteady-flow routing. The two dimensional 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 Flow Areas behind levees
- Combined 1D channels with 2D floodplain areas
- 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 Breaching 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).

Two-dimensional (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 you already know 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 RAS Mapper features. For assistance with 1D unsteady flow modeling, and how to use the User Interface, please review the HEC-RAS User's manual.

A. HEC-RAS Two-Dimensional Flow Modeling Advantages/Capabilities

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

1. **Can perform 1D, 2D, and combined 1D and 2D Modeling.** HEC-RAS can perform 1-Dimensional (1D) modeling, 2-Dimensional (2D) modeling (no 1D elements), and combined 1D and 2D modeling. The ability to perform combined 1D and 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. **Full Saint Venant or Diffusion Wave Equations in 2D.** The program solves either the full 2D Saint Venant equations 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. While the 2D Full Saint Venant equations are more applicable to a wider range of problems. However, many modeling situations can be accurately modeled with the 2D Diffusion Wave equations. Since users can easily switch between equation sets, each can be tried for any given problem to see if the use of the 2D Full Saint Venant equations is warranted.

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 approach provides a measure of improved stability and robustness over traditional finite difference and finite element techniques. The wetting and drying of 2D elements is very robust with the finite volume solution algorithm in HEC-RAS. 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 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, each time step, as the interior area fills up. Additionally, flow can go back out 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 structured or unstructured computational meshes. This means that computational cells can be triangles, squares, rectangles, or even five and six-sided elements (the model is limited to elements with up to eight sides). The mesh can be a mixture of cell shapes and sizes. The outer boundary of the computational mesh is defined with a polygon. The computational cells that form the outer boundary of the mesh can have very detailed multi-point lines that represent the outer face(s) of each cell.

6. **Detailed Hydraulic Table Properties for Computational Cells and Cell Faces.** Within HEC-RAS, computational cells do not have to have a flat bottom, and Cell Faces do not have to be straight line, with a single elevation. Instead, each Computational Cell and Cell Face is based on the details of the underlying terrain. Each cell, and cell face, of the computational mesh is pre-processed in order to develop 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 mesh 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 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 less

computations, which means much faster run times. An example computational mesh with detailed terrain below is illustrated in Figure 1.

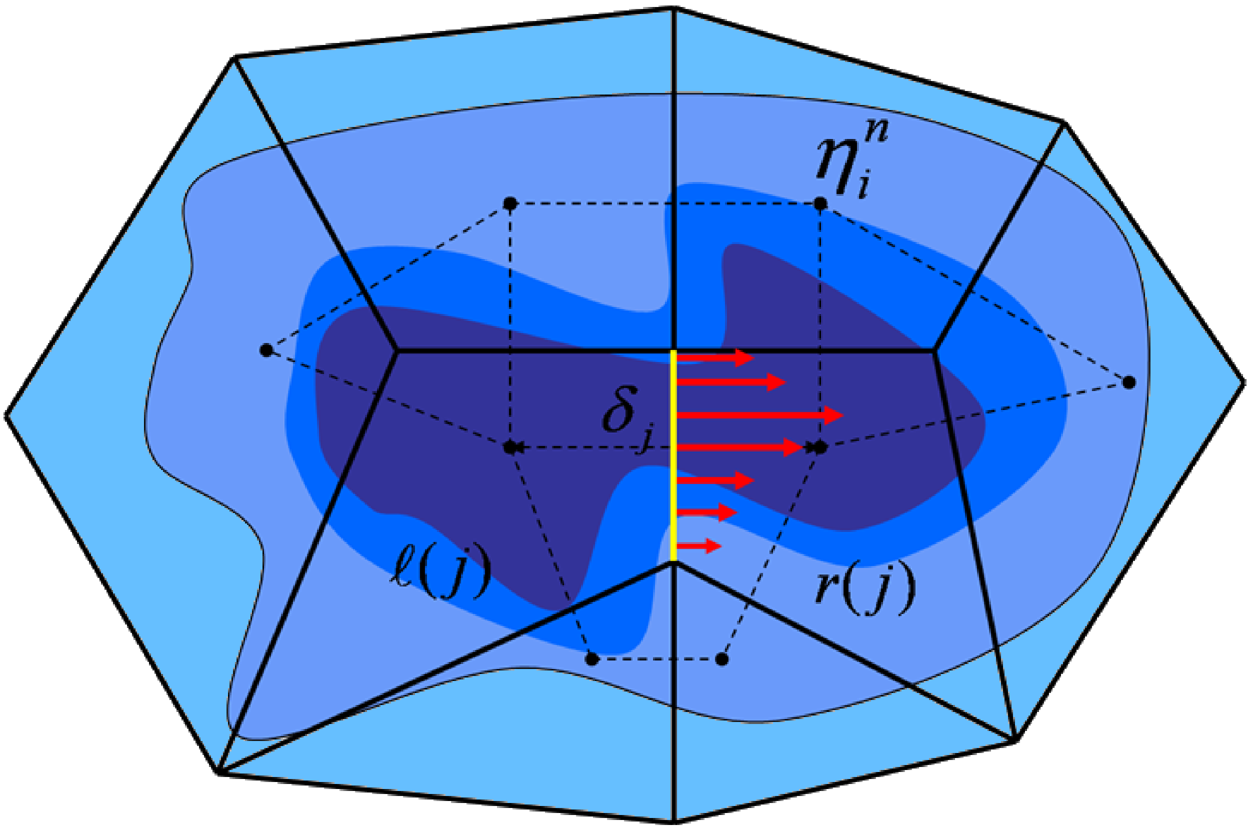


Figure 1. Unstructured computational mesh with detailed sub-grid terrain data.

Shown in Figure 1, is an example computational mesh over terrain data depicted with blue-shaded contour data. The computational cells are represented by the thick black lines. The cell computational centers are represented by the black nodes 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 2. The example shown in Figure 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 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.

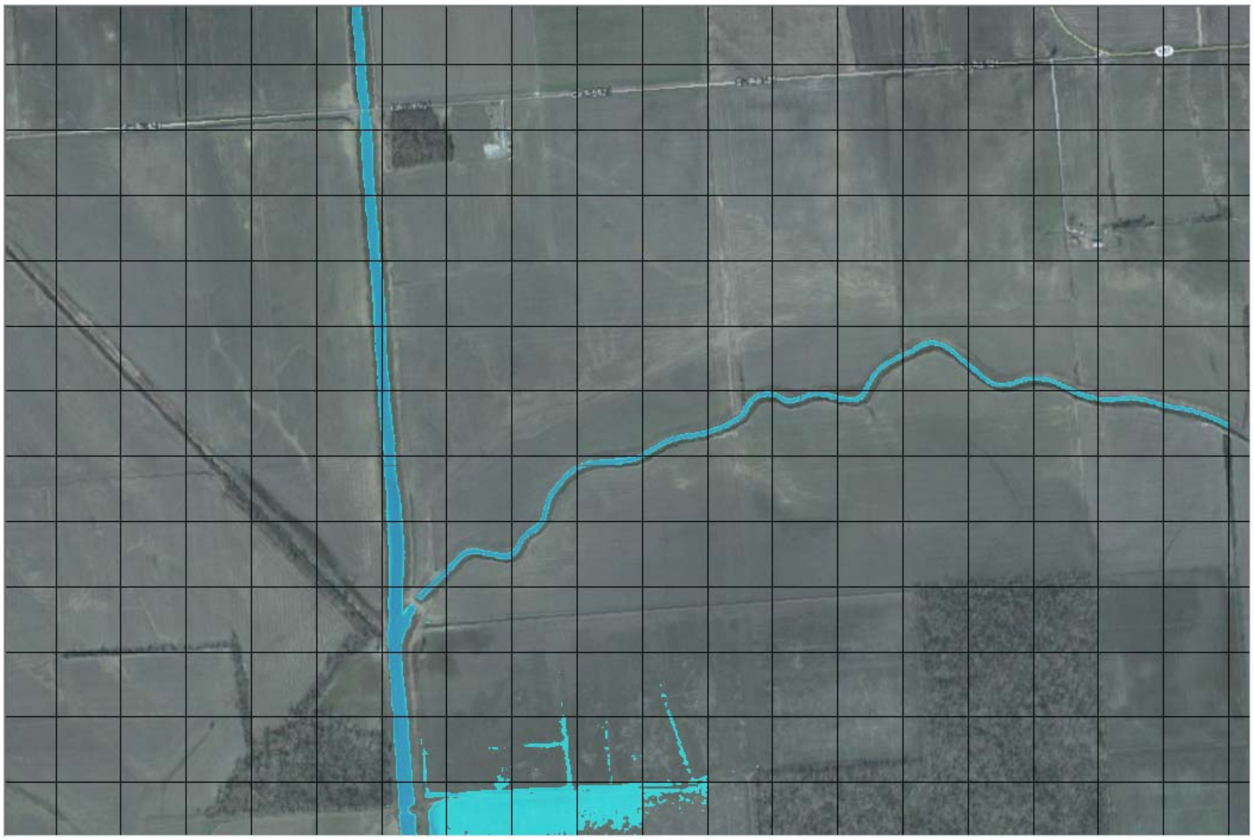


Figure 2. Example showing the benefits of using the detailed sub terrain for the cell and face hydraulic properties.

7. **Detailed Flood Mapping and Flood Animations.** Mapping of the inundated area, as well as animations of the flooding can be done right inside of 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 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.** The 2D Flow Area computational solution has been programmed to take advantage of multi-processors on a computer (referred to as parallelization). This solution algorithm runs faster than non parallelized code. Computers that have more processors will be able to perform 2D flow modeling faster than single processor computers.

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.

B. Current limitations of the 2D modeling capabilities in HEC-RAS

The following is a list of the current limitations of the HEC-RAS two dimensional flow modeling software. All of these limitations will be addressed before a final public release of the software.

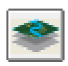
I. Each 2D Flow Area can only have one Manning's n-value to represent the terrain surface. In the first official release (Non Beta version) horizontal variations of Manning's n-values will be available by setting up Manning's n-value polygons, which will be intersected with the computational mesh in order to compute the roughness coefficients of the computational cells/faces. Future versions will also allow for vertical variation of roughness with depth.

II. Not enough automated tools for generating a detailed 2D computational mesh. While there are currently tools in HEC-RAS that allow the user to edit/modify the 2D computational mesh, in order to make it more or less detailed in specific areas, there are many tools that still need to be added to HEC-RAS to automate this process.

II. Developing a Terrain Model for use in 2D Modeling and Results Mapping/Visualization.

It is necessary to create a terrain model in RAS Mapper before you can perform any model computations that contain a 2D Flow Area, or before the user can visualize any 1D, 2D, or combine 1D/2D model 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.

A. 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 main HEC-RAS main window, then selecting **RAS Mapper**, or by pressing the RAS Mapper button  on the main HEC-RAS window. When this is done, the window shown in Figure 3 will appear.

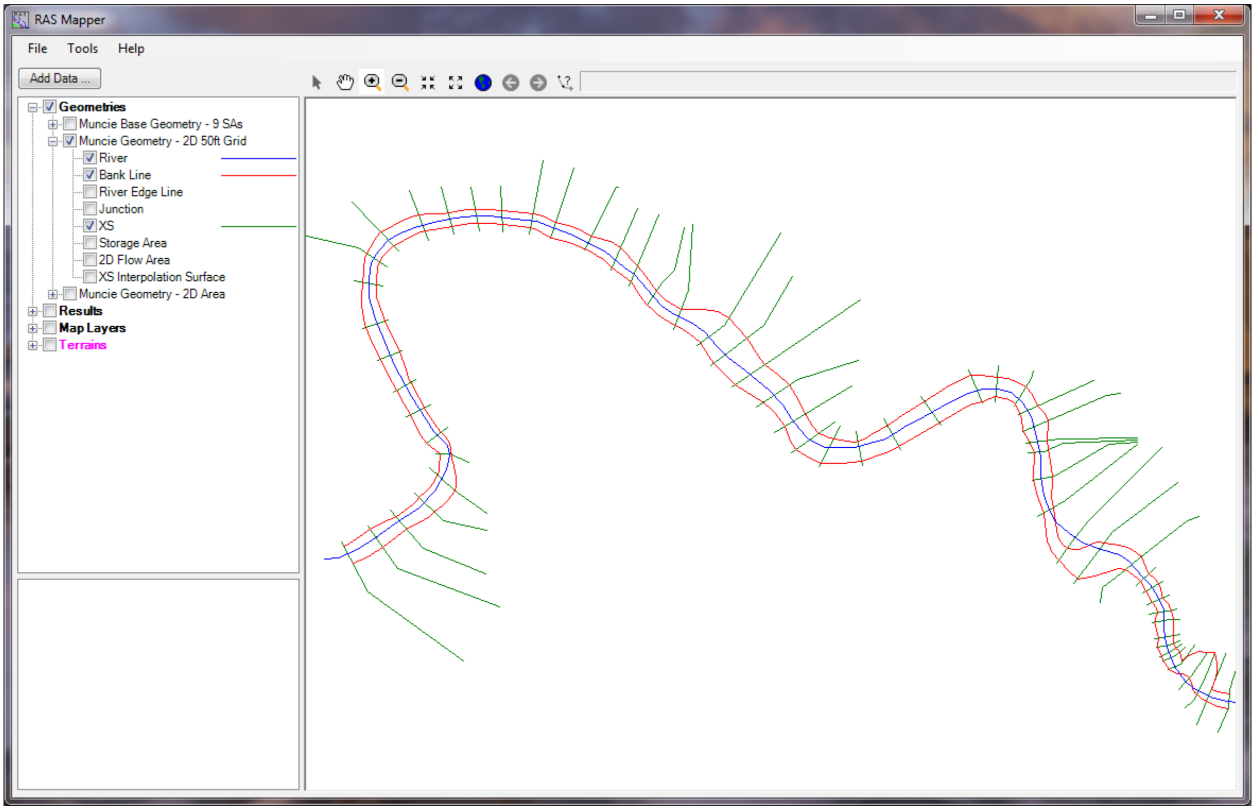


Figure 3. RAS Mapper with no terrain or other map layers loaded.

B. 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 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 4).

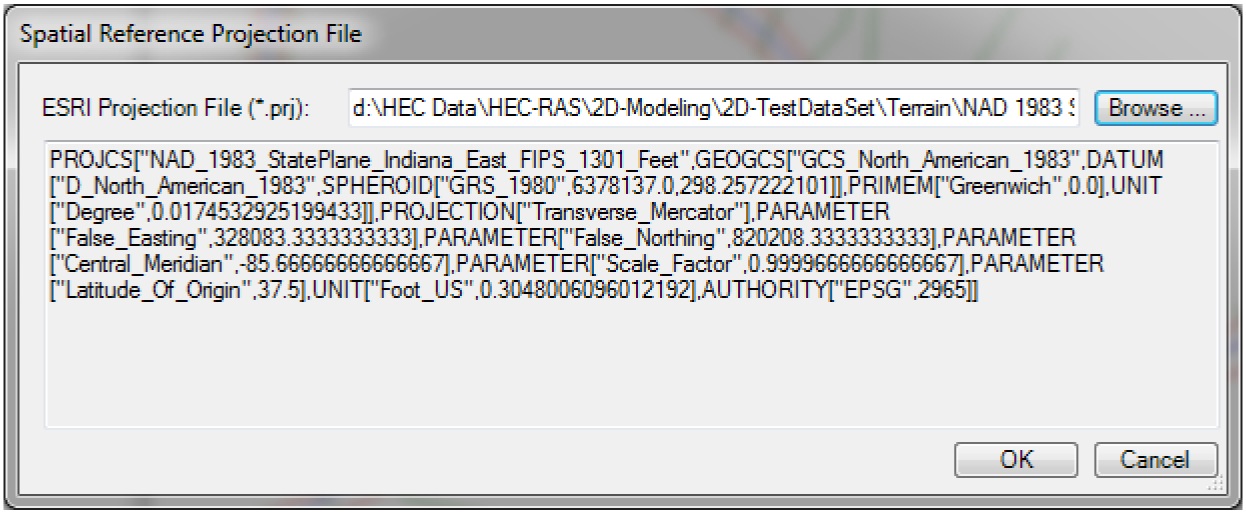


Figure 4. Editor to set the RAS project's spatial reference system.

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 (ArcGIS version 10.0. Otherwise, find an ArcGIS projection file 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.

C. 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 5). This dialog allows the user to provide a name for the new Terrain Layer (**Terrain Name** field, the default name is "Terrain"); select a directory for storing the terrain (**Terrain Folder** field); 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 (**Add Files** button). At this time, RAS Mapper can ingest terrain data that is in the floating point grid format (FLT - they have a .flt file extension), GeoTiff (.tif) format, esri grid files (*.adf file extensions), and several other formats. Use the **Add** Files button to get a file chooser, then select the floating point 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.

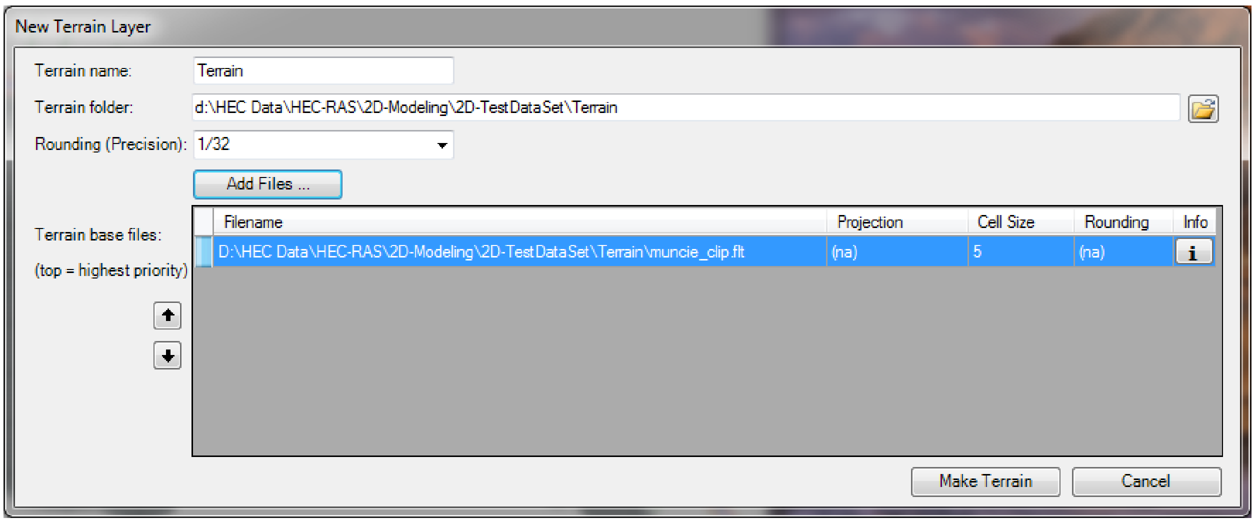


Figure 5. Example New Terrain Layer dialog.

If more than one grid file is loaded, use the 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 some others, you 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 most studies, leave the name as "Terrain", otherwise give it the name of your choice.

Once the grid files are selected, and placed in the appropriate priority order, press the **Make Terrain** button to create the new Terrain Layer. Once the **Make Terrain** 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 removing 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 you zoom in, pan, or animate 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 6.

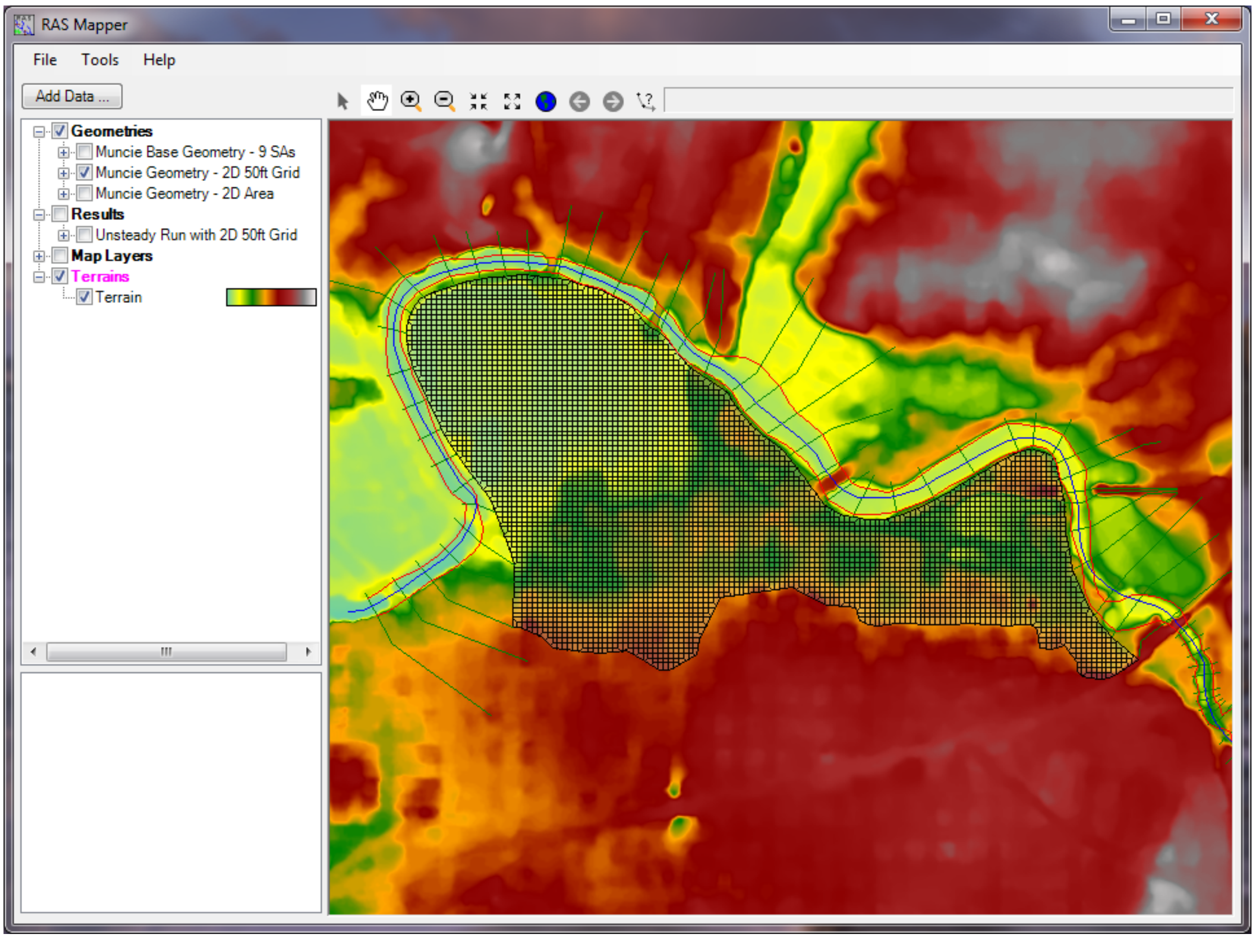


Figure 6. RAS Mapper with a Terrain Data Layer added.

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 7) allows you to: select and control the Surface Color Style; Create and plot Contour

Lines; and shade the terrain using a Hill Shading algorithm (This option makes the visual of the terrain much more realistic and semi 3D).

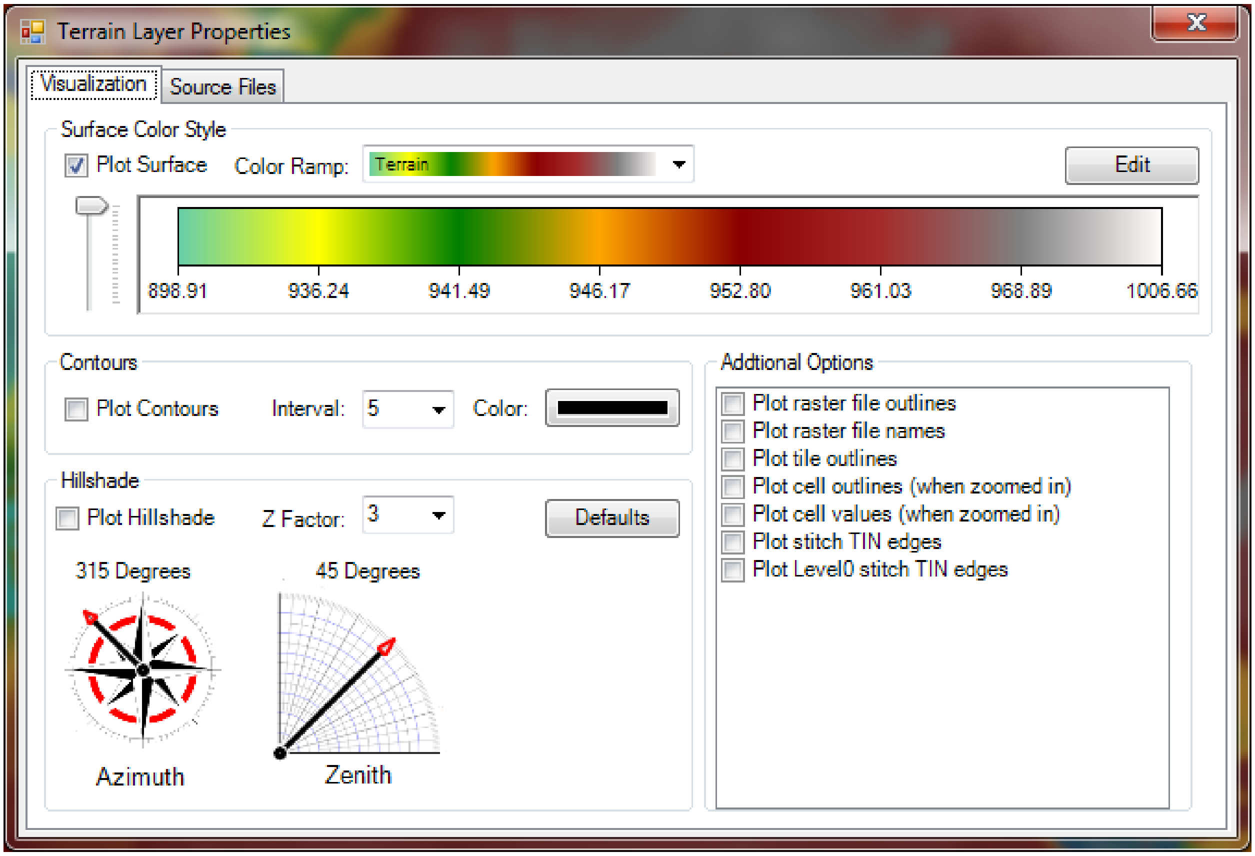


Figure 7. Layer Properties Window for the Terrain Data Layer.

An example of the Muncie Terrain data with some of the layer properties enhancements (Hill Shading on Contour Lines) turned on is shown in Figure 8.

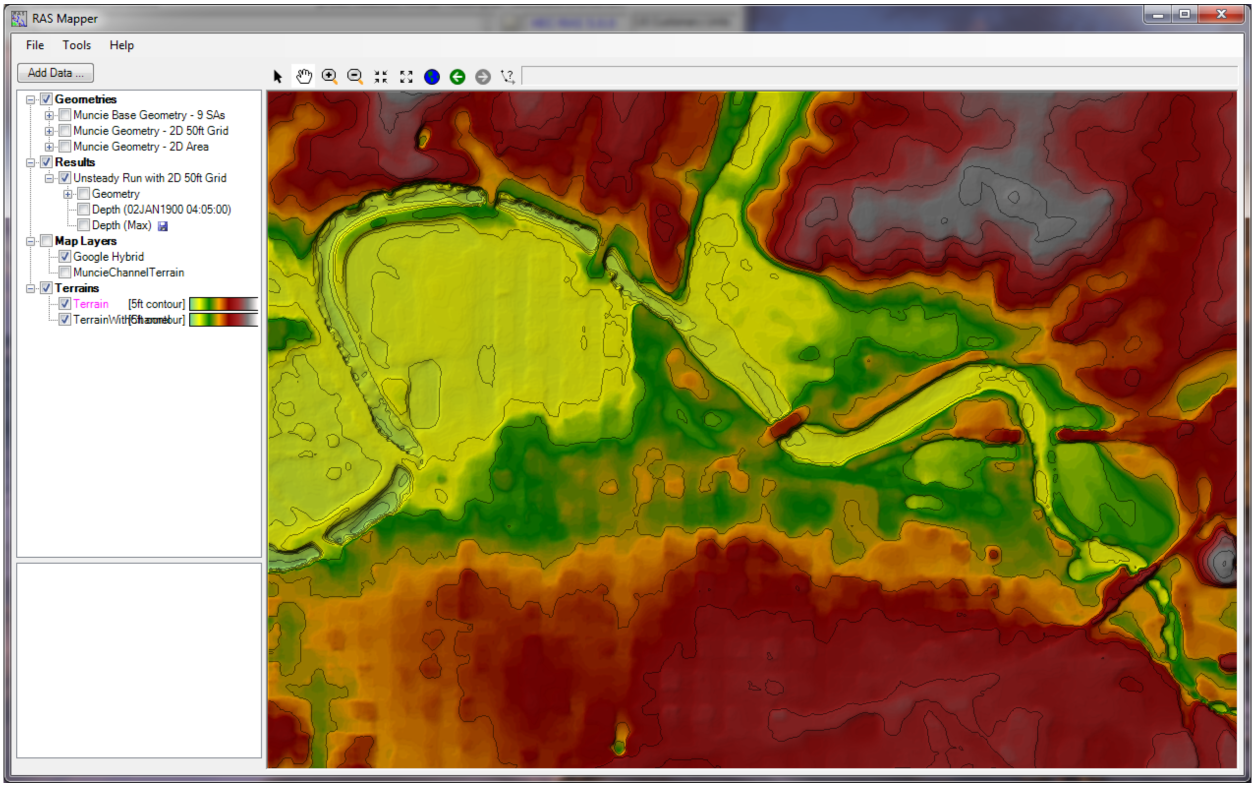


Figure 8. Terrain Data with Hill Shading and Contour Lines Turned On.

D. 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:

1. Create 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; 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 you want for a new channel terrain model (This is the information that is used to create the channel model). See an example in Figure 9 below:

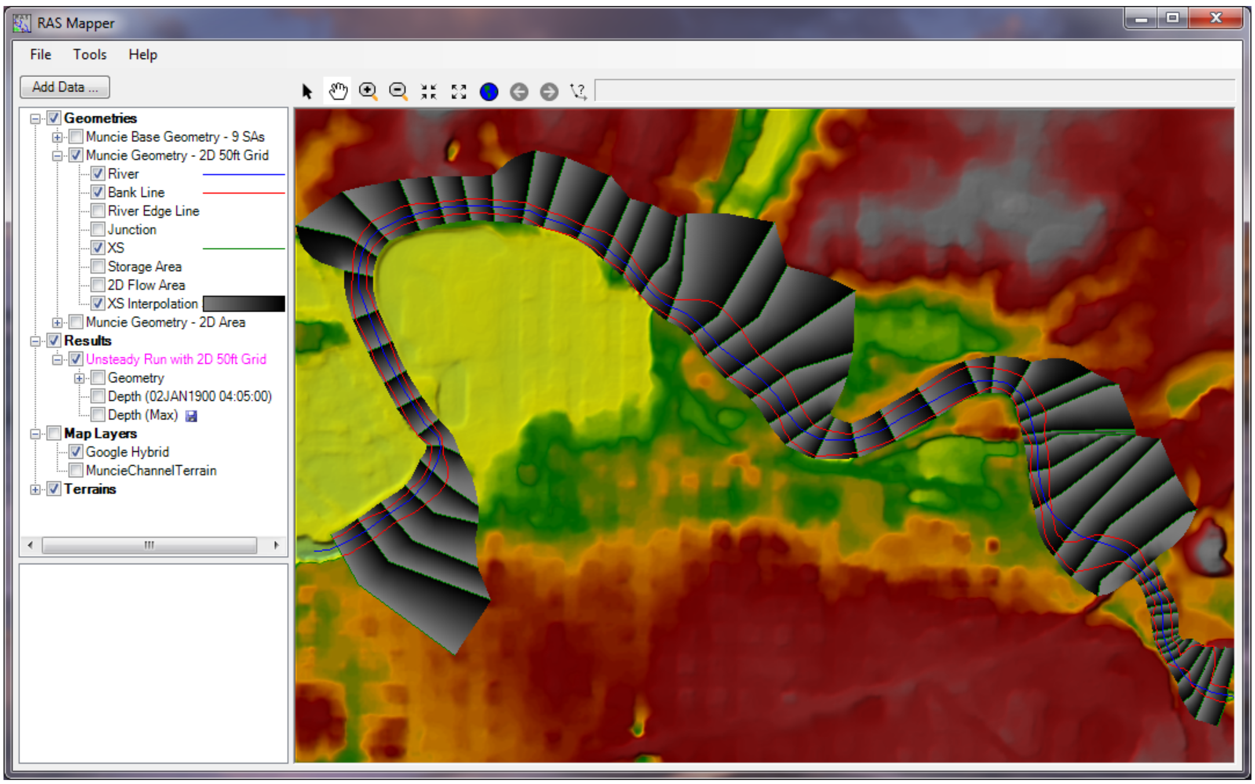


Figure 9. RAS Mapper with base terrain and Geometry Layers Displayed.

If all the geometry layers look good, then creating the channel terrain model is accomplished 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 you base terrain model has good overbank terrain information, you 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 Figure 10 below:

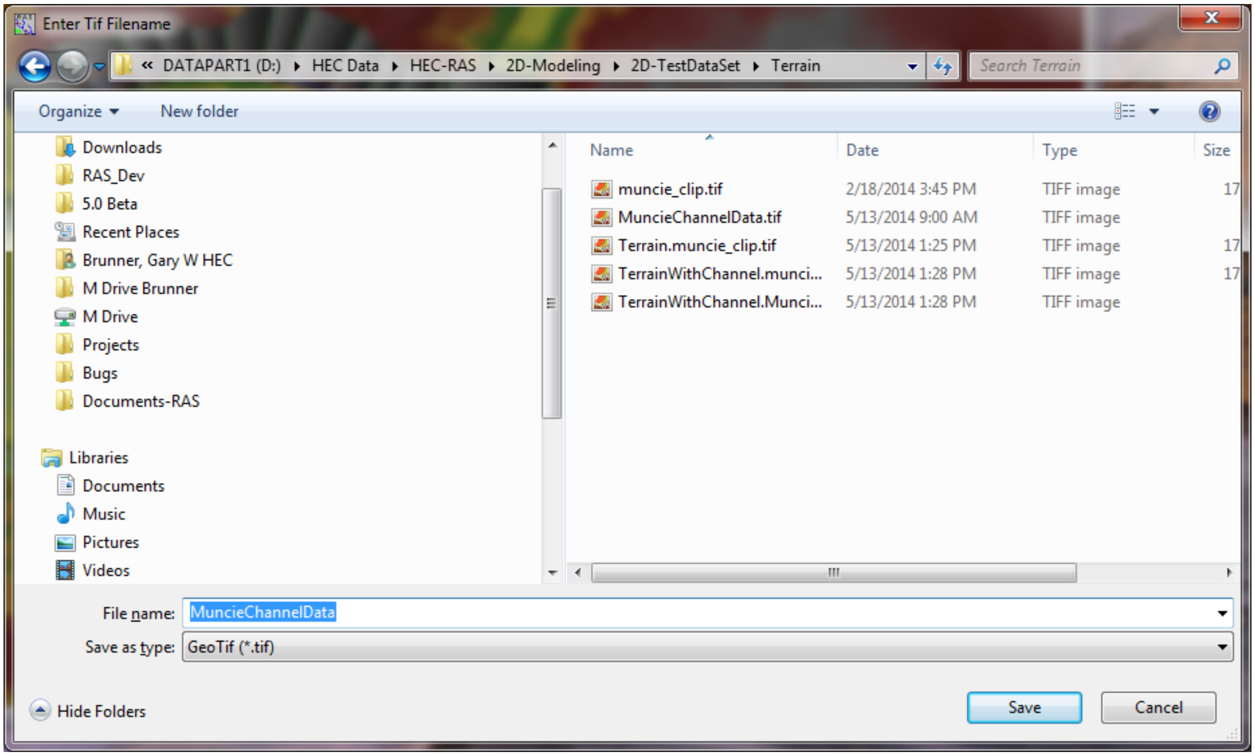


Figure 10. Terrain Export File Choose shown with Example Name and Directory.

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 you enter "5.0", then the new terrain model will of grids that are 5 ft by 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.

2. Making a Combine channel and Overbank terrain model

Once you have made a terrain model from the channel data, you can now make a new combine terrain model from the base terrain model (the terrain with the overbank/floodplain data) and your newly created channel only terrain model. To make the new combine 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 5). 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 (however, the precision should not be finer than the terrain files used to create this new terrain model). Then select the **Add Files** button and select the base terrain models Geotiff file, and the cross section only terrain models Geotiff file. Make sure that the new channel 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 **Make Terrain** 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 11 below:

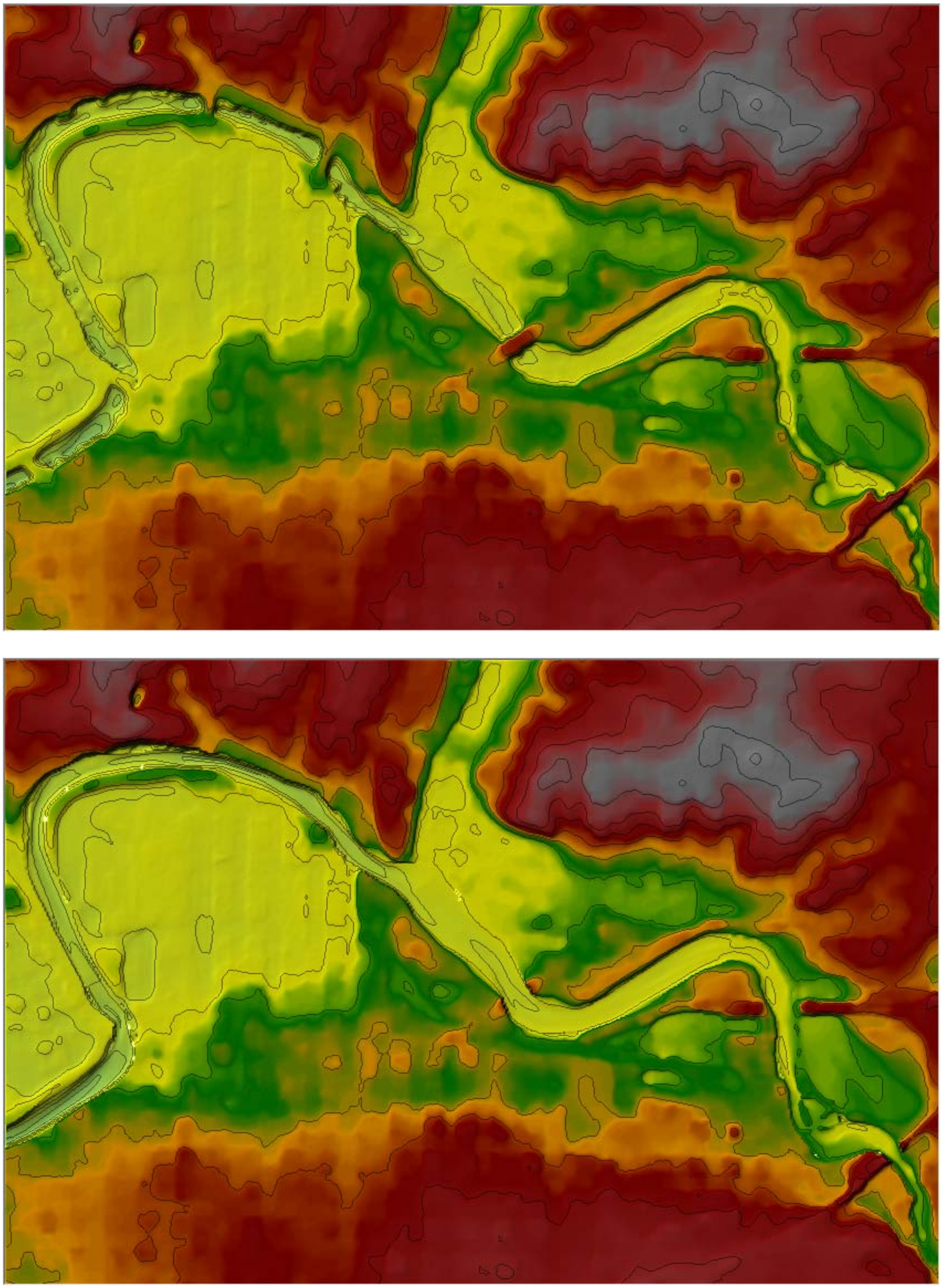


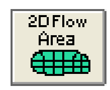
Figure 11. Original Terrain model (Top) and New Terrain model (Bottom) with Channel Data.

III. Development of a Combined 1D/2D Geometric Data Model

A. 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 the mesh editing tools. A 2D computational mesh is developed in HEC-RAS by doing the following:

1. Draw 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 you would create a Storage Area). The best way to do this in HEC-RAS is to first bring in a background image of either the underlying terrain, and/or an aerial image. Additionally, you may want to bring in a shapefile that represents the protected area, if you 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.

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 12). 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 you run out of screen real-estate, right-click to re-center the screen. Double-click the left mouse button to finish creating the polygon. Once you have finished drawing the 2D area polygon by double clicking, the interface will ask you for a Name to identify the 2D Flow Area. Shown in Figure 12 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".

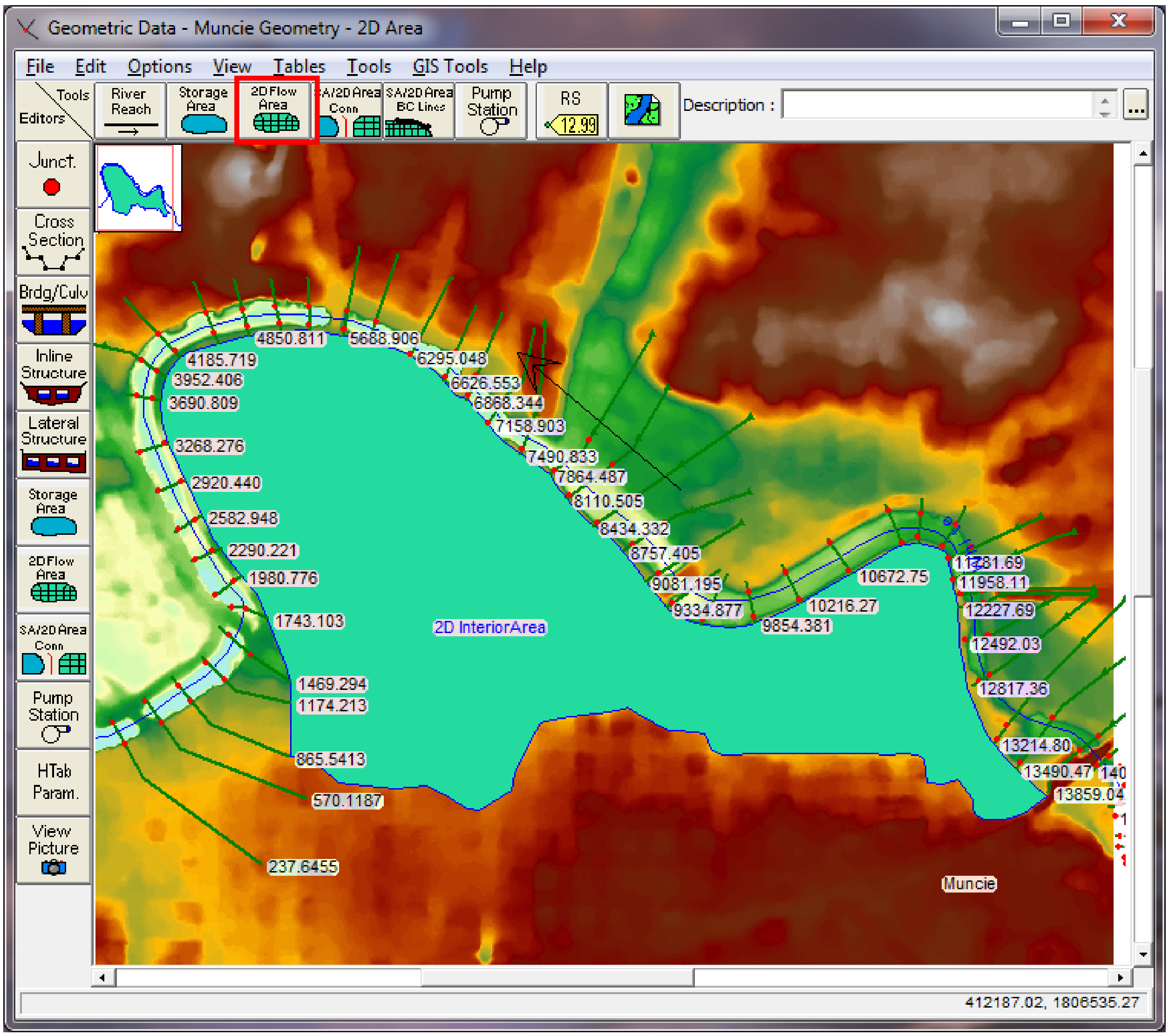


Figure 12. Example 2D Flow Area polygon.

2. Creating the 2D 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:

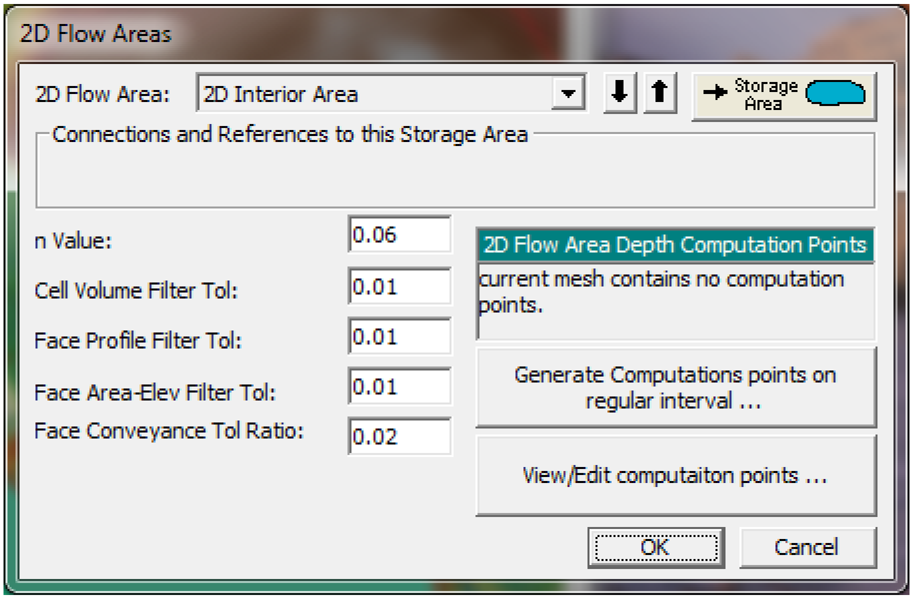


Figure 13. 2D Flow Area Mesh Generation Editor.

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. The user must also enter a Manning's n-value for the 2D Flow Area (right now, HEC-RAS is limited to a single n value for the 2D Flow Area, however, this will be changed for the first official release). To use this editor, first select the button labeled "**Generate Computational points on regular Interval …**". This will open a popup window that will allow you to enter a nominal grid size. The editor requires you to enter a **Computational Point Spacing** in terms of DX and DY (See Figure 14). The points it is referring to are the computational grid-cell centers. For example, if you enter DX = 50, and DY = 50, you will get a computational mesh that has grids that are 50 X 50 everywhere, except around the outer boundary. Since the user can enter an irregular boundary for the extents of the 2D Flow Area, the mesh generation tools will automatically generate cells around the boundary that are close to the area of the nominal grid-cell size you selected, but they will be irregular in shape. The popup editor has an option to enter where you 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 station offset (i.e. starting Point Offset), allows you to shift the origin of the grid, and therefore the location of the points.

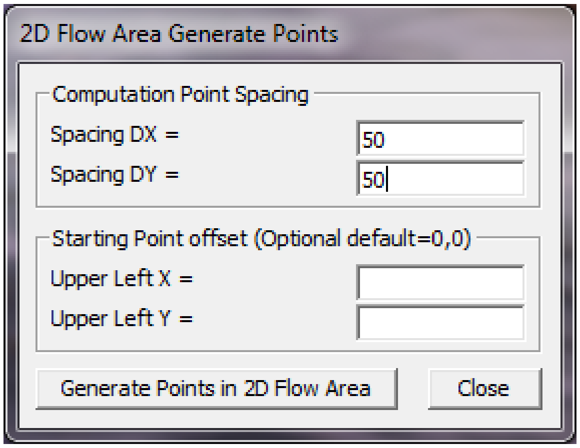


Figure 14. 2D Computational Point Spacing Editor.

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** option, which brings the points up in a table. The user can cut and paste these into a spreadsheet, or edit them directly if they desire (it is not envisioned that anyone will edit the points in this table or Excel, but the option is available).

There are four additional fields on the 2D Flow Areas editor (Figure 13) that are used during the 2D pre-processing. These fields are:

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, and a Manning's n-value has been entered, 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 15).

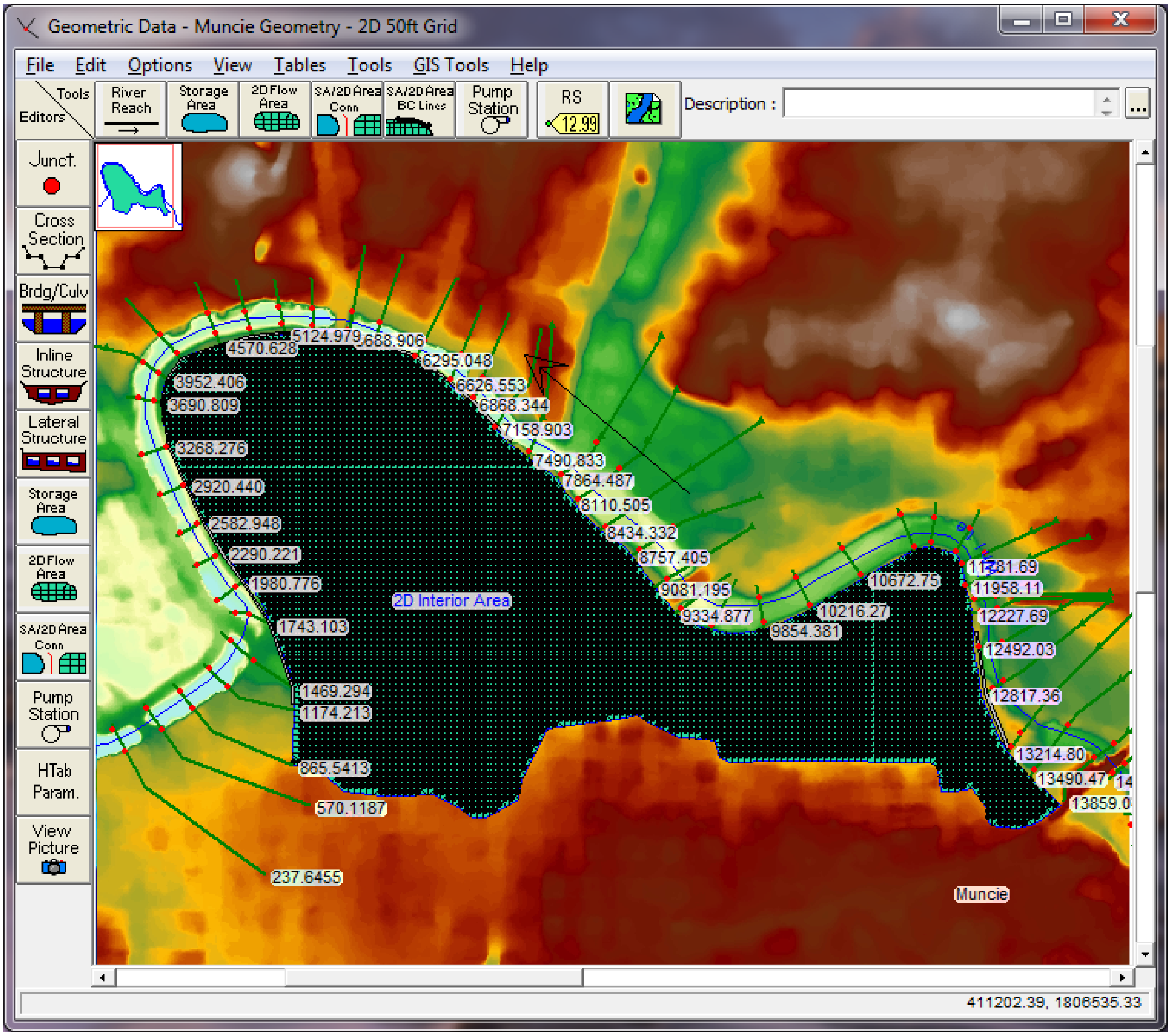


Figure 15. Example 2D computational mesh for an interior of a levee protected area.

As mentioned previously, cells around the 2D Flow Area boundary will be irregular in shape, in order to conform to the user entered 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 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. The result of combining cells along the boundary is illustrated by the zoomed in view of the 2D Flow Area computational mesh in Figure 16.

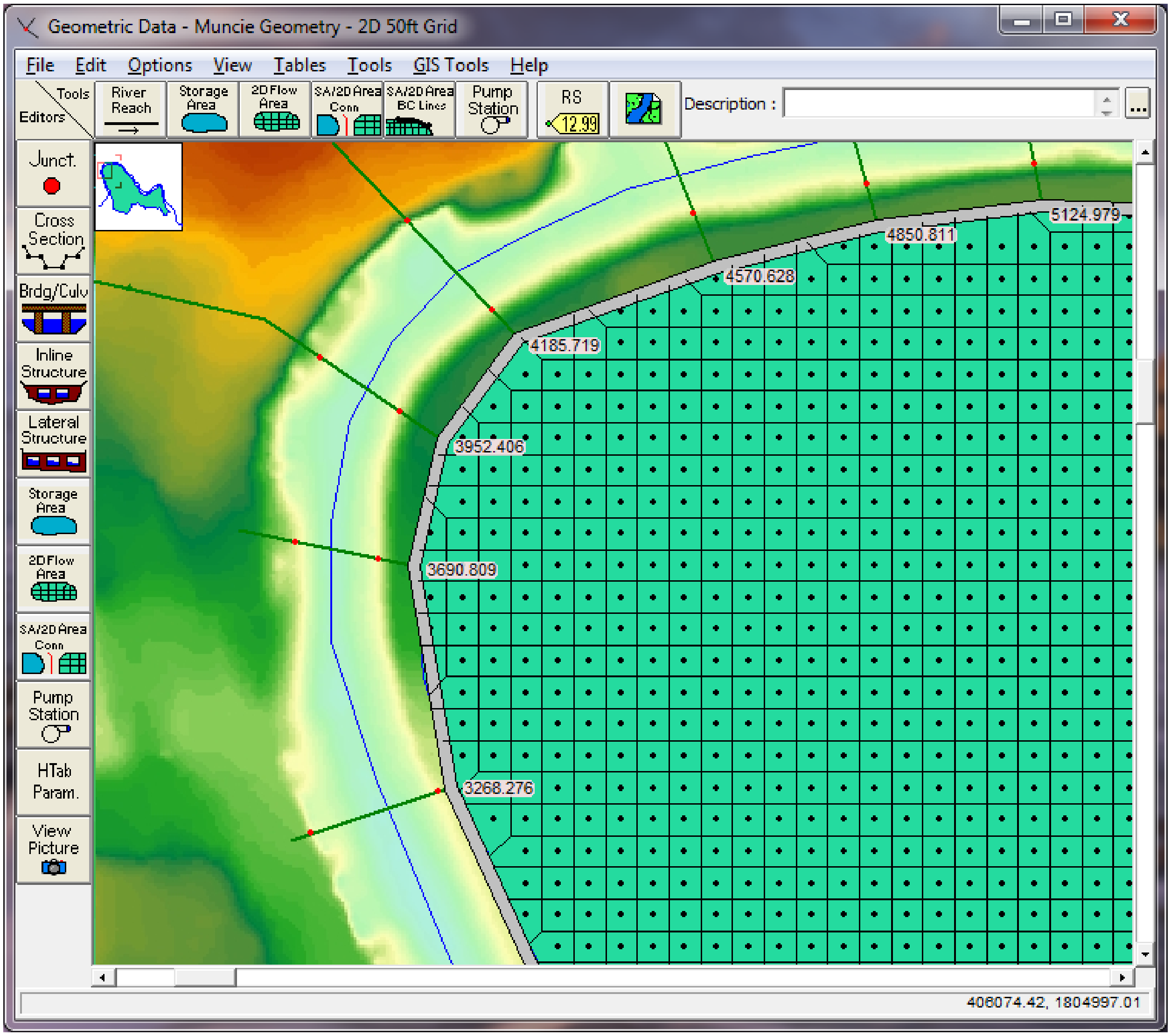


Figure 16. Zoomed in view of the 2D Flow Area computational mesh.

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 17):

Cell Center: The computational center of the cell. This is where the water surface elevation is computed for the cell.

Cell Faces: These are the cell boundary faces. Faces are generally straight lines, except along the outer boundary of the 2D Flow Area, in which case a cell face can be a multi-point line.

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 Lateral Structure.

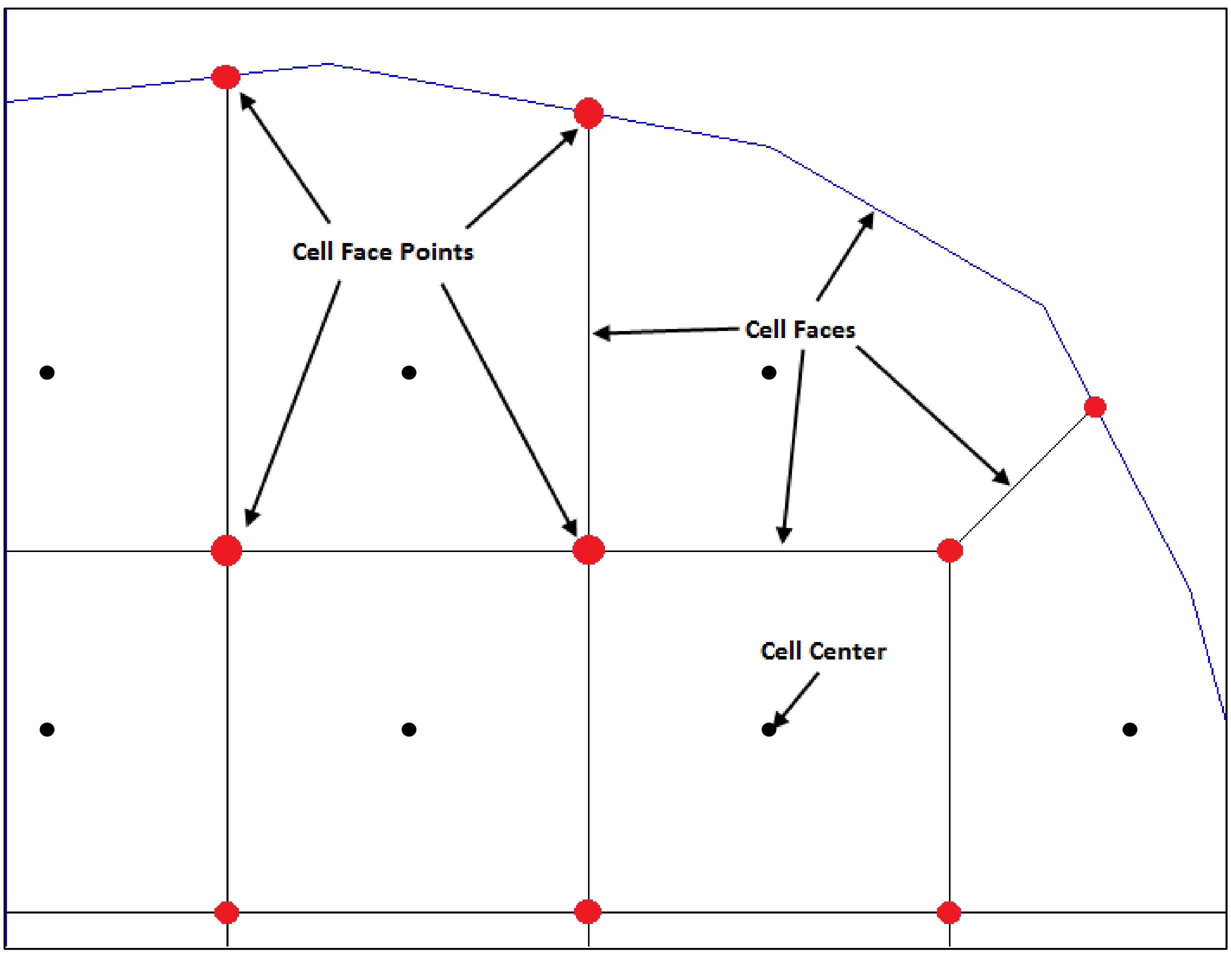


Figure 17. Description of HEC-RAS 2D modeling computational mesh terminology.

3. Edit/Modify 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 pre-processed 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, regardless of the computational cell size. However, there are still limits to what cell size you should use, and important considerations for where you should have smaller detailed cells verses large 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 are required to define significant changes to geometry and barriers to flow such as high ground, roads, smaller interior levees, etc…

In the current version of HEC-RAS, the mesh Editing/Modifying tools are limited to three options. They are: moving points; adding points, and deleting points. All of these mesh manipulation tools are available under the **Edit** menu of the HEC-RAS Geometric Editor. If the user selects "**Edit**" then "**Move Points/Object**", you can grab 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 you left-click within the 2D Flow Area, a new cell center will be added, and the neighboring cells will be changed (Once the Mesh is updated). The mesh only updates once you have turned off the editing feature. This is done on purpose in order to save computational time for large meshes. If the user selects "**Edit**" then "**Delete 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 (once the mesh is regenerated). HEC-RAS makes the computational mesh by following the Delaunay Triangulation technique, and then constructing a Voronoi diagram (See Figure 18 below, taken from the [Wikimedia Commons](), a freely licensed media file repository):

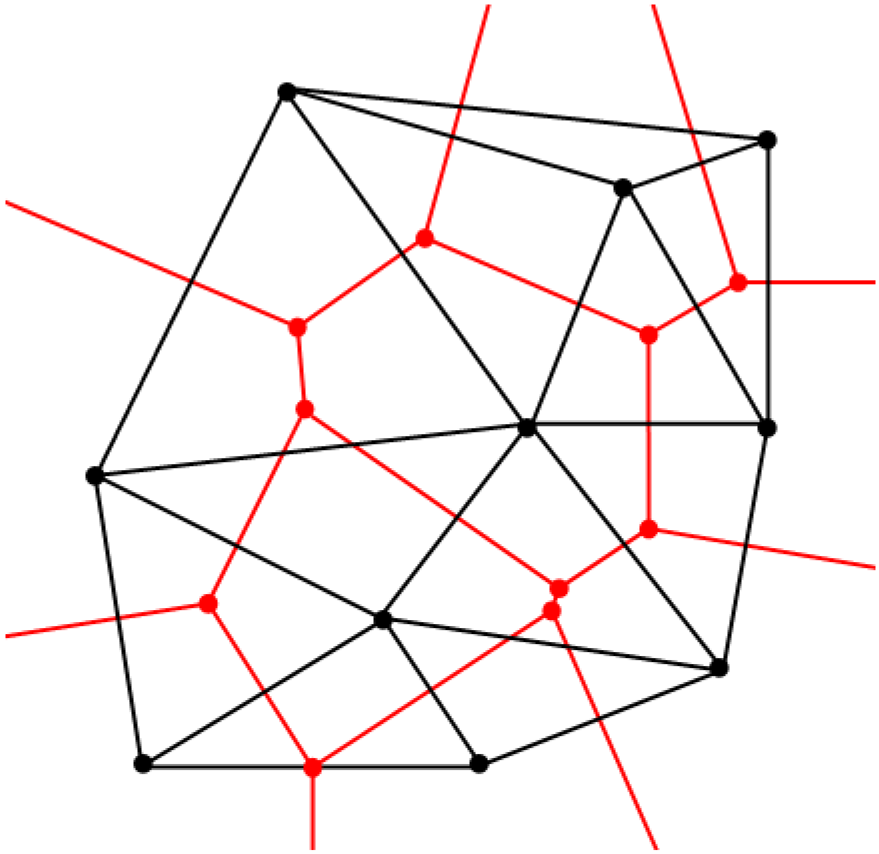


Figure 18. Delaunay - Voronoi diagram example.

The triangles (black) shown in Figure 18 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.

You may want to add points and move points in areas where you need more detail. You may also want to remove points in areas where you know you need less detail. Because cells and cell faces are pre-processed into detailed hydraulic property tables, they represent the full details of the underlying terrain. In general, you should be able to get away with larger grid cell sizes than what you would be able to with a model that does not do this pre-processing of 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.

Right now the Computational mesh generation tools in HEC-RAS are few, and somewhat limiting. There are many enhancements that could be made to the mesh generation tools. For example, we plan to have a tool in which the user can draw an internal line along linear features that controls flow movement, such as the high ground of a flow barrier. After this line in drawn, the mesh generation tools will automatically match the cell faces along that line, such that those cell faces pick up the high ground elevations of that terrain barrier, thus making a much more detailed model in that area, but without having to have very small cells. We also have ideas for other automated mesh generation tools that would be based on making contours of the terrain first, then using those contours to automatically generate the computational cells. Any ideas for improving the mesh generation tools should be sent to the HEC-RAS team.

4. Potential Mesh Generation Problems

The Automated mesh generation tool in HEC-RAS works extremely well, however, nothing is perfect. On occasion a bad cell will be created due to the combination of the user define polygon boundary and the selected nominal cell size. Here is a list of some problems that are possible, and how to fix them with the mesh editing tools described above:

- **More than One Cell Center in a Single Cell**: Sometimes on the outer boundary cells, the automatic mesh generator will create a cell with more than 1 cell center point inside the cell (see Figure 19 below). Computationally this is not allowed. To fix it, go to the Geometric Editor "Edit" menu and select the "Remove Points" Option. Then just click on the points you want to remove in the cell that has more than one cell center.

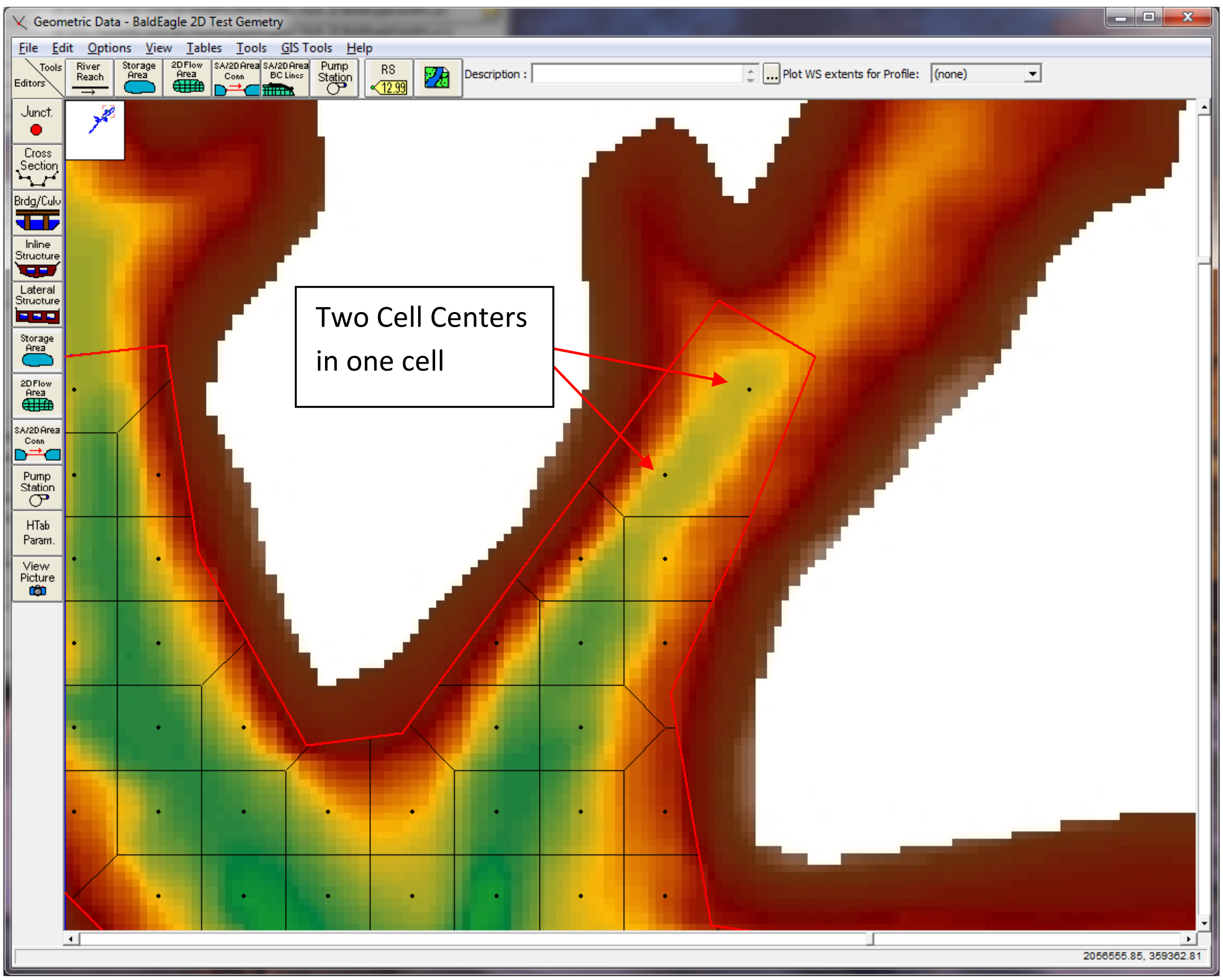


Figure 19. Example of a cell with two cell centers.

- **Cell has no Cell Center:** Every computational cell must have one and one only cell center. On occasion the automatic mesh generator may create a cell with no cell center (Figure 20). To fix this move the 2D area boundary points, or cell centers.

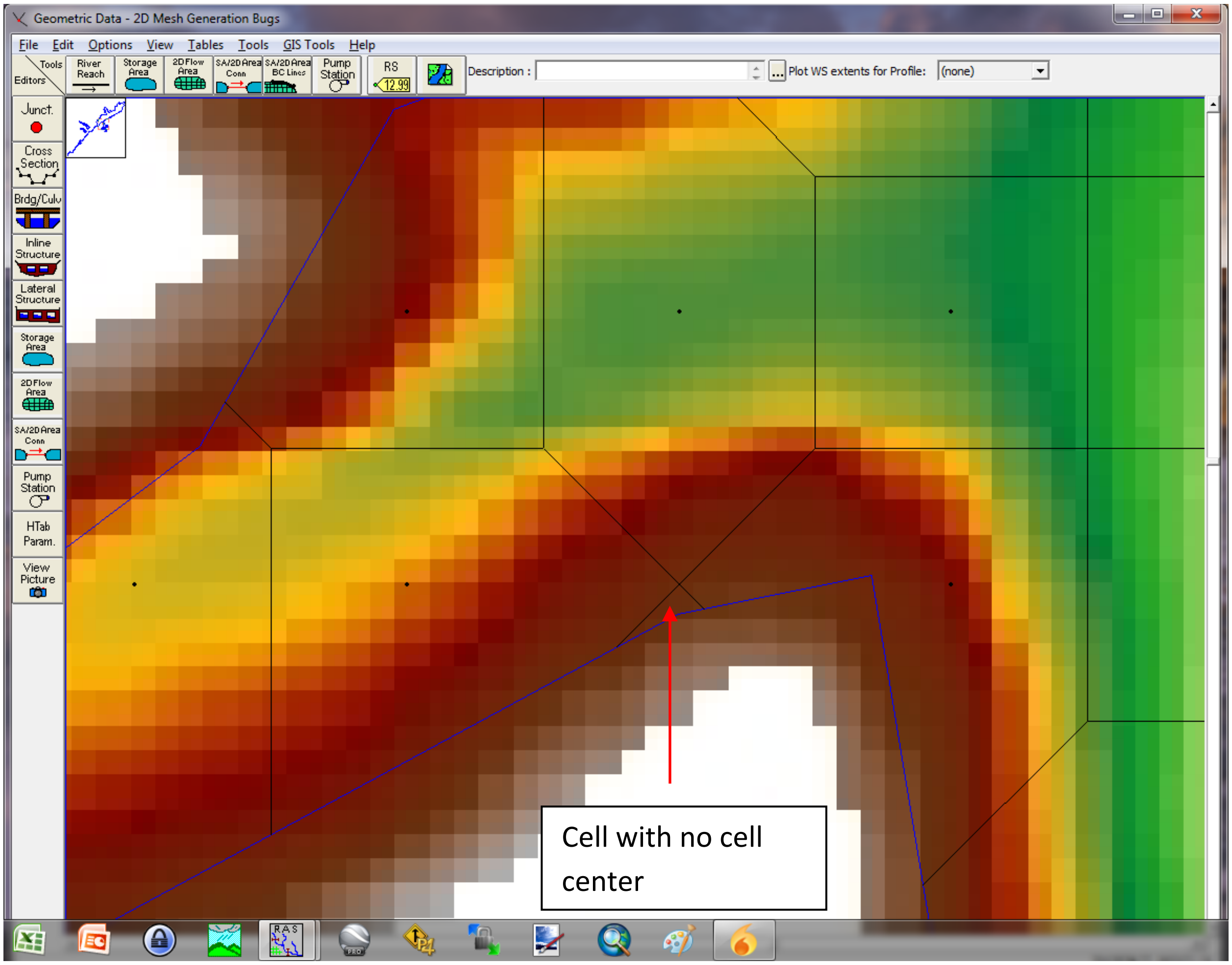


Figure 20. Example of a cell with no cell center.

- **Cell Face Crosses over into Multiple Cells:** On a rare occasion, the automated mesh generator may create a cell face that extends way past that single cell (Figure 21). This only occurs for boundary cells, and usually where the boundary has a very sharp corner. To fix this you can add points to the boundary polygon and smooth out the boundary. You can also add more cell centers, delete cells, or move some cell centers. 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.

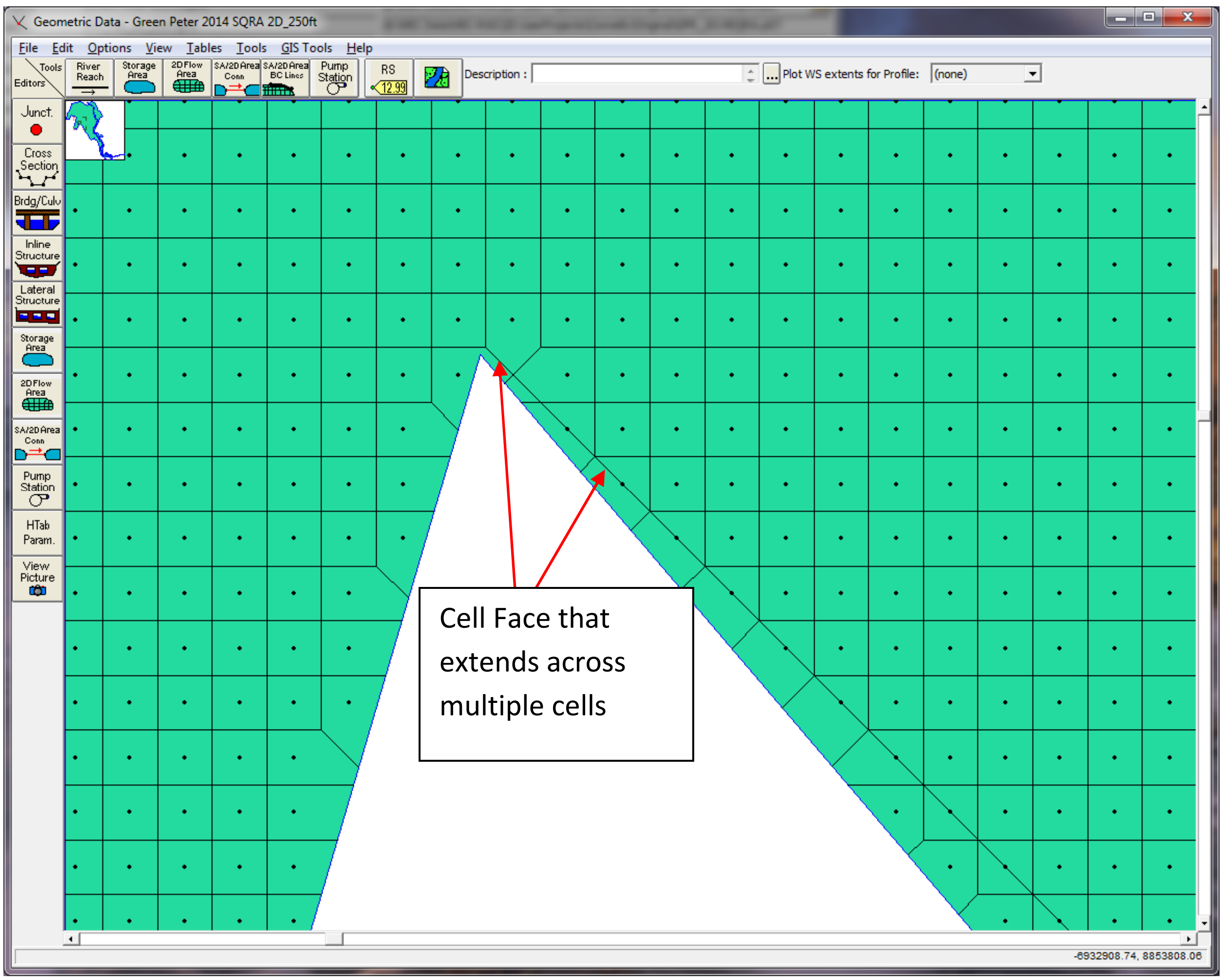


Figure 21. Example of a cell face that extends way beyond that single cell.

- **Cell has more than One Outer Boundary Face:** If you have an area that the 2D Polygon necks down, the automatic mesh generator may create a cell that has two outer boundary faces (Figure 22). Currently this is not working computationally (we will fix this and get it to work in a future version). You must break up that cell into multiple cells, such that each cell has only one outer boundary face. A boundary face can be made up of many points/sides, but it must be continuous.

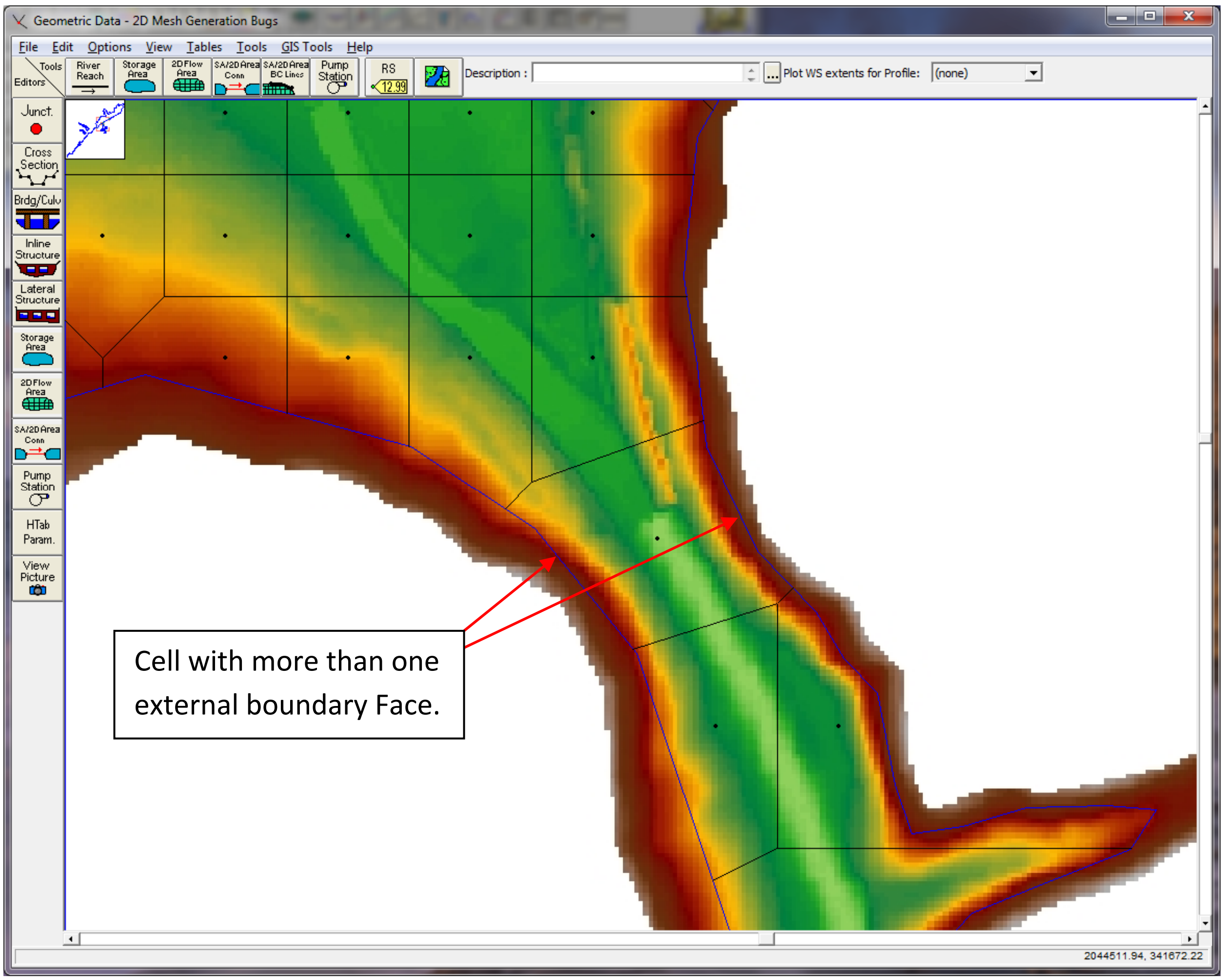


Figure 22. Example of a Cell with two Outer Boundary Faces.

- **Too Many Faces (sides) on a Cell:** Each cell is limited to having 8 Faces (sides). 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.

B. 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). These relationships are derived from the details of the underlying terrain used for the model. So 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 Section II 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 it is good enough for performing a hydraulic model study, 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:

1. 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 any of the Geometry files listed in the top layer list (on the left hand side of the RAS Mapper window), then selecting the **Associate Terrain Layer** option from the popup menu. When this is done a window will appear, as shown in Figure 23, in which you 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.

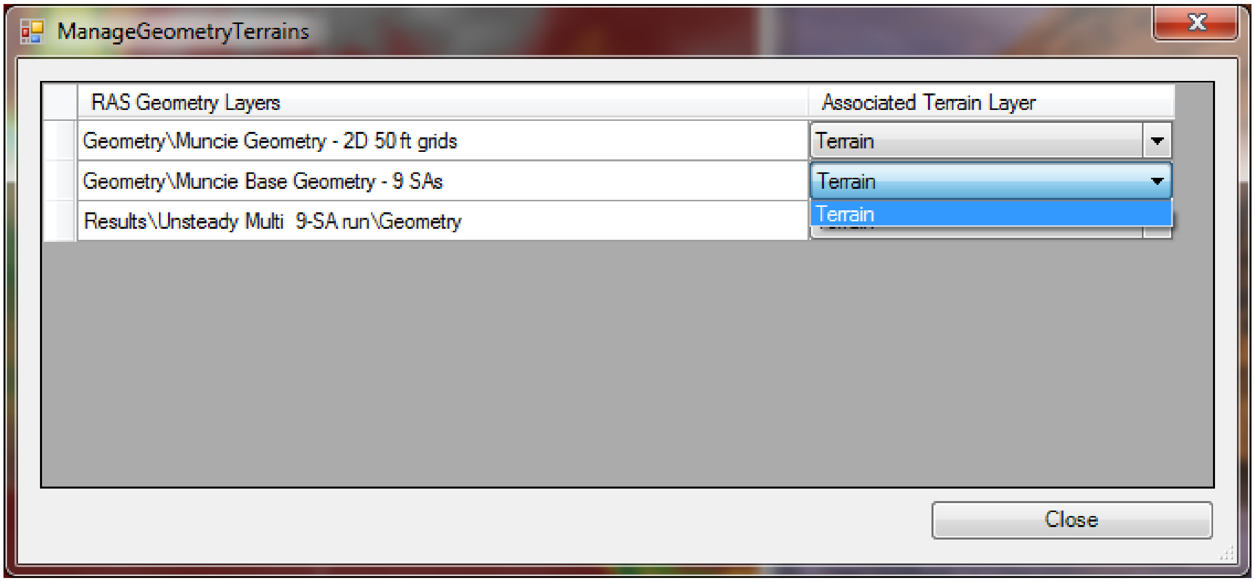


Figure 23. Terrain Association Editor.

After associating all of the geometry files with the terrain layer(s), select the RAS Mapper **File** menu and select the **Save** option. This will ensure that these associations are saved.

2. 2D Cell and Face Geometric Pre-Processor

Overview of Cell and Face Properties

Each cell, and cell face, of the computational mesh is pre-processed in order to develop detailed hydraulic property tables based on the underlying terrain used in the modeling process. The 2D mesh pre-processor computes a detailed elevation-volume relationship for each cell. Each 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 means less 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 elevation for each cell and face.

An example of how HEC-RAS pre-processes cells and faces into detailed property tables is shown in Figures 24 through 27. Shown in Figure 24 are the details of the underlying terrain within a single computational cell.

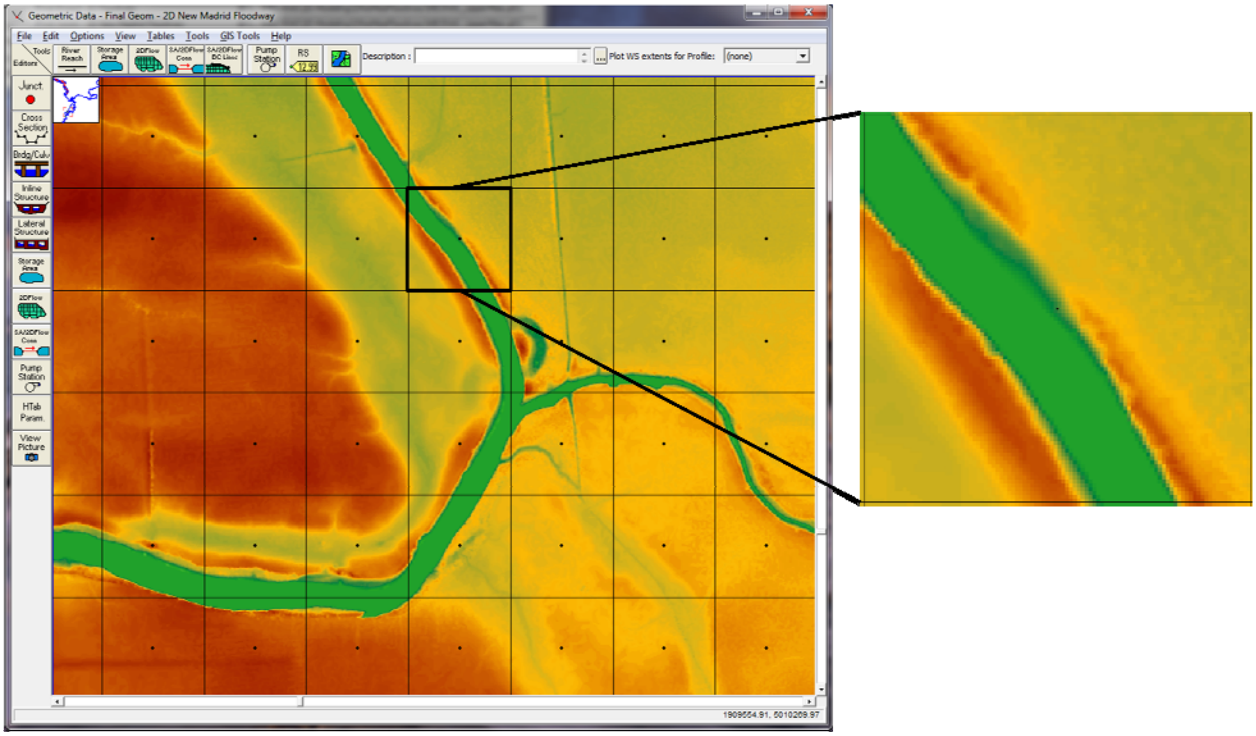


Figure 24. Details of Underlying Cell Terrain Data.

When the 2D Geometric Pre-processor runs a detailed elevation-volume relationship is developed for each cell. See an example of this in Figure 25 below.

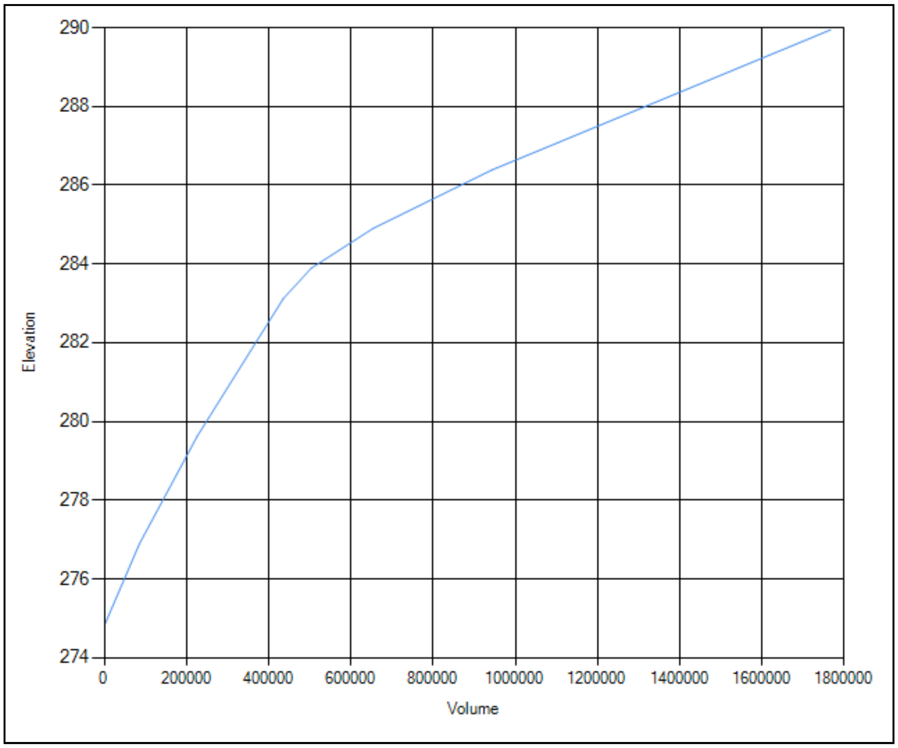


Figure 25. Elevation – Volume relationship for a 2D cell.

In addition to the processing of the cells, the faces of the cells are pre-processed into tables of elevation versus area, wetted perimeter, and roughness. See Figure 26 below:

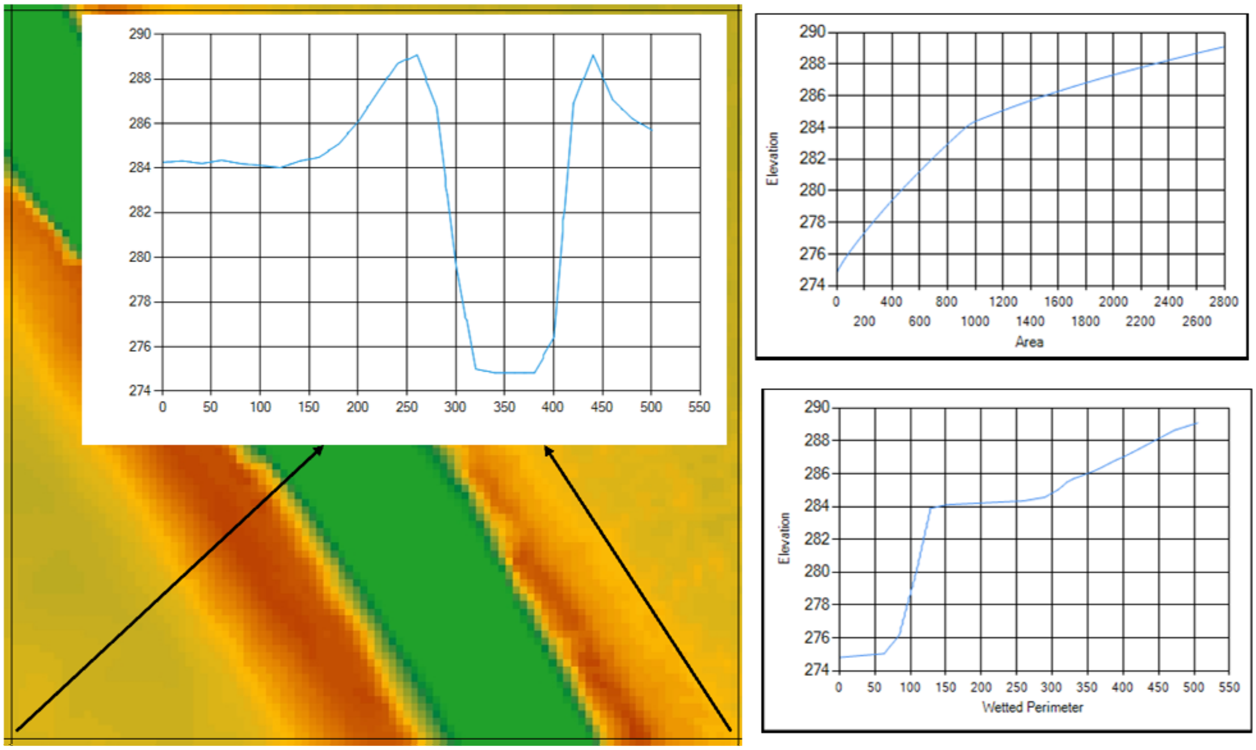


Figure 26. Example of how Cell Faces are processed into detailed hydraulic tables.

As shown in Figure 26, each 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 benefits of this are 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 27.

Figure 27. Example of detailed channel moving through larger cells in HEC-RAS

Shown in Figure 27 is an example of how the computational cells in HEC-RAS contain enough hydraulic detail that flow can move through a channel, even though the channel is smaller than the cell size. In the above example, the cells are 500 ft by 500 ft. Water will move though 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 is 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 you would need to use a cell size smaller than the channel to capture two dimensional flow effects within the channel itself. However, if you only need to capture the two-dimensional flow affects on the floodplain, then this is a very viable option.

The 2D flow capabilities in HEC-RAS can be used in many ways. You can develop a mesh with very small cell sizes that can be used to model both channels and floodplains in great detail. Or you can use larger cell sizes, which will give you less details in the channel, but still 2D flow hydraulics in the floodplain. The level of detail you use depends on what you are trying to model, and the purpose of your study. HEC-RAS provides you the maximum amount of flexibility in how you model the details of a channel and the floodplain in 2D. Pre-processing the cells and 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 Pre-Processor

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 you do not run the 2D Geometric Pre-Processor 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 28). 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 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 (add, move, delete cells, change manning's n-values, etc…), then the 2D pre-processor step will be rerun during the unsteady flow computational process.

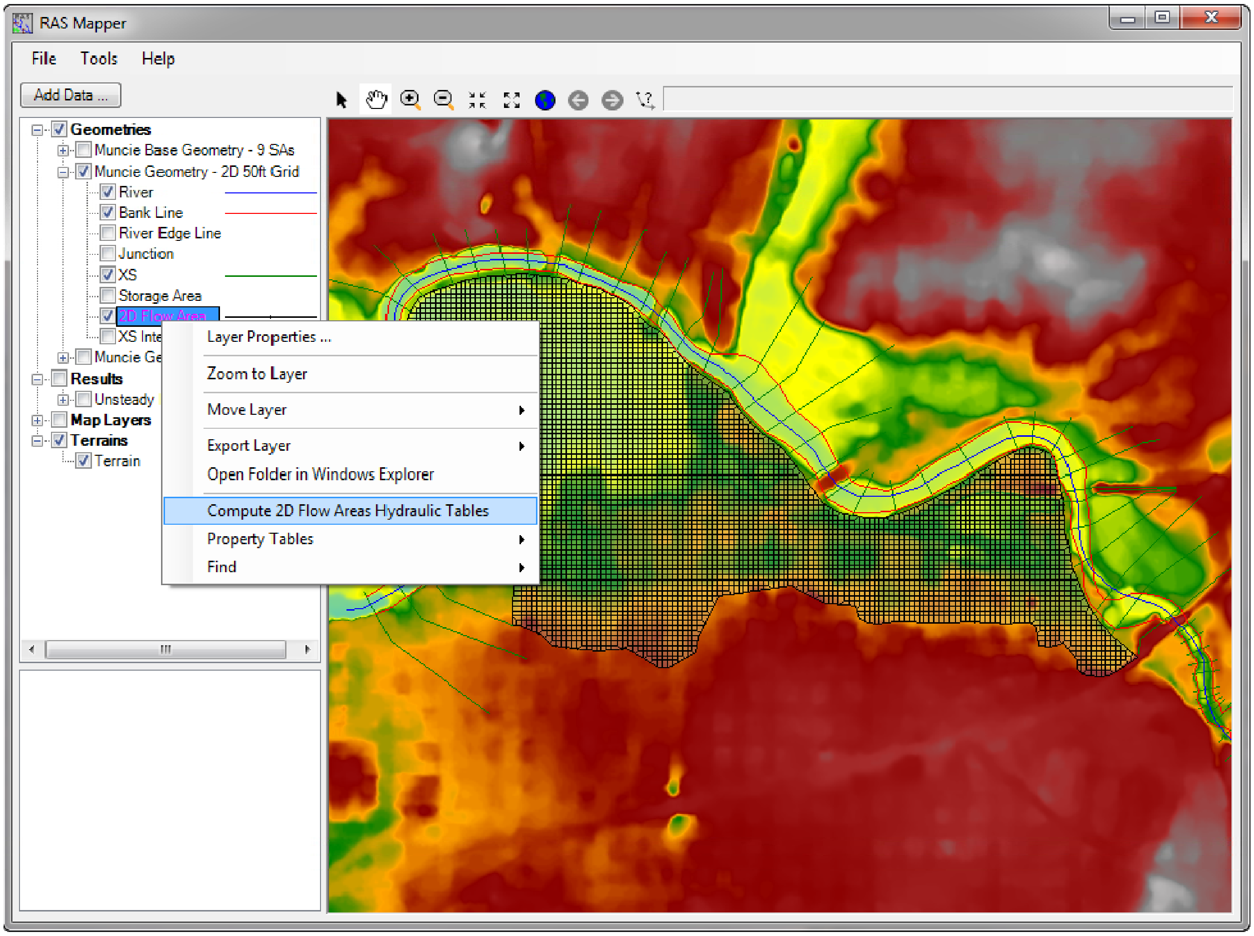


Figure 28. Computing 2D Flow Area Hydraulic Tables from RAS Mapper.

C. 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 hooking up a 2D Flow Area to other hydraulic elements is accomplished in the HEC-RAS Geometric editor.

1. 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 29).

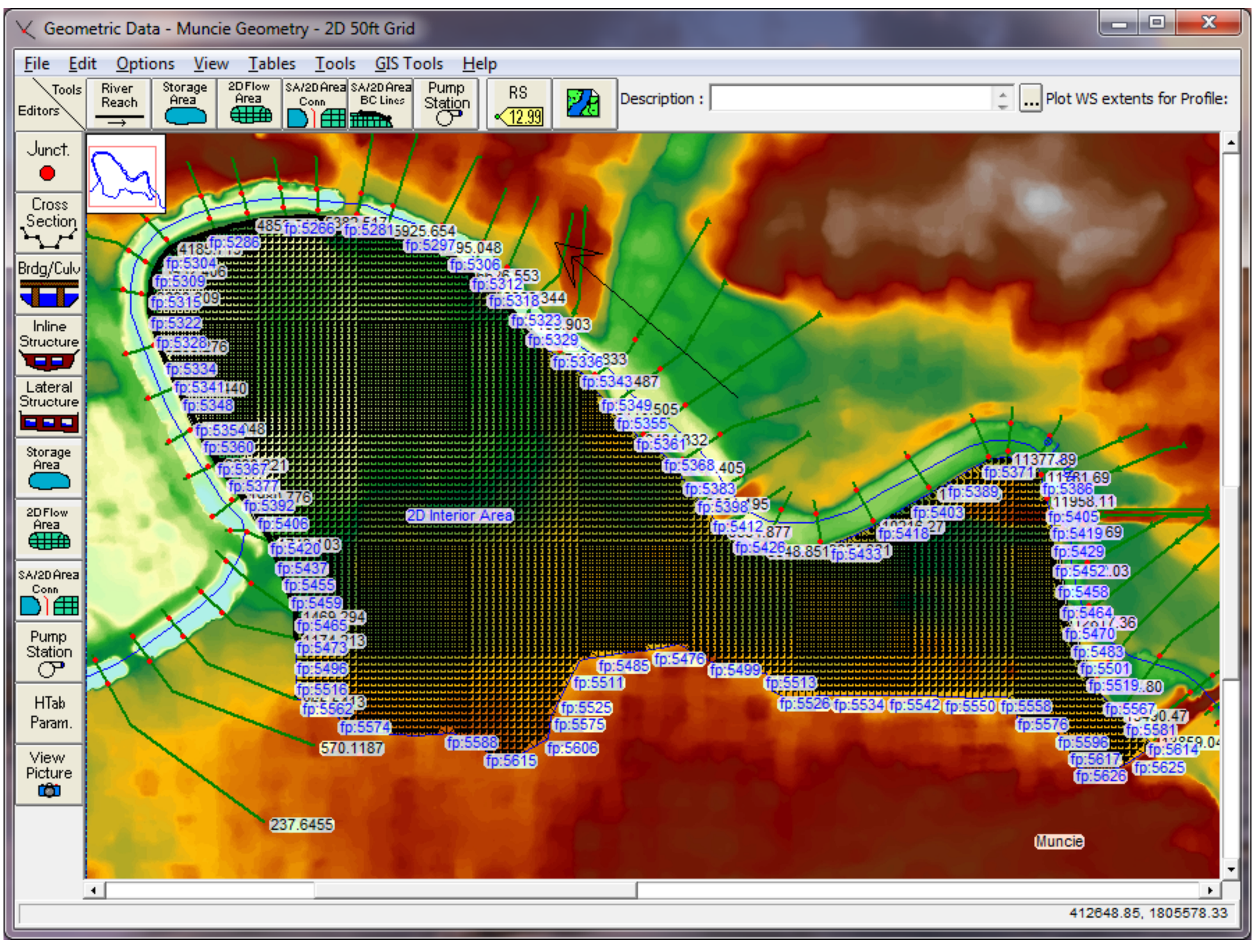


Figure 29. RAS Geometric Editor with the 2D Flow Area boundary Face Points displayed.

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 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 linked up levee is shown in Figure 30.

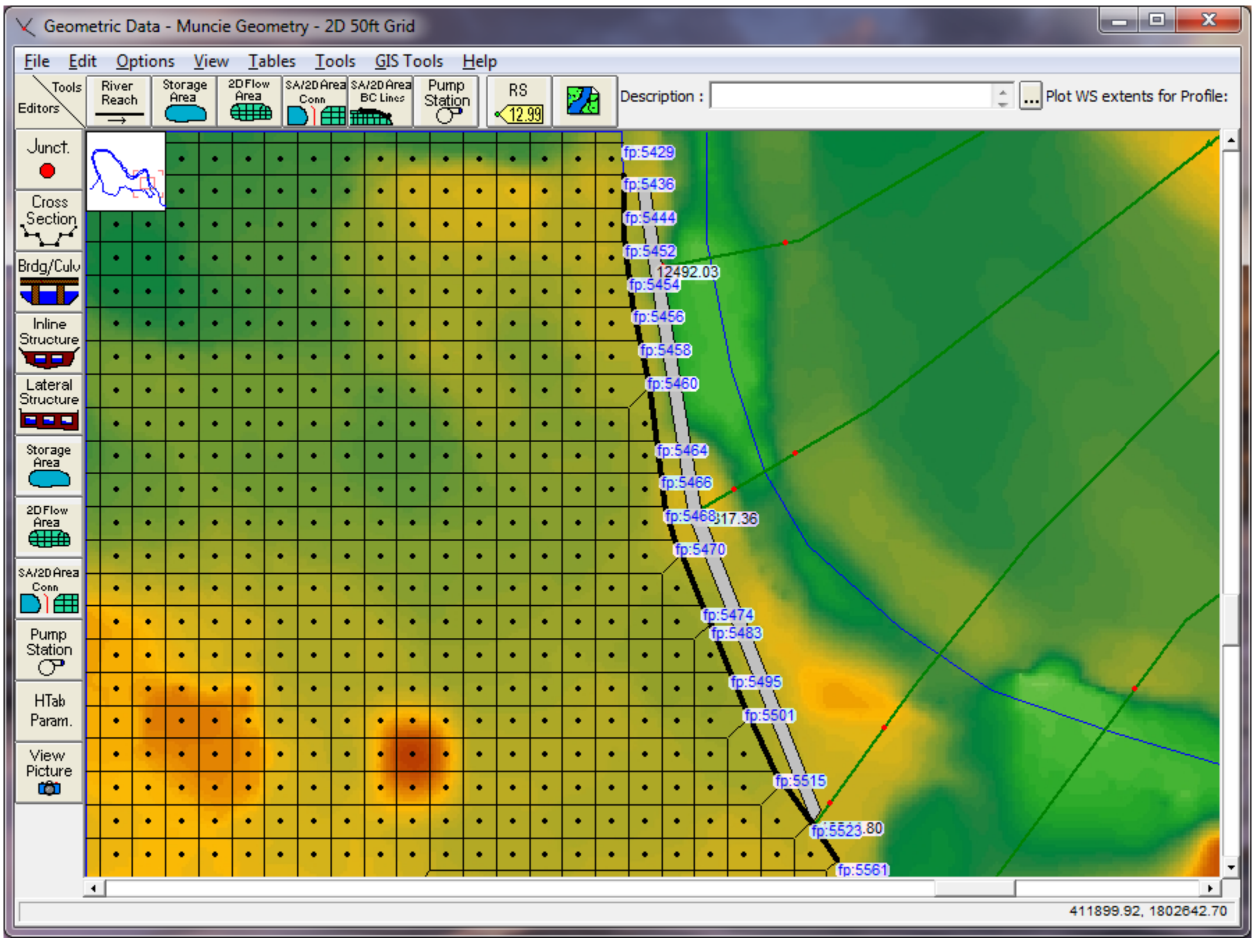


Figure 30. Example of a Lateral Structure (levee) hooked up to a 2D Flow Area.

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. The process of hooking up a Lateral Structure to a 2D Flow Area is listed below:

1. Add the Lateral Structure as you normally would 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 centerline of the top of the structure).
2. For the "Tailwater Connection" option on the Lateral Structure editor, select the **Type** as "**Storage Area/2D Flow Area**". Then for the "**SA/2DFA**" field, select the name of the 2D Flow Area you are going to connect the lateral structure too, 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 31).

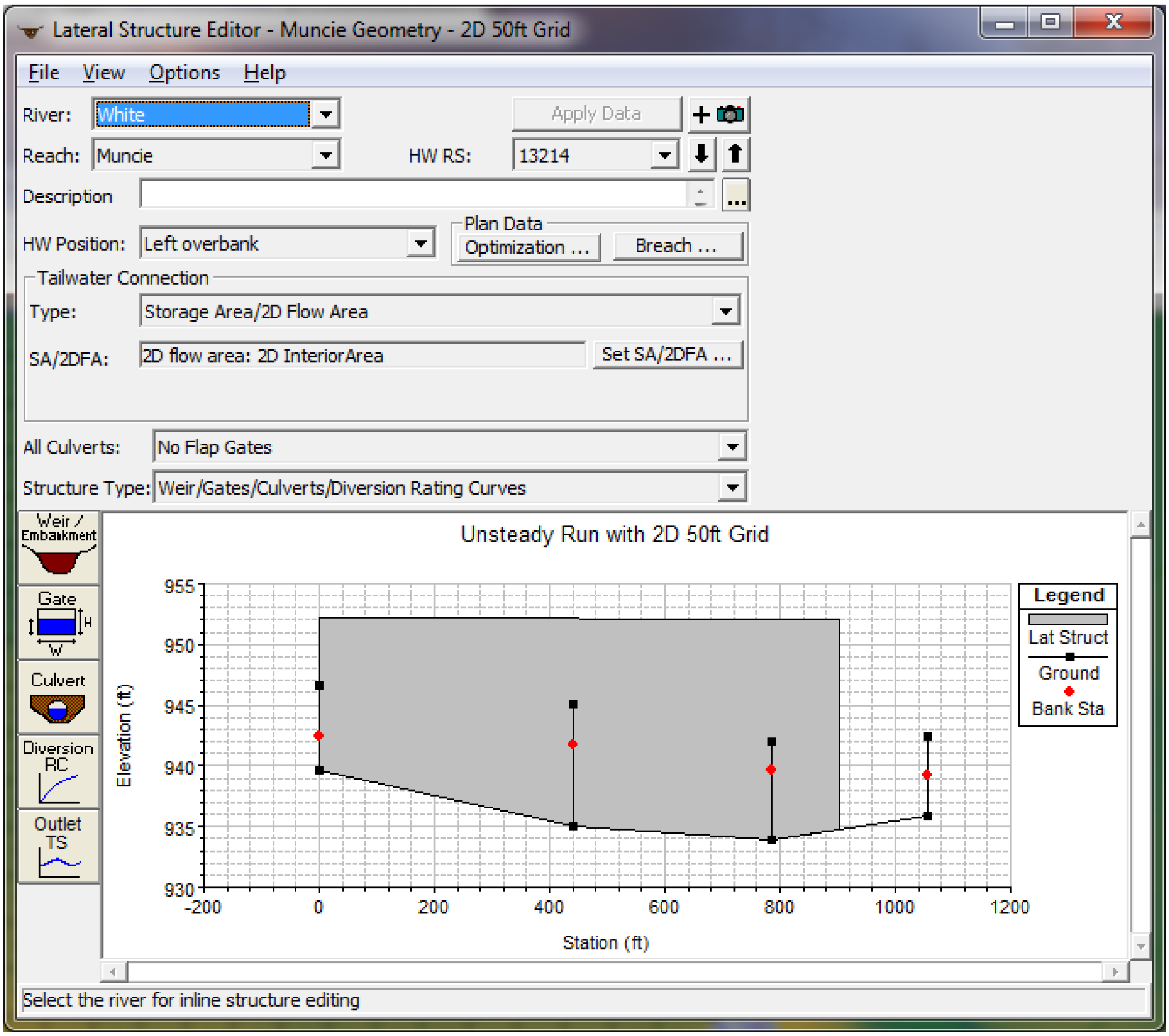


Figure 31. Lateral Structure Editor with tailwater connection to a 2D Flow Area.

3. Next, select the "**Weir/Embankment**" button on the left side of the graphic window. This will bring up the editor that will allow you to define the top profile of the embankment, as well as line the lateral structure up with the 1D river cross sections (the headwater side of the structure), and link it to the 2D area Face Points (the tailwater side of the structure), as shown in Figure 32.

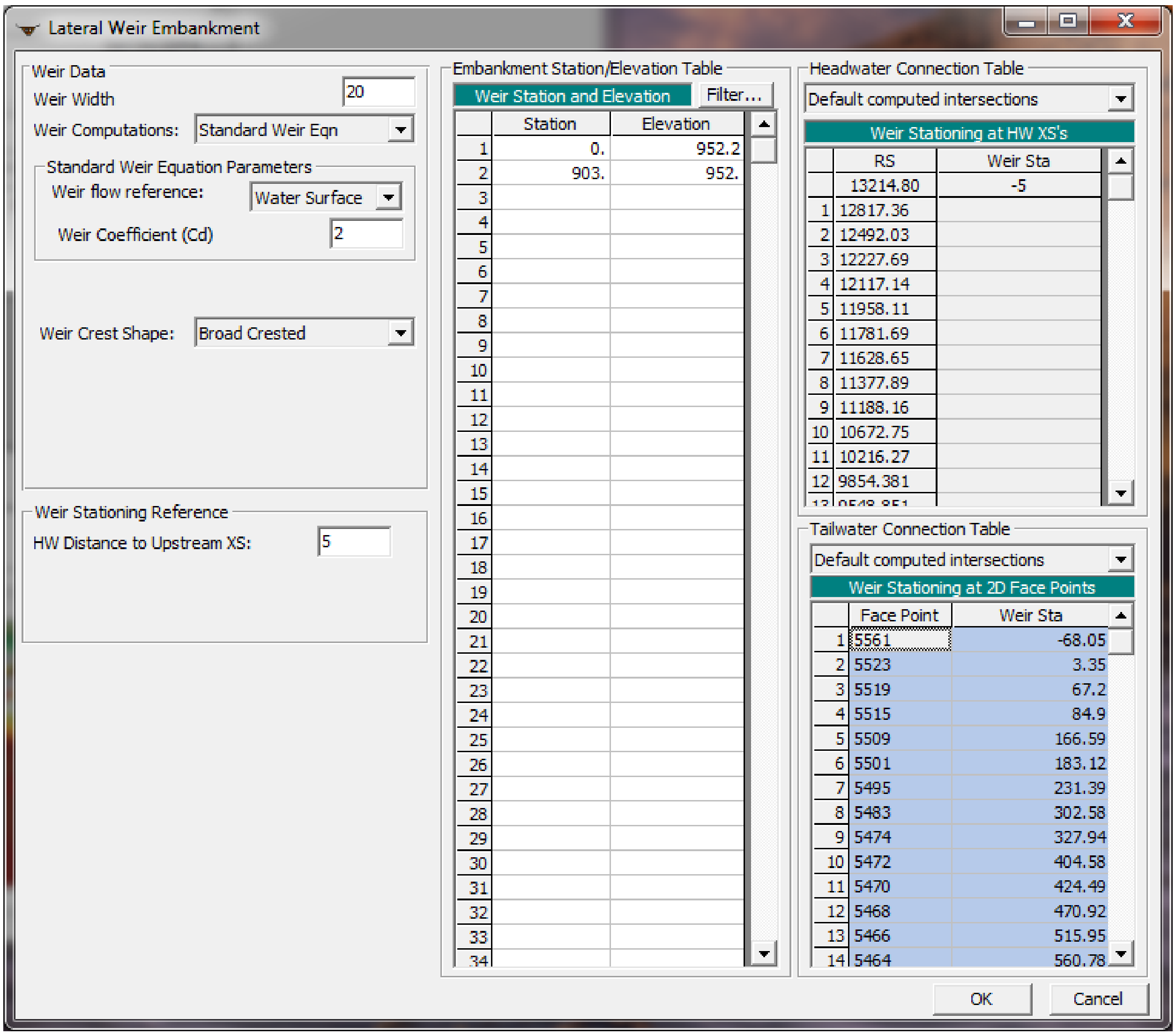


Figure 32. Lateral Structure Editor with structure Station/Elevation data, and 2D Face Point stationing.

As shown in Figure 32, the user goes about the normal process of entering a Lateral Structure in RAS by entering the: weir width, weir coefficient, HW Distance to Upstream XS, and the Weir Station and Elevation points. This will define the top of the lateral structure (levee) profile. For the Headwater (HW) connection to the 1D cross sections, the user can use the default, which is to have RAS compute the intersection of the 1D cross sections with the Lateral Structure based on the cross section overbank reach lengths (or channel lengths if user selected) and the Lateral Structure weir profile stationing (See Chapter 6 of the User's manual, "Entering and Editing Lateral Structure Data" section, for more detailed discussion).

4. The last step is to link the 2D Flow Area Face Points to the stationing of the Lateral Structure. This is done in the table on the lower right hand side of the Lateral Weir Embankment editor, as shown in Figure 32. By default, the software will come up with the Tailwater Connection table set to "**Default computed intersections".** In this mode, HEC-RAS will automatically determine the connections between your 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, or even modify the HEC-RAS computed intersections. This is done by selecting "User specified intersections" from the drop down menu at the top of the table. Once you have selected "**User specified intersections"** you can enter/change/modify the table as you see fit. However, realize that you must not skip any Face Points as previously discussed. To connect an HEC-RAS Lateral Structure to a 2D Flow Area correctly, you enter the 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 putting in 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: You cannot skip over (exclude) any of the Face Point numbers. If any face point along the boundary is skipped the model will not run, and it will give you an error message saying the connection to the 2D Flow Area is bad.

Note: If you make 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. When this occurs, the HEC-RAS automated Face Point connections will be adjusted such that the Lateral Structure weir stationing will be adjusted to make sure that the total length of the weir 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 you choose to enter the Tailwater connection using "User specified connections", then you have to figure this out on your 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 (which can be measured on the screen).

Once you have 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 you need/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 33). 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.

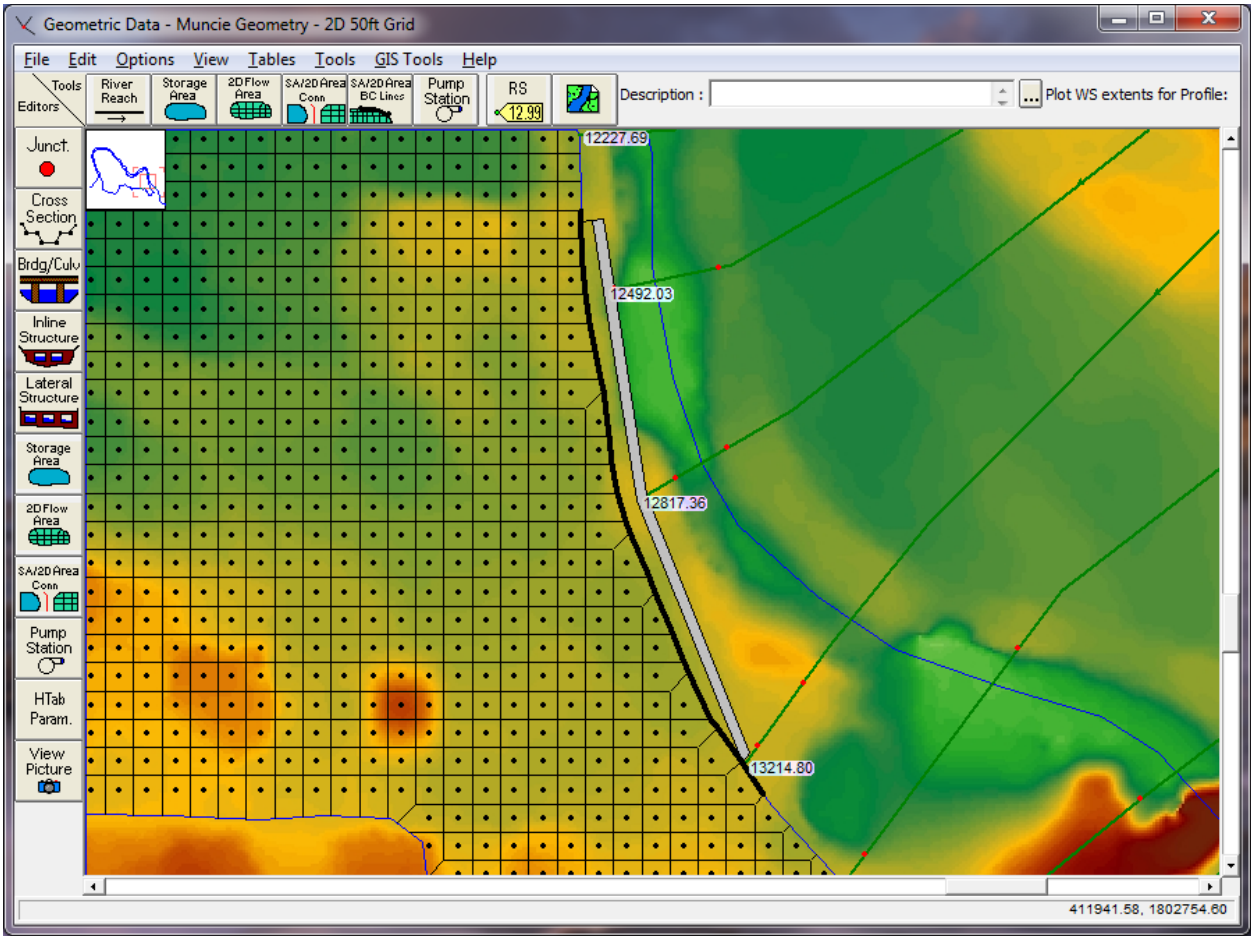


Figure 33. HEC-RAS Geometric Editor showing a thick black line for the connection of a Lateral Structure to the 2D Flow Area Face Points.

For this example a breach location/data has also been added for the analysis of this upper levee. Shown in Figure 34 is the Breach Data for this Levee/Lateral Structure.

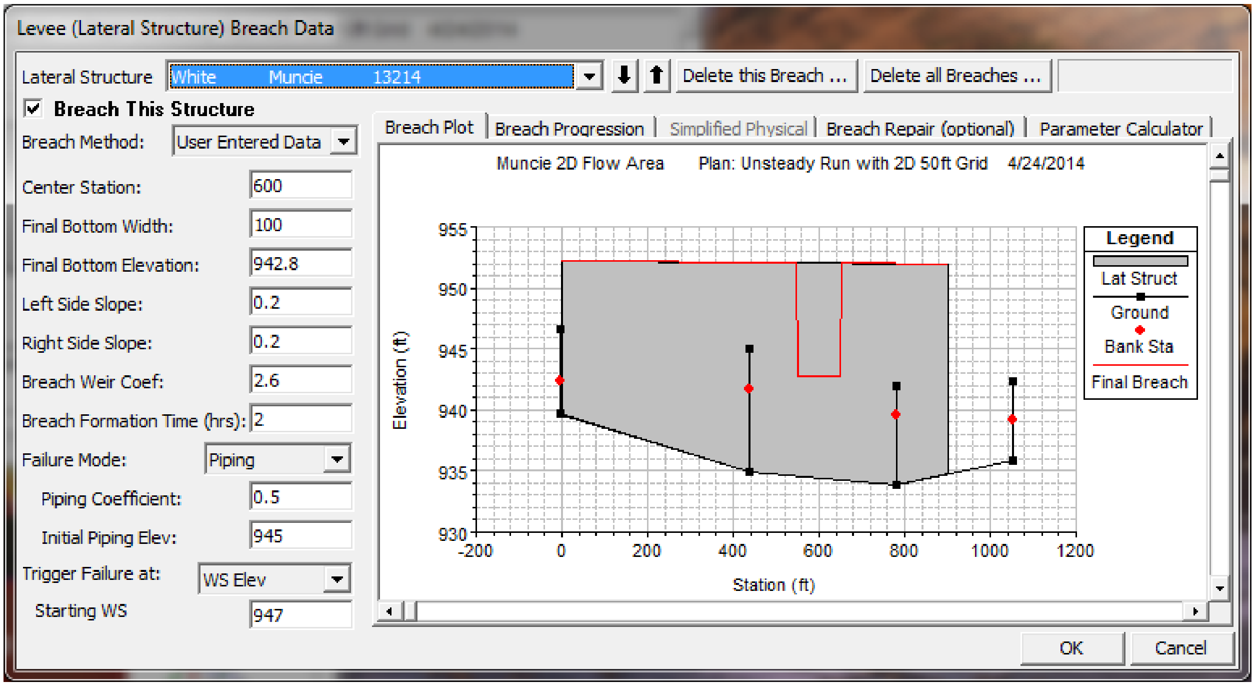


Figure 34. Breach Data Editor, with breach data for the upstream levee.

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

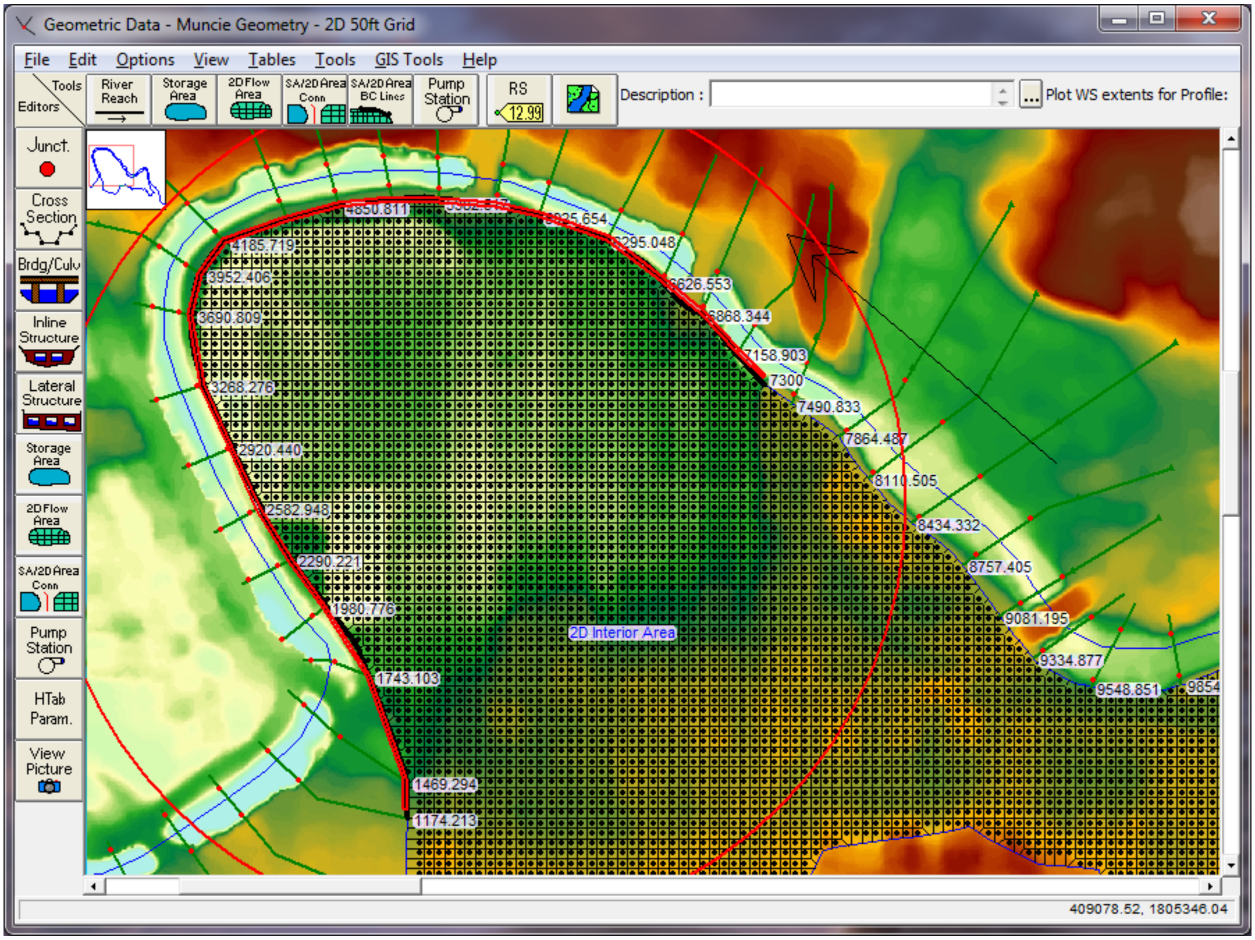


Figure 35. HEC-RAS Geometric Editor with downstream Levee (Lateral Structure) highlighted in red.

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 36.

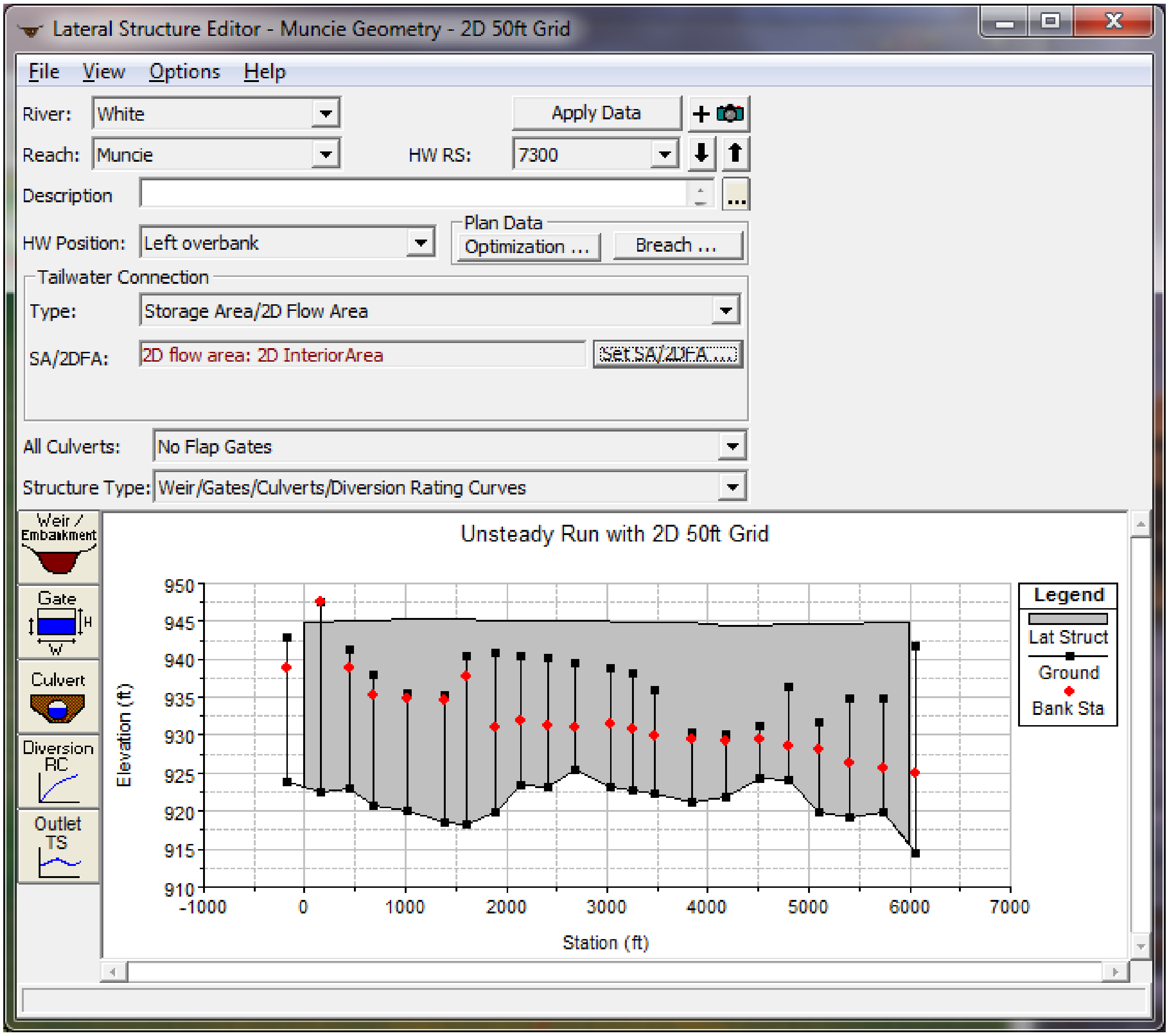


Figure 36. Downstream Levee (Lateral Structure) with a tailwater connection to the 2D Flow Area.

Shown in Figure 37, is the Weir Embankment Editor, with the data for the Lateral Structure stationing and elevations, as well as the Lateral Structure linked to the 2D Flow Area Face Points.

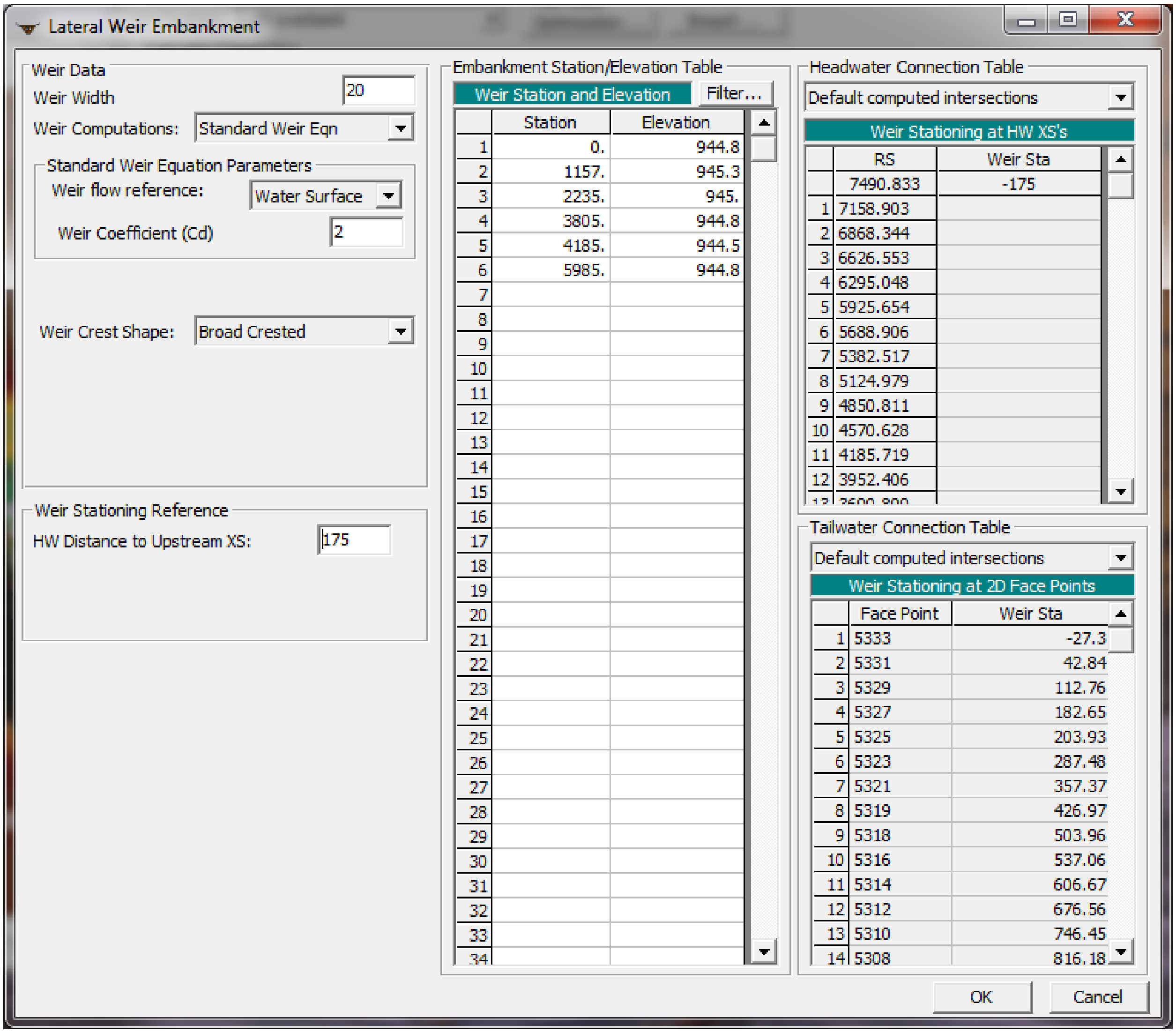


Figure 37. Lateral Weir Embankment editor with data for the downstream levee and linked to the 2D Flow Area Face Points.

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 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 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.	**0.1** to **0.5** SI Units: 0.06 to 0.28

Note: The number 1 problem HEC-RAS users have been having when interfacing 1D river reaches with 2D Flow Areas, is using to high of a weir coefficient 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 0.1 to 0.5 must be used to get the right flow transfer and keep the model stable.

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.

2. 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 38.

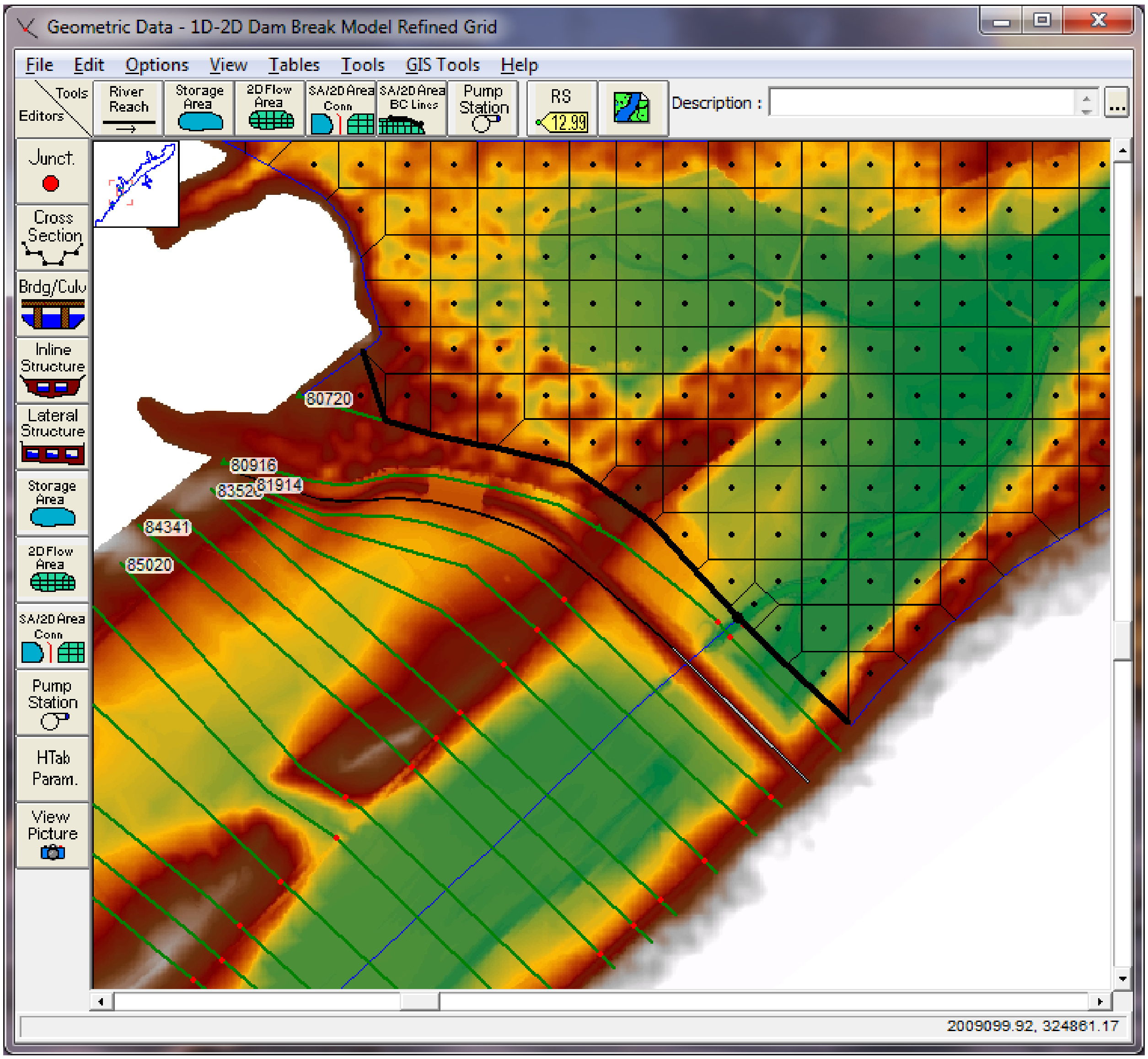


Figure 38. Example of an upstream 1D River Reach connected to a downstream 2D Flow Area.

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 too. Flow is distributed to the 2D cells based on the conveyance distribution in the cross section, and the stationing of how the cells are 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 Editor, and turn on the Option to "Move Points/Objects".
- Move the last point of the stream centerline inside of the 2D Area. The software will ask you if you want to connect the 1D River Reach to the 2D Flow Area. Select "Yes".

Once the 2D 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 you how it is connected. That is all that needs to be done for the connection.

Note: 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: Dry; 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.

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 (see below).

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.

3. 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 39.

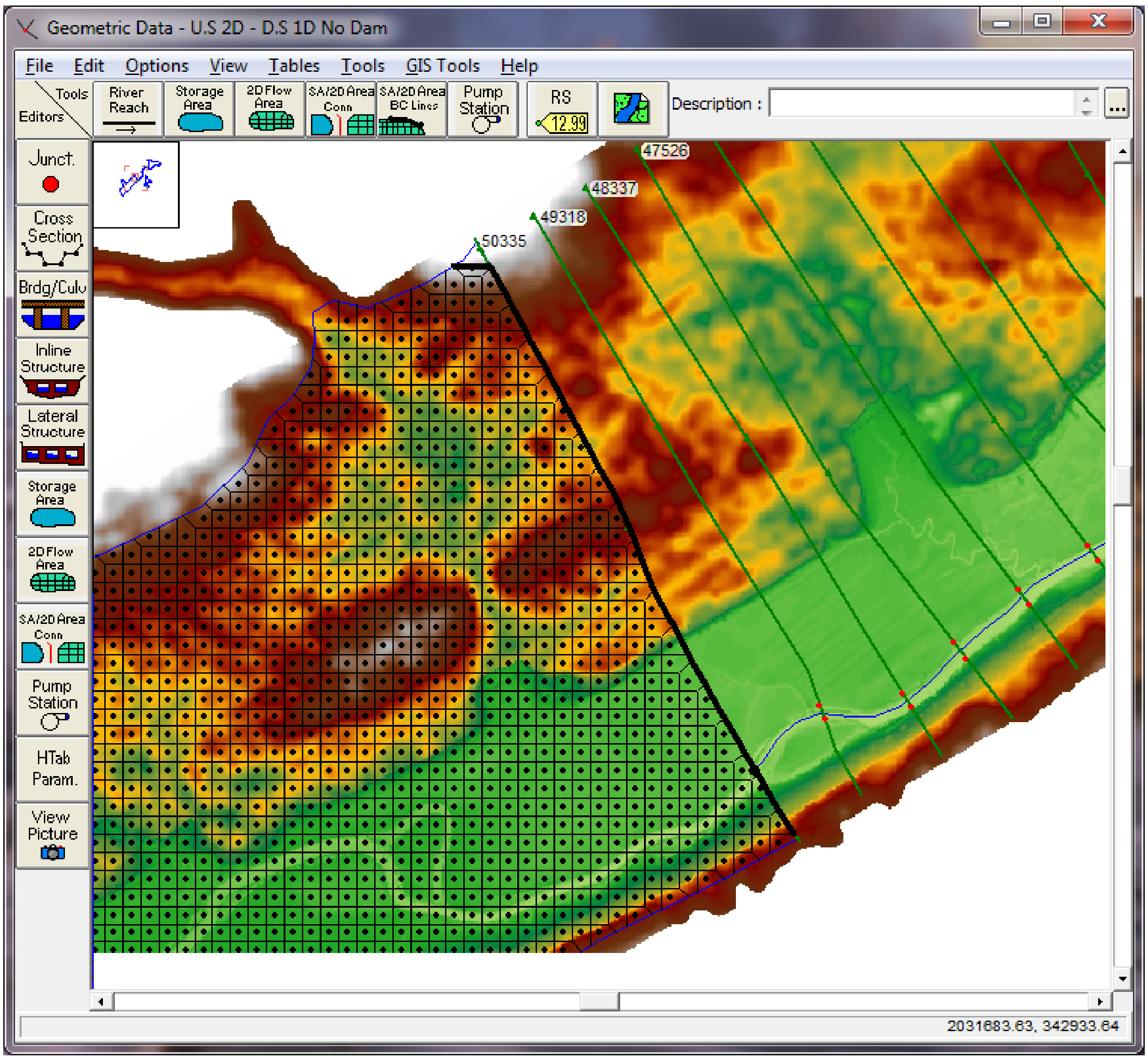


Figure 39. Example of an upstream 2D Flow Area connected to a downstream 1D River Reach.

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 Stages 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, you 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 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 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".

Once the connection between the 2D area and the 1D River Reach is made, the software will draw a black line along the 2D Flow Area cells outer boundary to show you how it is connected. That's 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.

4. 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 below in Figure 40 below.

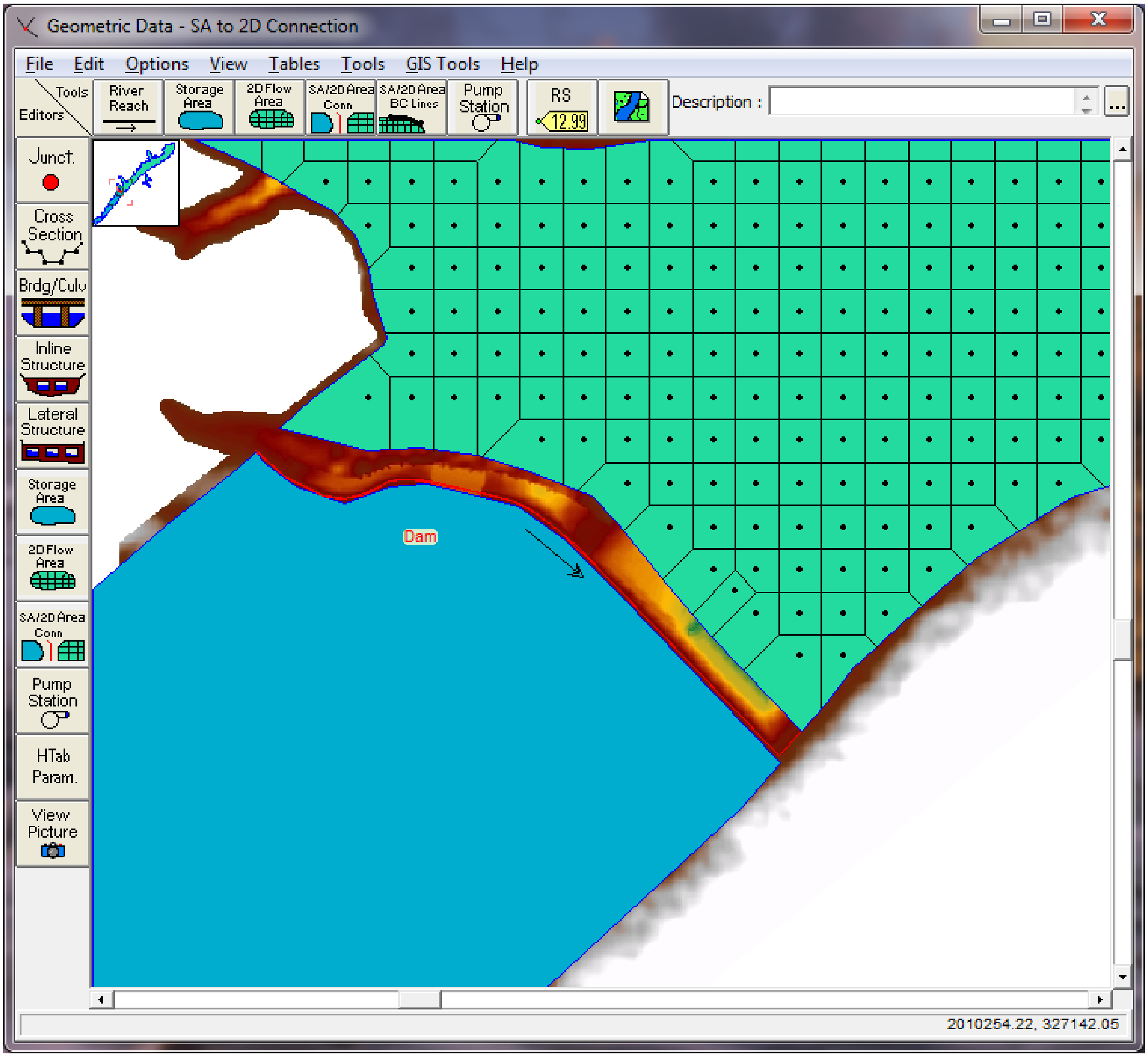


Figure 40. Example of a Storage Area connected to a 2D Flow Area.

In the example shown in Figure 40, 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 35, 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 (Inline Structure) 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 you want 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 you want 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 too. 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 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 you for a label to define the hydraulic structure. See the red line shown in Figure 40.
- Next, select the Storage Area/ 2D Flow Area Hydraulic Connection (**SA/2D Area Conn**) editor on the left panel of the Geometric data editor. This will bring up the editor shown in Figure 41.

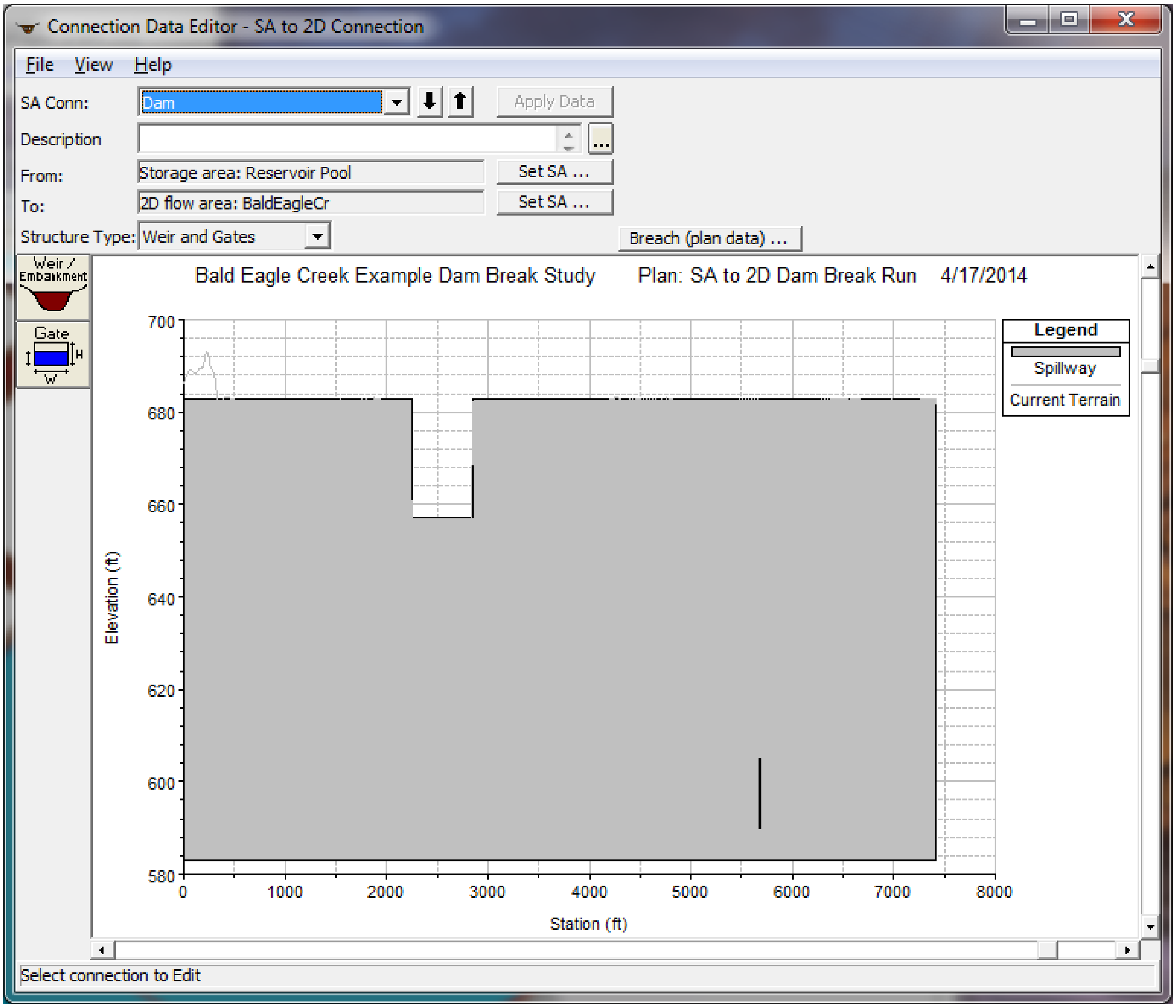


Figure 41. SA/2D Area Hydraulic Connection editor.

- On the **SA/2D Area Conn** editor set the "From" and "To" by selecting the buttons labeled "Set SA/2D Area". 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 41, 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.

5. 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 42.

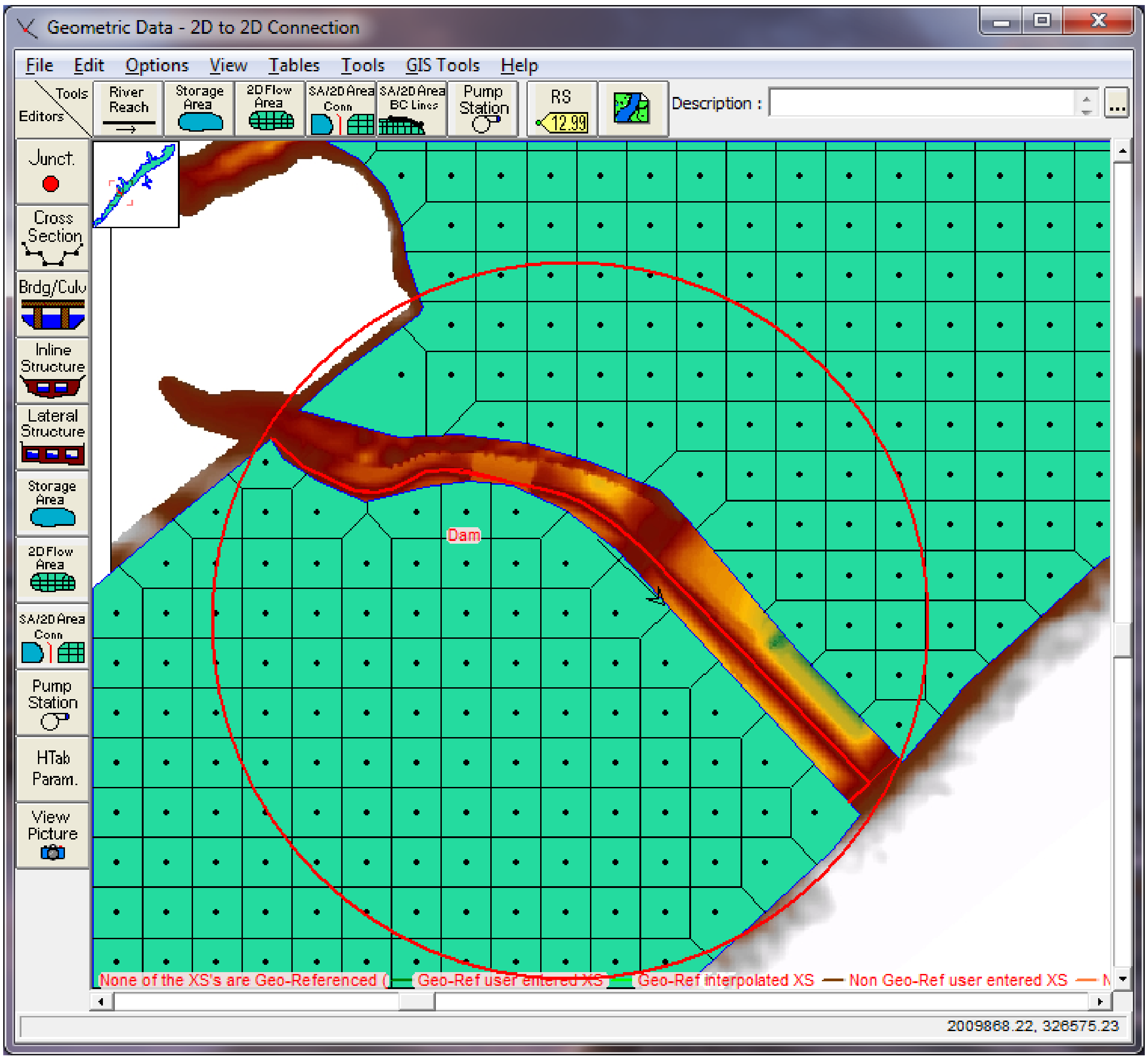


Figure 42. Example of connecting one 2D Flow Area to another with a Hydraulic Structure.

In the example shown in Figure 42, 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. In the example shown in Figure 42, a 2D Flow Area is being used to represent a reservoir pool. 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 you want to be used for the water surface in the hydraulic calculations of flow over and through the hydraulic structure.
- Select the Drawing tool at the top of the Geometric 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 you for a label to define the hydraulic structure. See the red line shown in Figure 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 43.

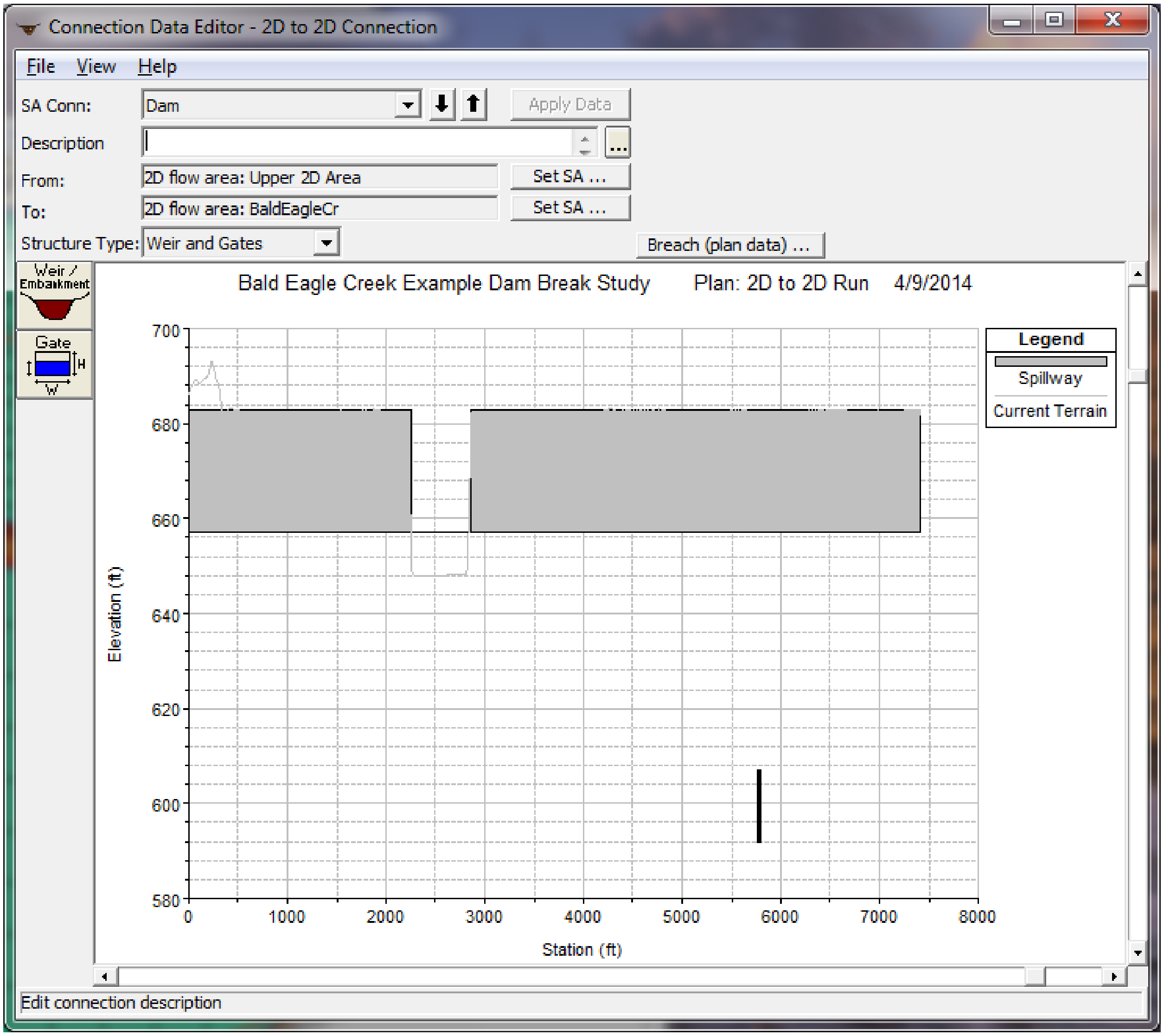


Figure 43. SA/2D Area Hydraulic Connection editor.

- On the **SA/2D Area Conn** editor set the "From" and "To" by selecting the buttons labeled "Set SA/2D Area". 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 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 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.

6. Multiple 2D Flow Areas in a Single Geometry File

HEC-RAS has the ability to have any number (within your 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 44.

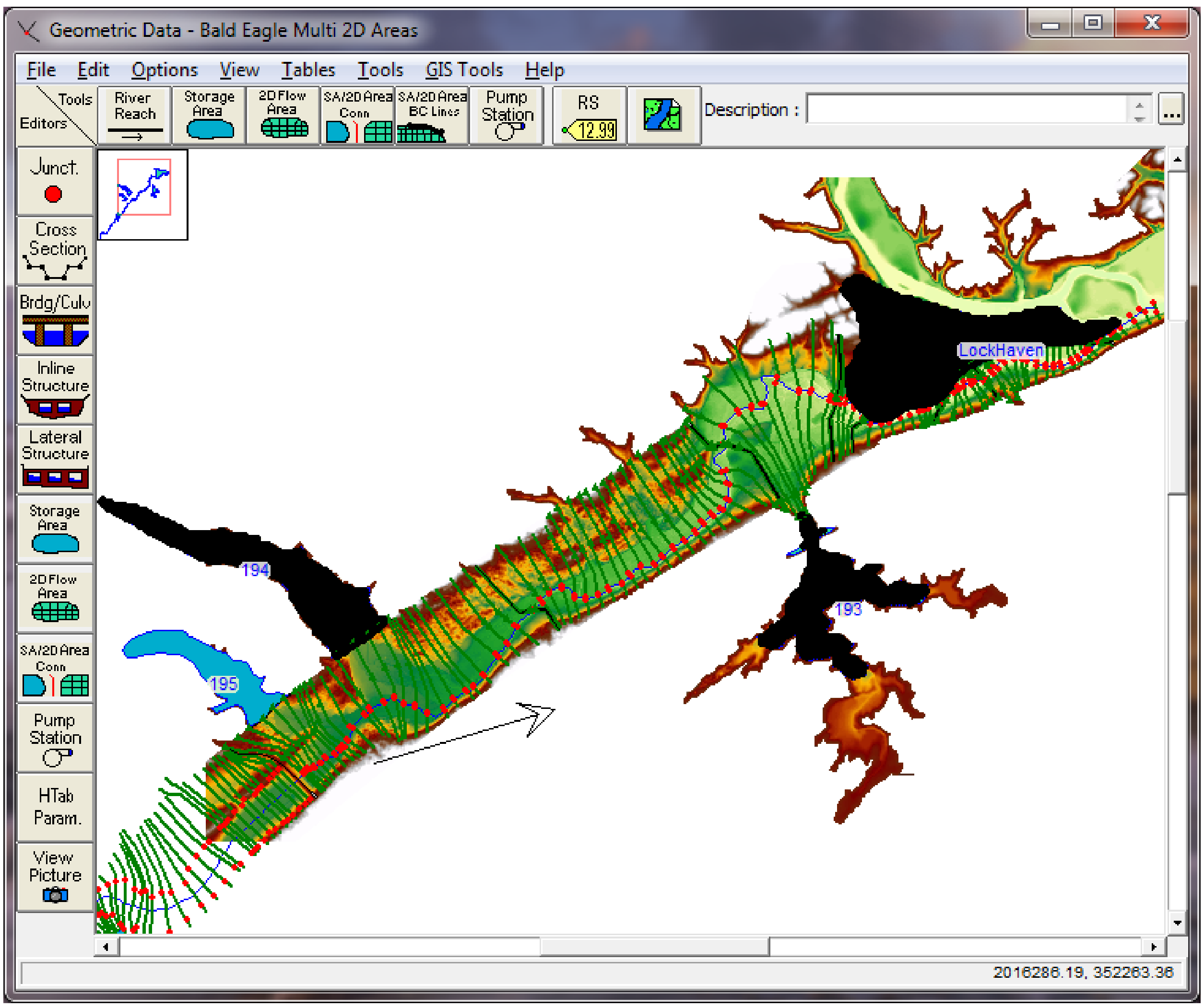


Figure 44. Multiple 2D Flow Areas in a single geometry file.

7. 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 (2D Cell Faces control flow movement). See Figure 45.

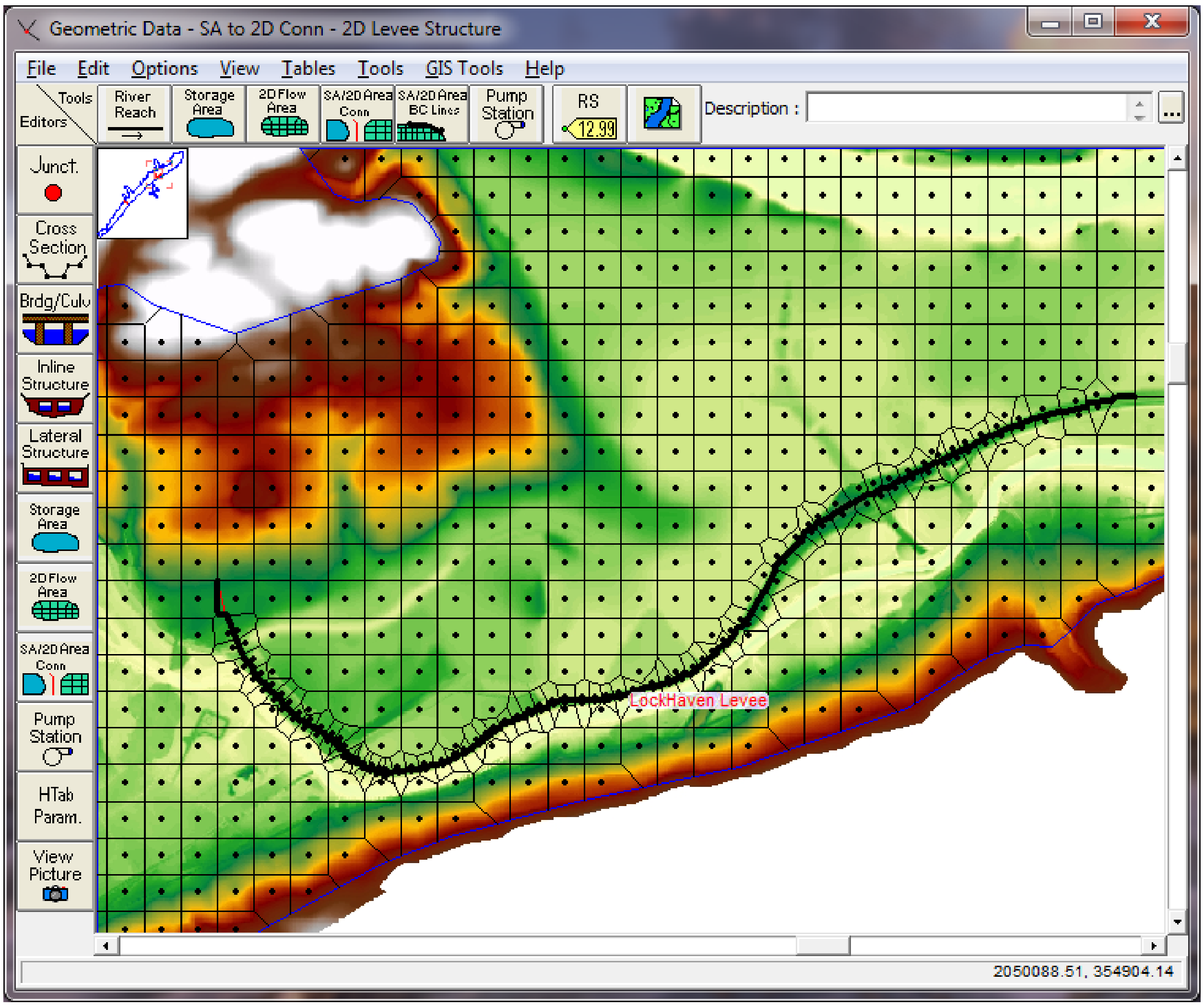


Figure 45. Example hydraulic structure inside of a 2D Flow Area.

To add a hydraulic structure inside of a 2D Flow Area do the following:

- First modify the 2D Flow Area mesh so that the faces of the cells go along the centerline of the top of the hydraulic structure. For example, as shown in Figure 45, a levee is being modeled inside of a single 2D Flow Area. The 2D Flow Area mesh was modified (cell center points were added and moved) to have cells on both sides of the levee, such that the faces between the levee lined up on top of the levee. This requires adding enough cells to get the correct detail, as well as placing cell centers at equal distances apart on each side of the structure. [Note: currently this is tedious with the HEC-RAS mesh editing tools. However, future versions of HEC-RAS will allow you to draw a break line on top of the levee (or hydraulic structure) and it will automatically modify the mesh to align the cell faces with the structure].
- Next, select the Drawing tool at the top of the Geometric editor labeled "**SA/2D Area Conn**". Then draw a line directly down the center of the hydraulic structure along the cell faces that represent the structure (this line should 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 be 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 you for a label to define the name of the hydraulic structure. See the black line in Figure 40.
- Next, select the Storage Area/ 2D Flow Area Hydraulic Connection (**SA/2D Area Conn**) editor on the left panel of the Geometric data editor. This will bring up the editor shown in Figure 46.

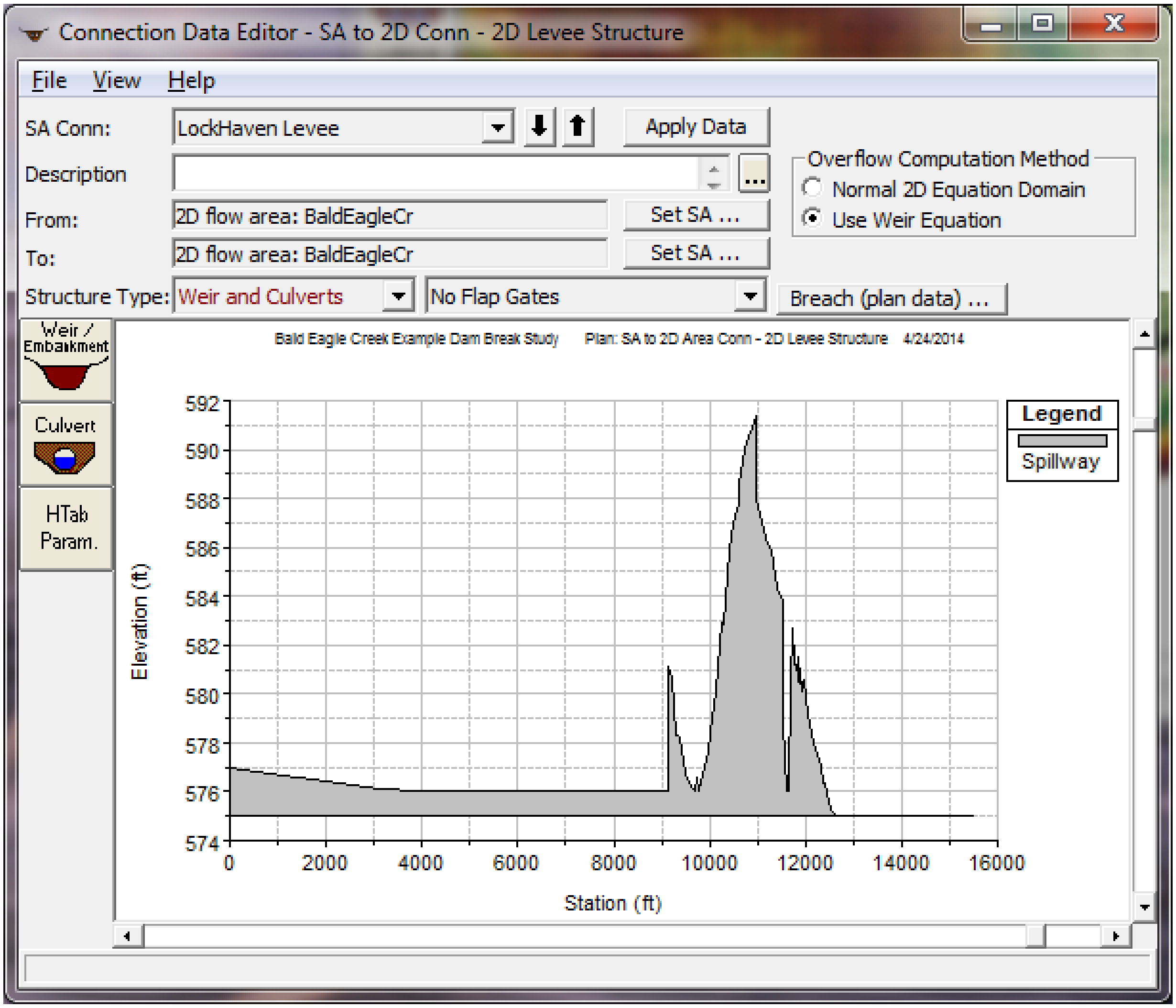


Figure 46. Example of using SA/2D Area Conn to put a hydraulic structure inside a 2D Flow Area.

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 (The user entered weir line is not allowed to be lower than the natural ground. However, a breach line can go lower, see below). 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 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 you 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.

D. External 2D Flow Area Boundary Conditions

1. Overview

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

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

The Normal Depth and Rating Curve boundary conditions can only be used at locations where flow will leave the 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.

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** (see Figure 47). 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 draw the external boundary condition, click the left mouse button one time at the location along the outside perimeter of the 2D Area where you want the boundary condition to start. Next you can add points by single clicking along the perimeter, then double click to end the boundary condition line at the location where you want it to end. Once you double click 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 47, 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, you can use any name you want.

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. You can also have one or more stage hydrographs linked to the same 2D Flow Area. And you can have Rating curves and Normal Depth boundary conditions hook up at multiple locations to allow flow to leave the 2D area.

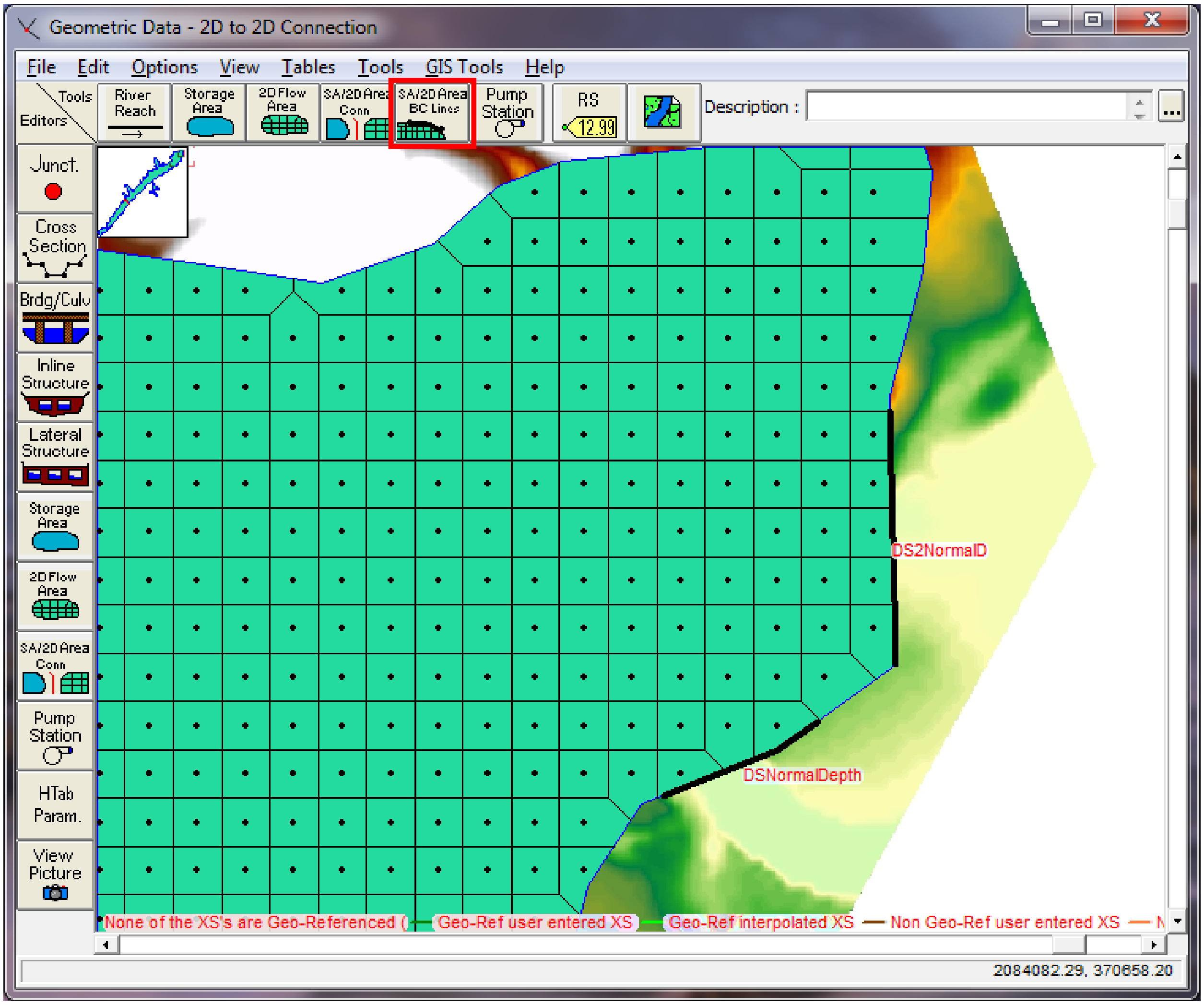


Figure 47. Example of adding an External 2D Flow Area boundary condition location.

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 48).

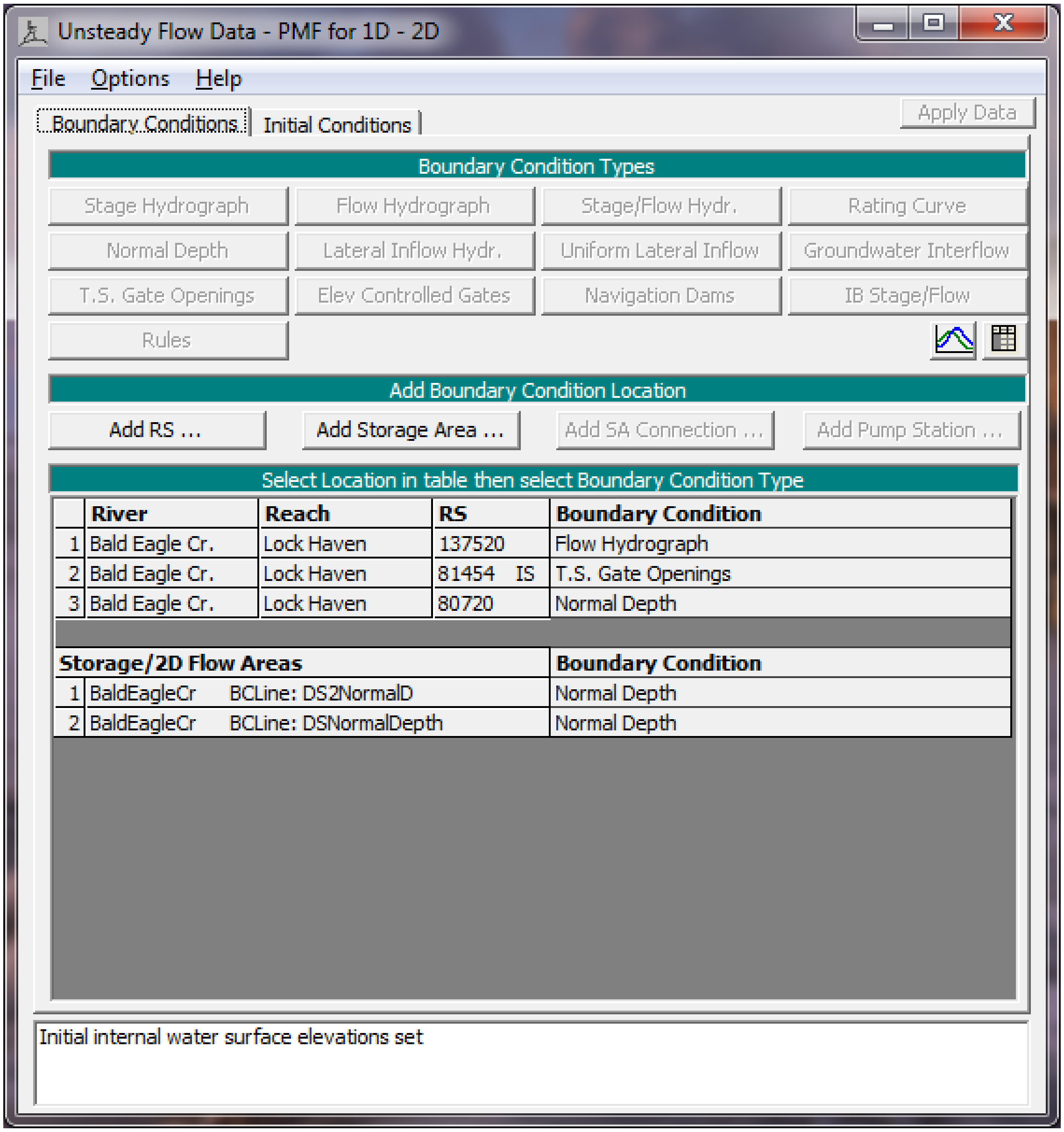


Figure 48. Example of adding external boundary conditions directly to a 2D Flow Area.

As shown in Figure 48, 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 47, 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.

2. 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 along the Boundary Condition Line. This depth is then put into the boundary Line cross section, and used for distributing the flow into the cells that fall under the boundary condition line. A flow distribution in the cross section is 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 each cell.

3. Stage Hydrograph

A Stage Hydrograph can be used as 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.

4. 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 give 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.

5. 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.

E. 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 "Initial Conditions Ramp up Time" option at the beginning of the run.

1. 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 49). 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).**

2. 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 49). More sophisticated starting water surfaces is another item planned for future versions.

3. 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 use to start a model with a different equation set (i.e. you 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.

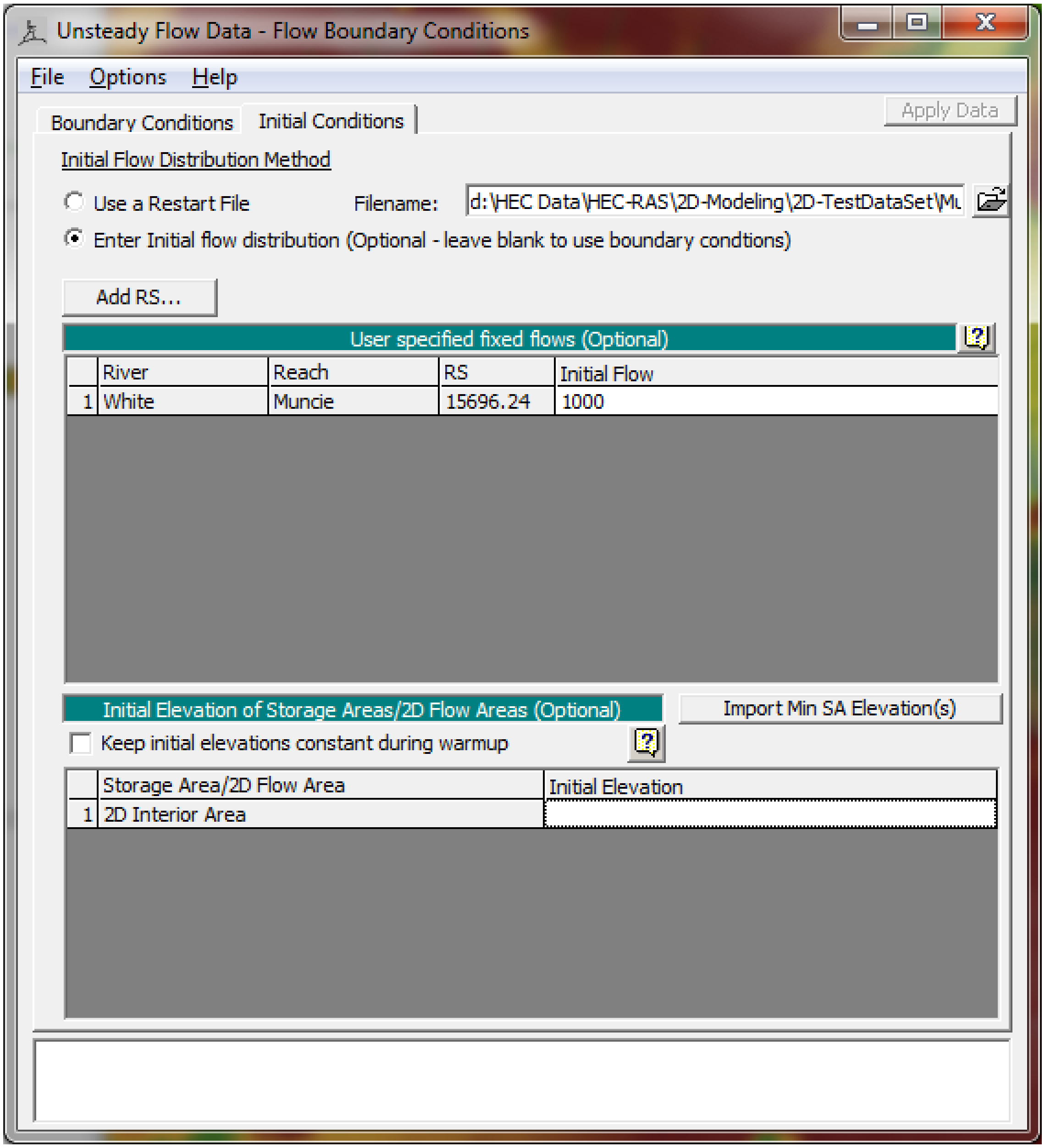
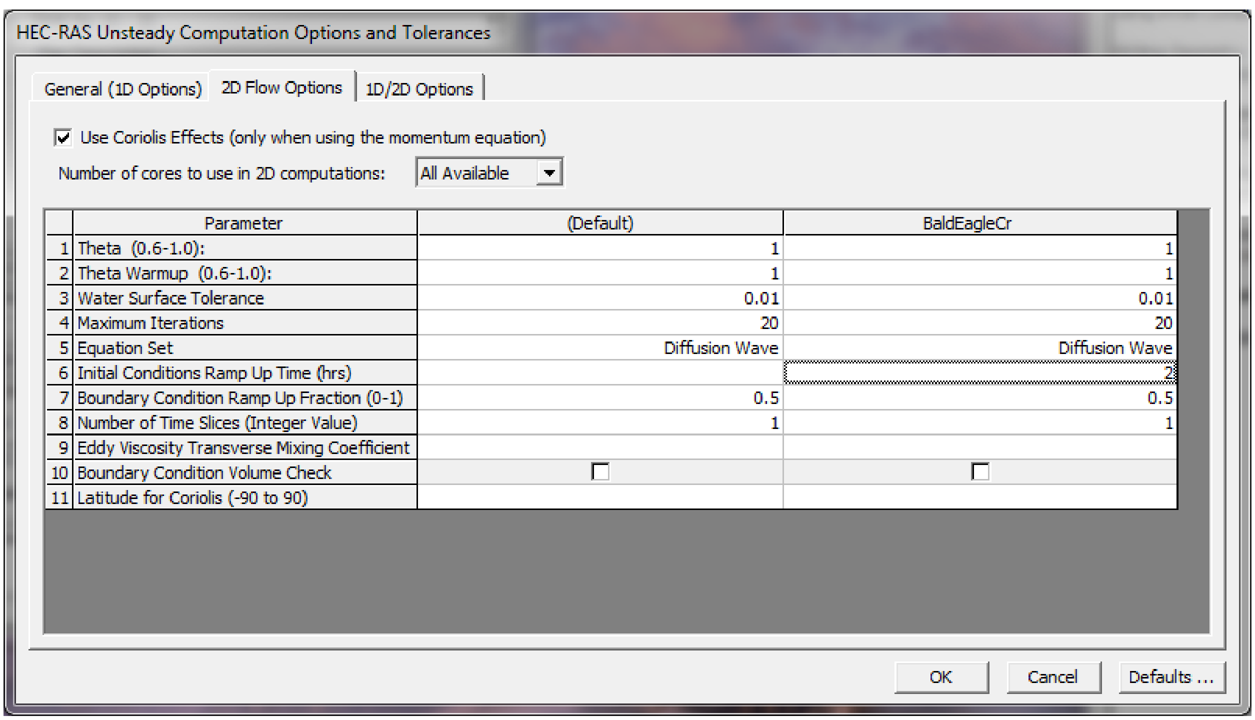


Figure 49. Unsteady Flow Data Editor with 2D Flow Area initial conditions.

4. Using the 2D Flow Area Initial Conditions Ramp up Time 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) this flow is allowed to transition during the first part of the warm-up period. This can reduce shocks to the system, especially 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” can 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 50 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).



HEC-RAS Unsteady Computation Options and Tolerances

General (1D Options) | 2D Flow Options | 1D/2D Options

☑ Use Coriolis Effects (only when using the momentum equation)

Number of cores to use in 2D computations: All Available

	Parameter	(Default)	BaldEagleCr
1	Theta (0.6-1.0):	1	1
2	Theta Warmup (0.6-1.0):	1	1
3	Water Surface Tolerance	0.01	0.01
4	Maximum Iterations	20	20
5	Equation Set	Diffusion Wave	Diffusion Wave
6	Initial Conditions Ramp Up Time (hrs)		2
7	Boundary Condition Ramp Up Fraction (0-1)	0.5	0.5
8	Number of Time Slices (Integer Value)	1	1
9	Eddy Viscosity Transverse Mixing Coefficient		
10	Boundary Condition Volume Check	☐	☐
11	Latitude for Coriolis (-90 to 90)		

OK | Cancel | Defaults ...

Figure 50. 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 feet 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 transition from 0 cfs up to 1000 cfs. The downstream stage boundary will transition from a depth of 0 feet up to a depth of 10 feet (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 and the depth at 10 feet.

The initial conditions, if any, are computed separately for each 2D area (in a “standalone” mode). The flow and stages from any boundary conditions hooked directly into 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 structures 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 normally.) If the user has entered a starting water surface for the given 2D area, that is used. Otherwise the 2D area starts out dry.

IV. 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.

A. Selecting an Appropriate Grid Size and Computational Time Step

Picking 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 us a single elevation for each cell and cell face (normal structured grid based models). Models that use triangles (commonly Finite Element models) use three elevations and a planar surface to represent each triangle, 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 verses 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 the following:

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.

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

Full Saint Venant Equations:

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

Where:	C	=	Courant Number
	V	=	Velocity of the Flood Wave (ft/s)
	ΔT	=	Computational Time Step (seconds)
	ΔX	=	The average Cell size (ft)

Diffusion Wave Equations:

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

The way to use these guideline equations, is to find the area(s) where you have high velocities and rapid changes in water surface and velocity. Take the average cell size in that area for ΔX. Put in the higher velocities in that area for V. Then select a ΔT, such that the Courant Number (C) is less than the suggested value (i.e. 1.0 for Full Equations and 2.0 for Diffusion Wave). However, you may be able to get away with a Courant number as high as 3.0 for the full equations and 5.0 for the Diffusion Wave equations, and still get stable and accurate results.

Note: 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 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.

B. 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 51). 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 new check box for "**Floodplain Mapping".** If you have set up RAS Mapper correctly, by bringing in a terrain data set and associating that terrain with your geometry files, then this option will work. If you turn this option 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 of you want 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 for automating 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 you select 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**.

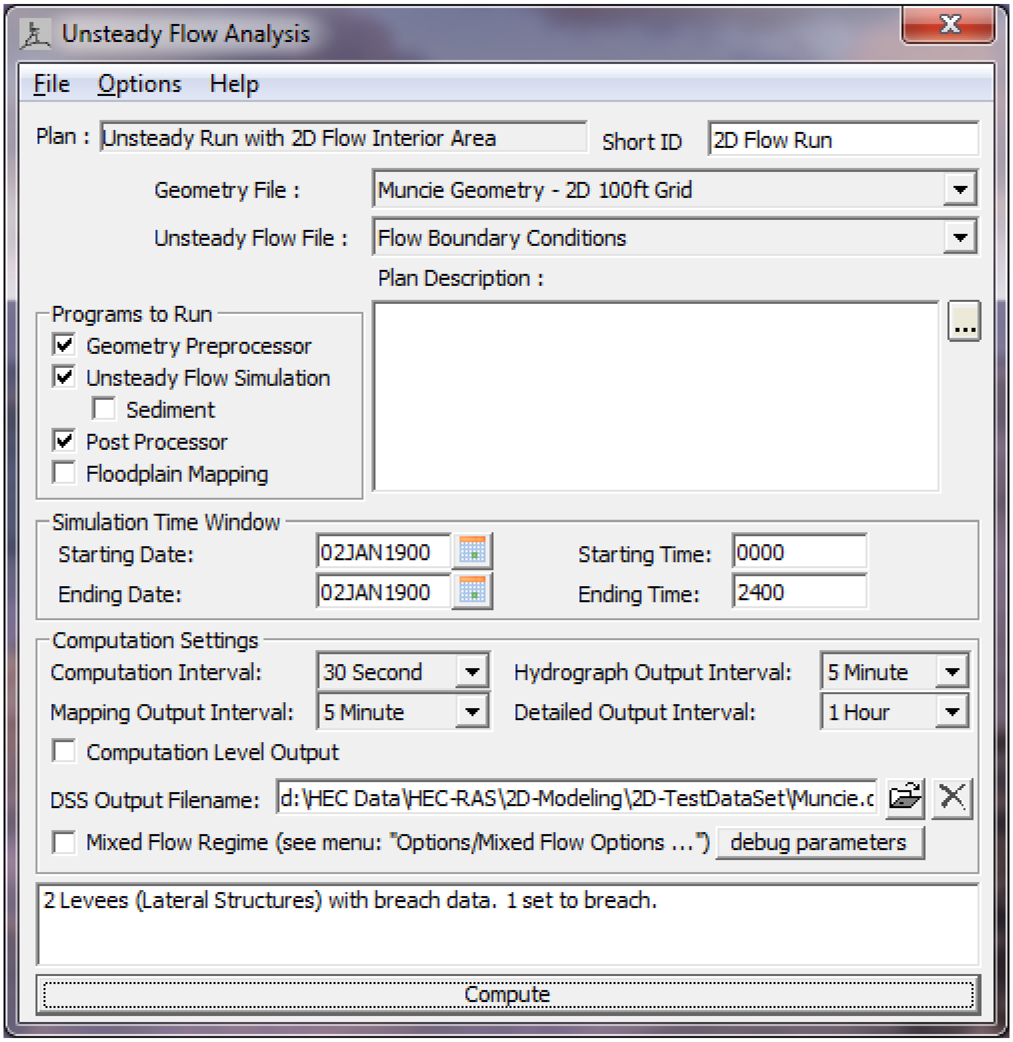


Figure 51. Unsteady flow Analysis Window with the new Floodplain Mapping feature.

Once you press 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, you do not need to run the Floodplain Mapping process, unless you 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 you want to look at results. Once you feel that you have a good result, you can make a static depth grid (stored to disk) from within RAS Mapper in order to send to HEC-FIA (Flood Impact Analysis) or a GIS program for display and analysis.

C. 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 52. 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.

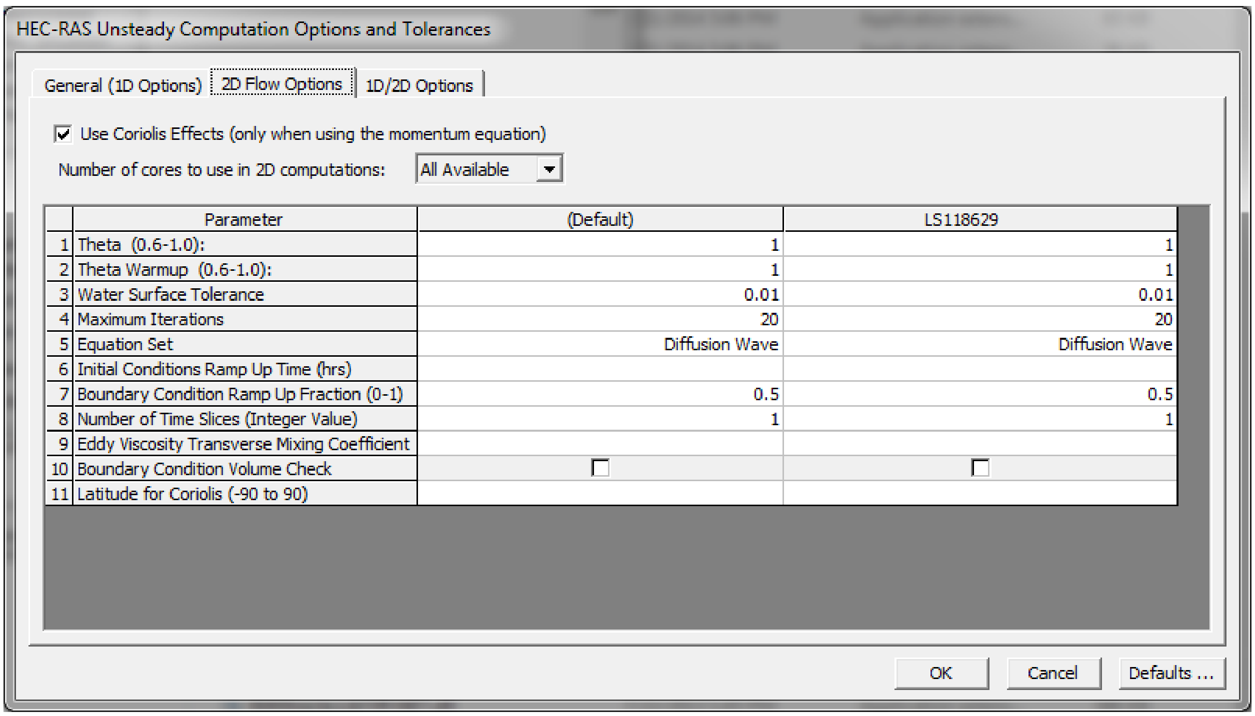


Figure 52. 2D Flow Area Calculation Options and Tolerances.

As shown in Figure 52, there are several computational options and tolerances that can be set for the 2D module. These Options are discussed below.

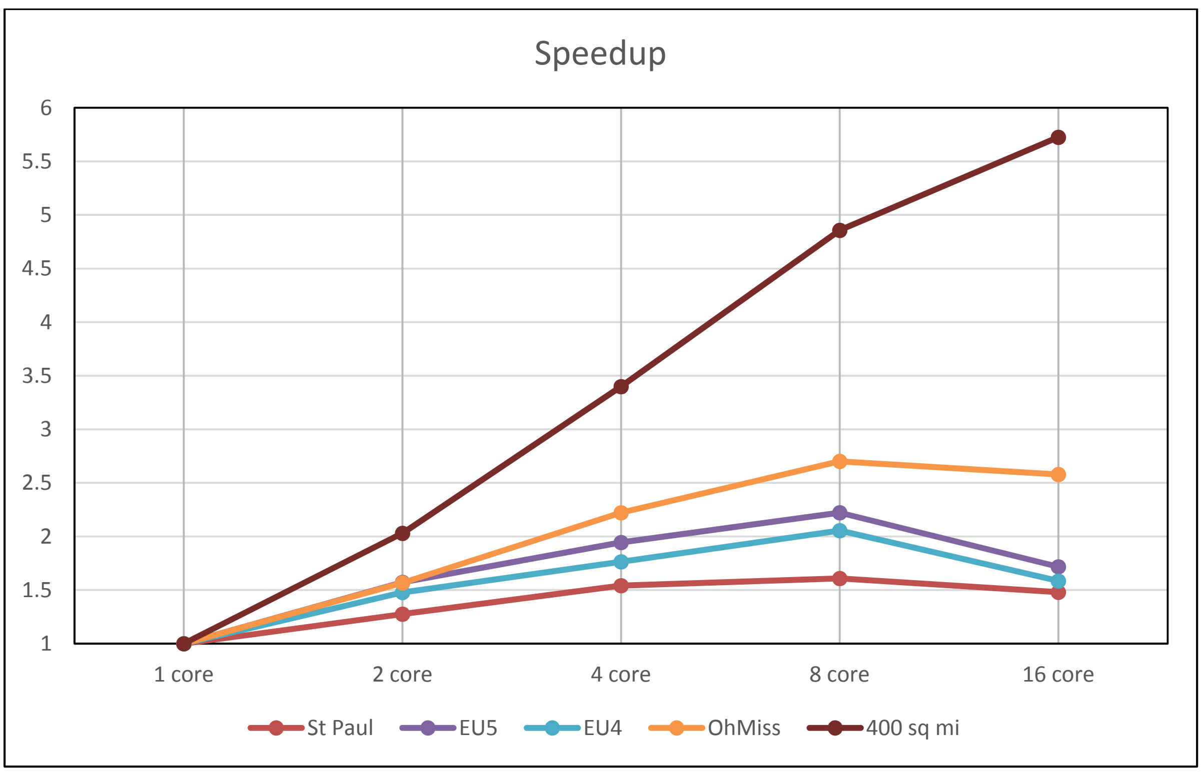
Use Coriolis Effects: Only used in the Full Momentum Equation

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 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 two-dimensional 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, you 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 you use more cores, 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) will almost always run faster with more cores, so use all that is available.

Shown below 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. 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 sets had speed improvements all the way up to 16 cores. This was the largest data set, with 250,000 cells.



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 solved solution 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 application of most real world flood runoff types of events, 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.

2D water surface calculation 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 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 Full Momentum Equation

The HEC-RAS two-dimensional computational module has the option of either running the **2D Diffusion Wave** equations, or the **Full 2D St. Venant** equations (sometimes referred to as the full 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 Full St. 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. More detailed discussions on the differences between 2D Diffusion Wave and 2D Full St. Venant will be available in the RAS User's Manual when HEC-RAS 5.0 is publically released to the general public.

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 a 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 may cause a model instability. So 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 Ram 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. User's 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 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).

Eddie Viscosity Transverse 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 Eddie Viscosity coefficient. To turn turbulence modeling on in HEC-RAS, enter a value for the Eddie Viscosity Transverse 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 Eddie Viscosity formulation can be obtained by entering a value greater than zero in this field. Below are some values for the Transverse Mixing Coefficient (D_T) that have been found to be appropriate under certain conditions.

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 answer. To turn on the 1D/2D iterations option, select the "**1D/2D Options"** tab. Then you can 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 you turn this option on, it is suggested that you start with a low value, like 3 or so. If the stability problem still exists with that number of iterations, then increase it from there.

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 (%)** gets used 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 (Currently in the 5.0 Beta version, this only works when an upstream 2D flow Area is connected to a downstream 1D river reach). 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, you may have 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 (put 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 wasting 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, the is most likely model specific.

New 1D Computational Options

We have added two new 1D only computational options, and got rid of one existing option. If you select the Tab labeled **General (1D Options)** on the **Computational Options and Tolerances** window, you will see we have changed the format of these options, as well as added two new options. The two new 1D options are:

1. **Maximum number of Iterations without improvement.** This option is off by default, but if you turn it on, it will monitor the maximum numerical error computed during the 1D Iterations, and if the error does not improve within X iterations, then the 1D solver stops iterating and goes on to the next time step. For example, let's say you have the default maximum number of iterations set to 20. If you set the "Maximum number of iterations without improvement" 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 you 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.

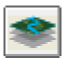
2. 1D Equation Solver. We use a matrix solution solver called "Skyline" which uses Gaussian elimination for reducing the size of the matrix. It has been streamline towards dendritic river systems, and is very fast. However, sometimes HEC-RAS models can be very large and have many interconnections (loops 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. it 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"** solve 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. Meaning, don't just compare the computational times, also compare the results to make sure they are the same.

Note: We also 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.

D. 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 comes 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). We plan on making the 64-bit version the “official” HEC-RAS release version.

V. Viewing Combined 1D/2D Output using RAS Mapper

Once you have completed an unsteady-flow run of your model, users 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 are viewed within RAS Mapper. Currently, you 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 RAS window). The RAS Mapper window shown in Figure 53 will appear.

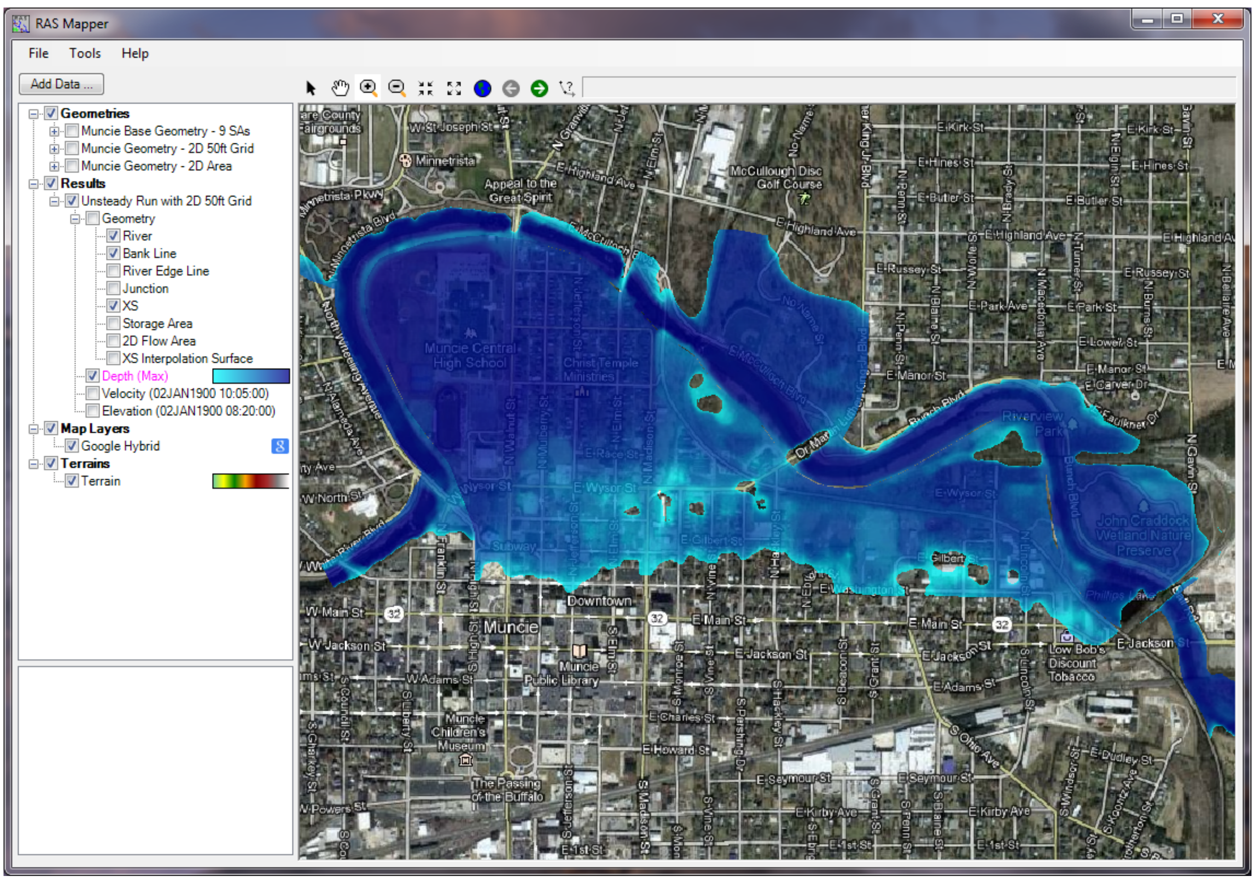


Figure 53. RAS Mapper window with combined 1D/2D flood inundation output displayed.

A. 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. Various types of map layer results can be generated, such as: depth of water, water surface elevations; flow (1D only right now); velocity (1D only right now); arrival time; and flood duration.
3. 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.
4. Computed model results can be animated or shown for a specific instance in time.
5. 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; velocity (2D node velocities, 2D average Face velocities, and 1D velocities); and 2D Face average shear stress
6. Users can query any active map layers value by simply moving the mouse pointer over the map.
7. Web Imagery, shape files, and point layers can be displayed as background layers behind the computed results.
8. You 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 your 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.

B. 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 Name for that run (see Figure 54 below). Beneath the **Results|Plan Name** Layer, by default there will be a **Geometry** Layer and a **Depth L**ayer. The Geometry Layer will contain the HEC-RAS Input Geometry Layers. The Geometry Layer includes sub layers of: River; Bank Line; River Edge Line; Junction; XS (Cross sections); Storage Area; 2D Flow Area; and XS Interpolation Surface. Any or all of these Geometry layers can be turned on for visualization of model elements.

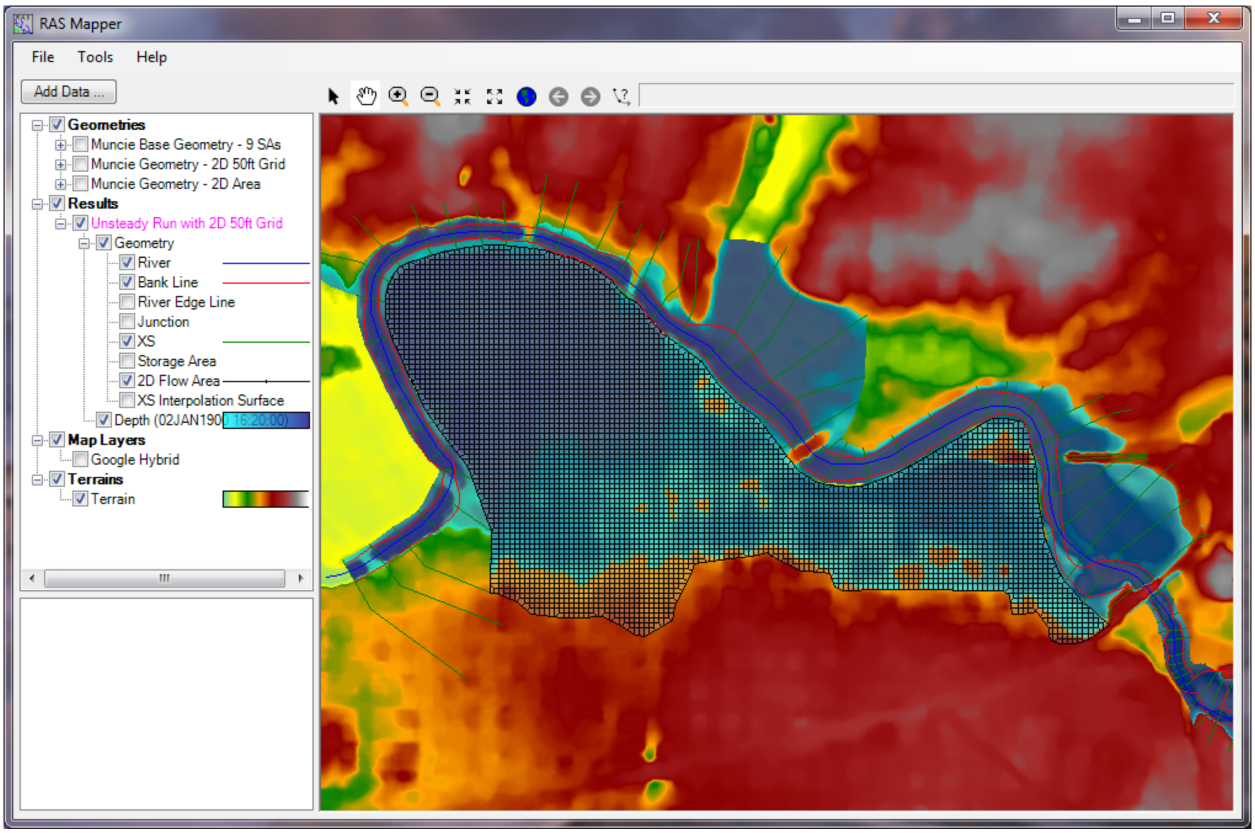


Figure 54. RAS Mapper with Default Results Layers shown.

By default, after a successful HEC-RAS model run, there will be a results layer called "**Depth**". The Depth layer 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 Depth layer will be computed and displayed on-the-fly, meaning it computes it in memory and displays it as needed. The underlying terrain used for computing depths is based on the view scale of the map. If you are zoomed in (large-scale mapping) the base (raw) data will be used for computing depths; however, if you are zoom out (small-scale mapping) a resampled version of the terrain is used. Therefore, the displayed depths may change slightly based on the scale at which the user is zoomed. By default, the depth layer is not a pre-computed depth grid stored on your hard disk.

Other results layers are available for visualization, but you have to request/create a results layer that you want 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 (Figure 55). 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 that window, to create a new results map layer.

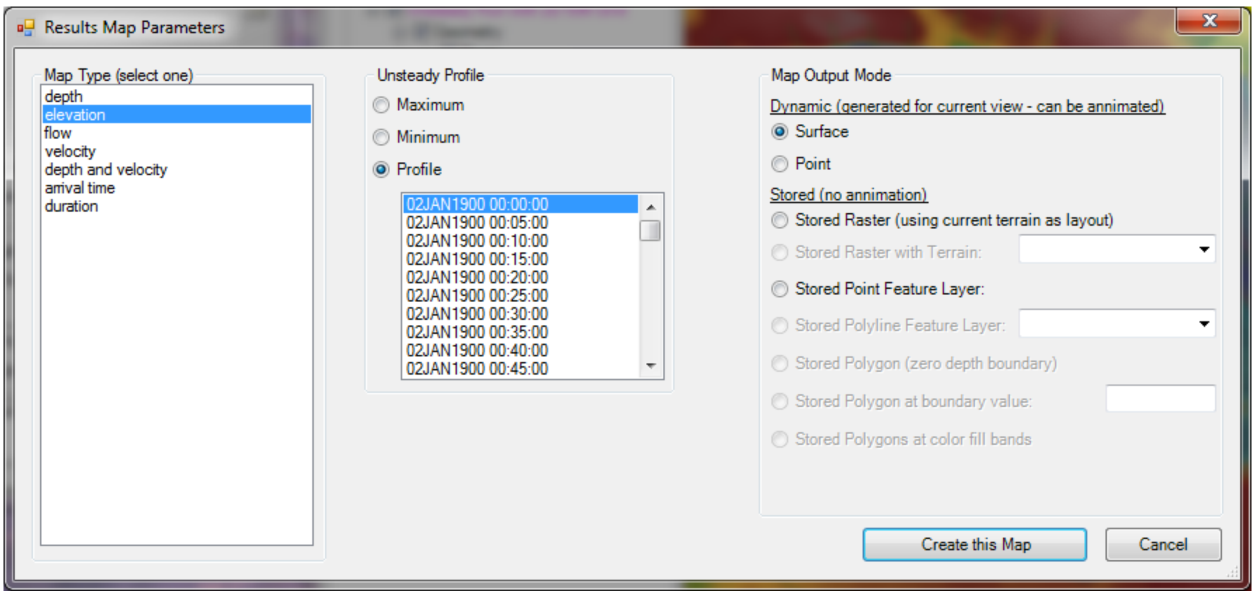


Figure 55. Example of the Results Map window used to create new results map layers.

As shown in Figure 55, the New Results Map Parameter window has three sections to select from. On the left is the "**Map Type**", where you select the parameter you want to map (create a layer for). After a parameter is picked, 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 you pick for the profile, you 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 you pick will be used for that static map.

The last thing to select on the window is the **"Map Output Mode"**. The map output mode is where you select whether the map will be a **Dynamic** layer, or else it will be a **Stored** 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 you want to create a depth grid, or other layer type, that you want 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 you can display it in a GIS and use it for another purpose.

Currently RAS Mapper is limited to creating the following map layers:

1. Depth of water
2. Water surface elevations
3. Flow (1D only right now)
4. Velocity (1D only right now)
5. Depth times velocity
6. Arrival time
7. Flood duration

Arrival time and flood 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 you may want to enter a higher value, like 1 or 2 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).

C. Dynamic Mapping

As shown in Figure 56, 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 "Unsteady Run with 2D 50 ft Grid". Under the "Unsteady Run with 2D 50 ft Grid" layer there are two sub layers: Geometry, and Depth. The Geometry Layer represents what

Geometry data was used in the run, and was written to the output file. The Depth layer is the Dynamic Mapping Depth Grid output layer. The key word here is "**Depth"**. The information following in the parentheses is just what current depth grid is showing in the plot. Inside the parentheses could be a specific Date and Time, or Max, or Min.

Right clicking on the "**Depth**" results layer for this Plan 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; Export Layer; and Animate Profiles (see Figure 56).

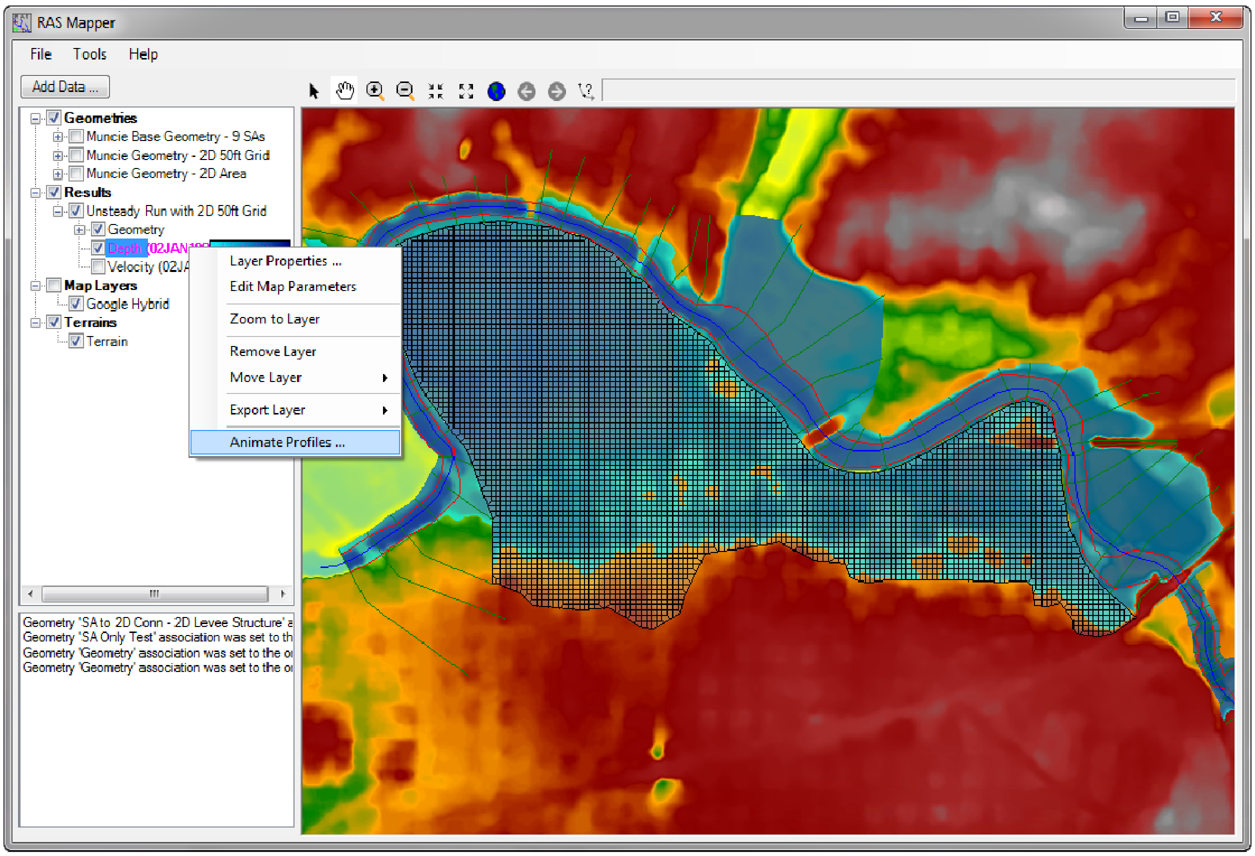


Figure 56. RAS Mapper with the Options menu displayed for a depth output layer.

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 is intended to allow 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 you to move a layer: Top; Up One; Down One; Bottom; Up a Level; and Down a Level. The user can also left-click on a layer and drag it up and down within the layers list.

Export Layer – This option has several export options available, such as: save feature to a shapefile; save feature to a GML file; convert feature to a point shapefile; convert feature to a multipoint shapefile; and Export Map for use in Google Map/Earth. Not all of these features are available for every map layer type.

Export Map for use in Google Map/Earth Viewers – 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 (256X256 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 57 will appear.

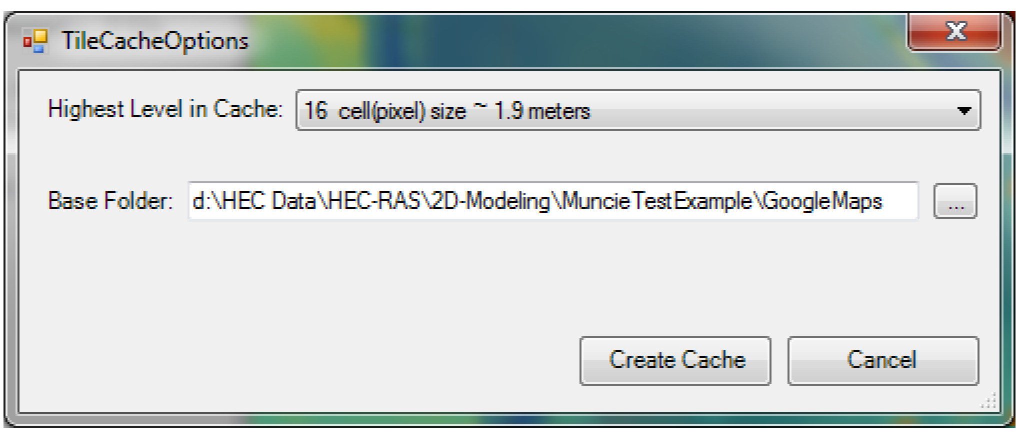


Figure 57. Tile Cache Options Editor.

The user is required to set the Base Folder in which these subfolders will be developed and the tiles will be stored. Additionally the user should select the cell size that will represent the most detailed tile level (when zoomed in). For example, if you pick level "16 cell (pixel) size ~ 1.9 meters" (as shown in the example above), your most detailed pyramid level will have tiles in which each pixel size represents 1.9 meters. If you plan on zooming way in and want great detail in the map, you would pick a higher level/smaller cell (pixel) size. However, the smaller you go, 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 your project.

Animate Profiles – This tool allows you to perform the Dynamic Mapping and the animation of inundation maps. The Animate Profiles tool bar is shown in Figure 58.

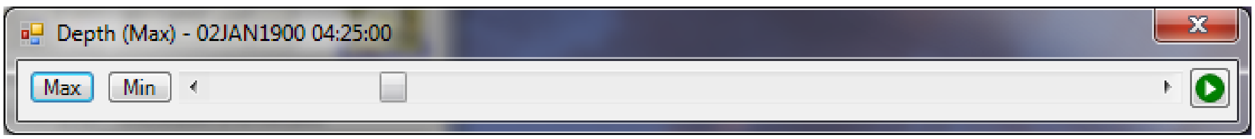


Figure 58. Dynamic Mapping/Animation Tool Bar.

As shown in Figure 58, when animating the "Depth" layer for a specific Plan, 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 you select the Play button (far right green arrow), then the map 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. You 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 you want to create a movie file use the Snagit software (or similar package) to capture the screen while animating the inundation results.

D. 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 59 will appear.

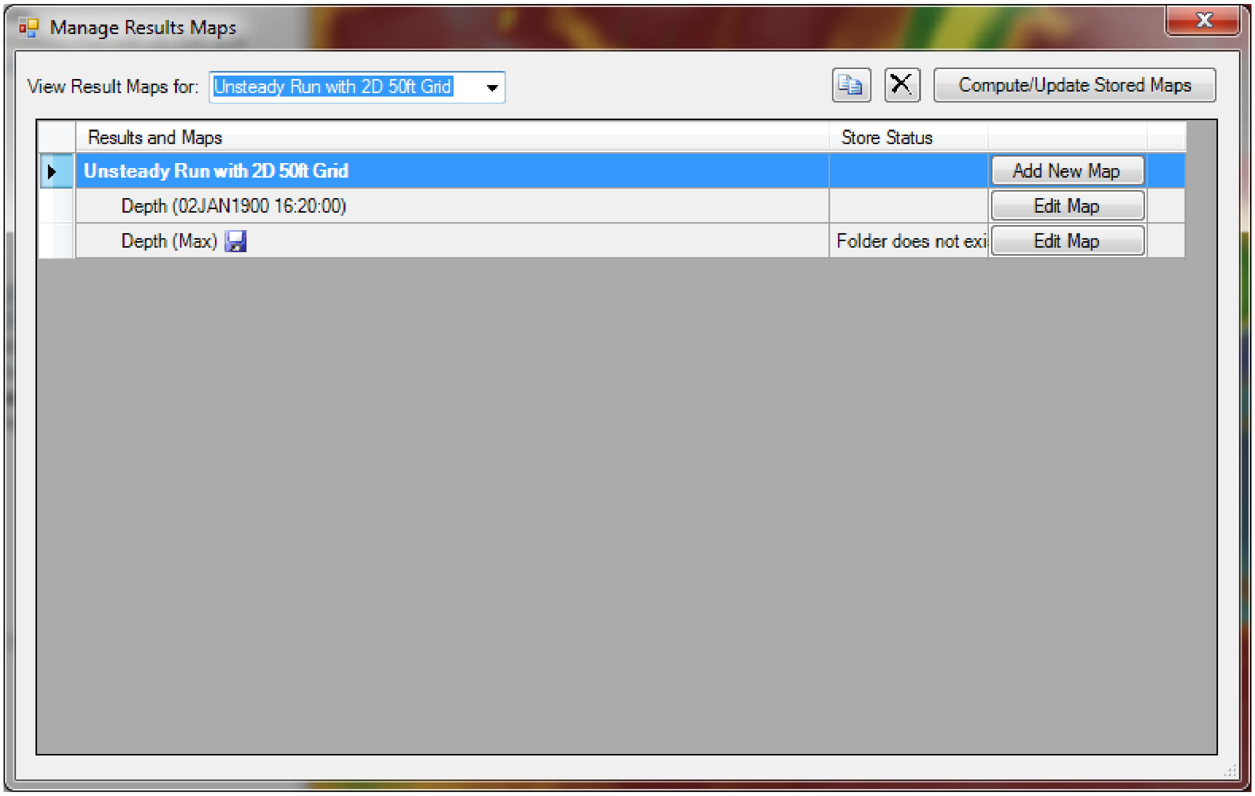


Figure 59. Results Mapping Window.

As shown in Figure 59, 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 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 you want to create a map from, this will bring up the window shown in Figure 60.

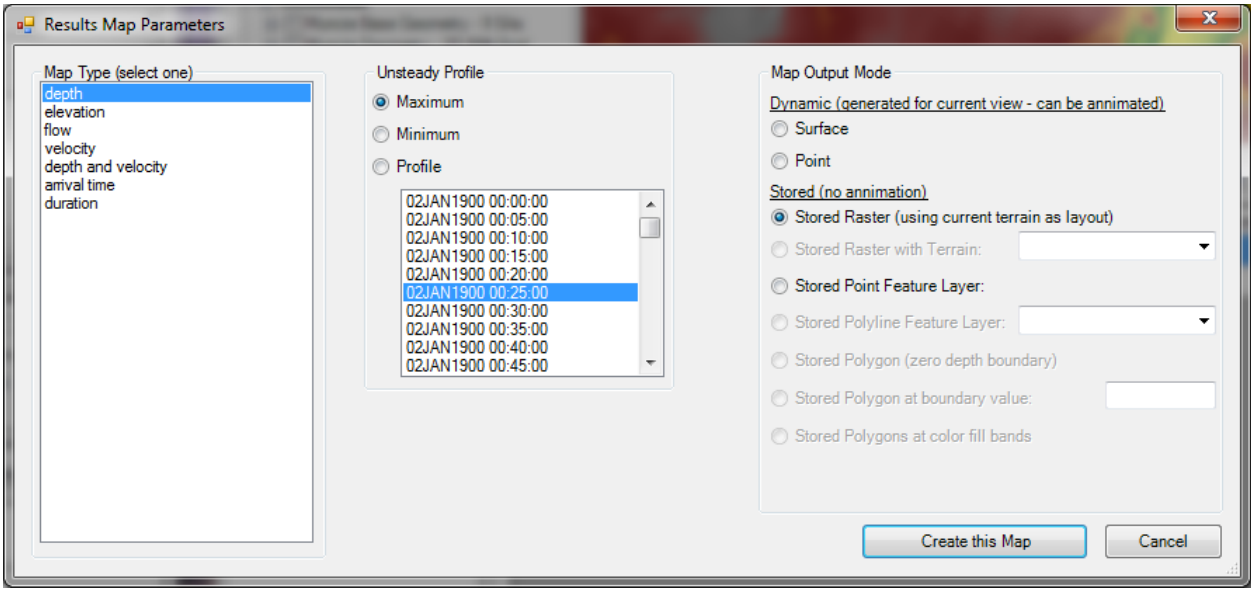


Figure 60. Add New Results Map Layer Window.

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 (no animation)" section**, then pressing the "**Create this Map**" button at the bottom of the window (see Figure 60). 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 "Folder does not exist", which means the stored map has not been created yet.

To create the stored map, press the button labeled "**Compute/Update Stored Maps**" (See Figure 59). This will start the process of creating/updating stored maps for all of the stored map layers that are out of date. When this process is complete, there will be a subdirectory within the project directory that is labeled the same name as the RAS Plan Name. 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. You can simply drag and drop the Geotiff files onto your 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". 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 you have more than one terrain grid for your 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 files and where they live spatially. If you drag that file over to a GIS, or import it, then it brings in all the tiles as a single collection in one layer, and you can have them all attributed the same.

E. 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 you must first click on the results map layer, to make it the active layer. When a results map layer is being displayed, and you click on that layer, the label will turn 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 you move the mouse pointer you will get the numerical value of that result displayed next to the mouse pointer. See the example below in Figure 61.

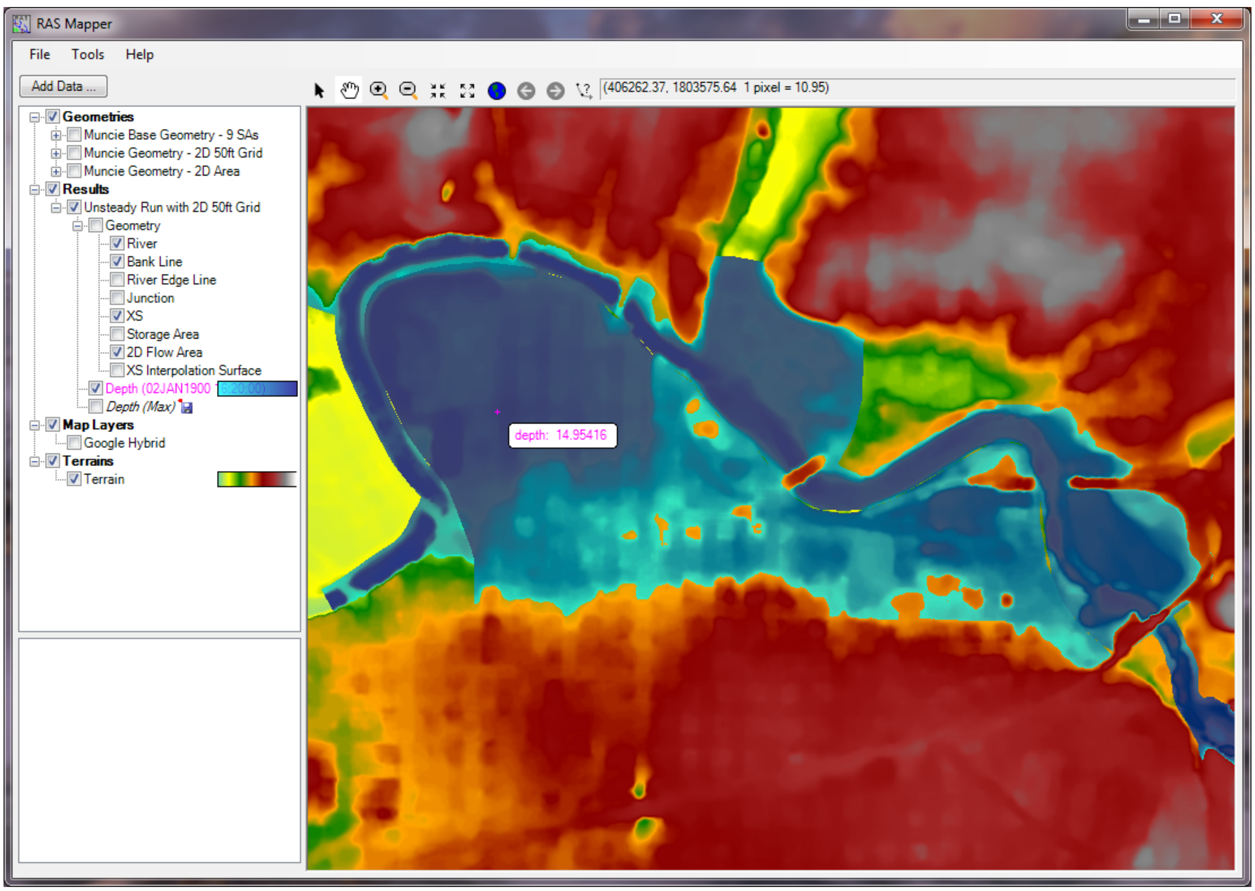


Figure 61. Example of Querying a value from the active map layer.

F. Time Series Output Plots and Tables

When a Results Layer is 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 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 layer and an option for plotting a “Water Surface Time Series” will be available in the popup window.

If 2DFlow 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 62) the following time series: Water Surface Time series; Cell Depth Time Series; 2D Face Point Velocity Time series (this is a point velocity of the closest Cell Face Point when selected); 2D Face Velocity Time Series (this is the Average Velocity across the Cell Face that is closest to the mouse pointer when selected); 2D Face Shear Time Series (this is the average Shear Stress across the cell face that is closest to the mouse pointer when selected); and the Property Tables (this is 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).

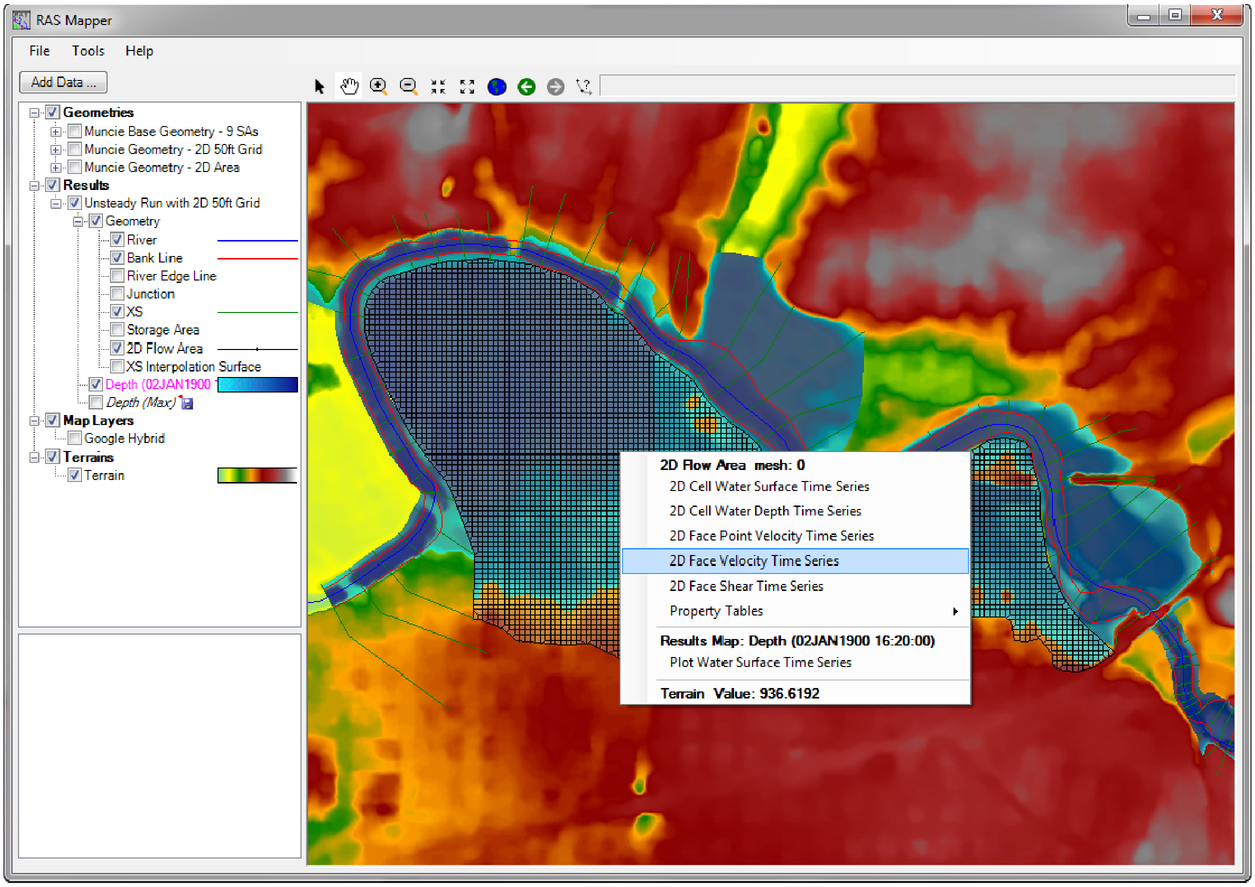


Figure 62. Example showing options for displaying 2D Model Output Time Series Results.

An example time series plot from RAS Mapper is shown in Figure 63.

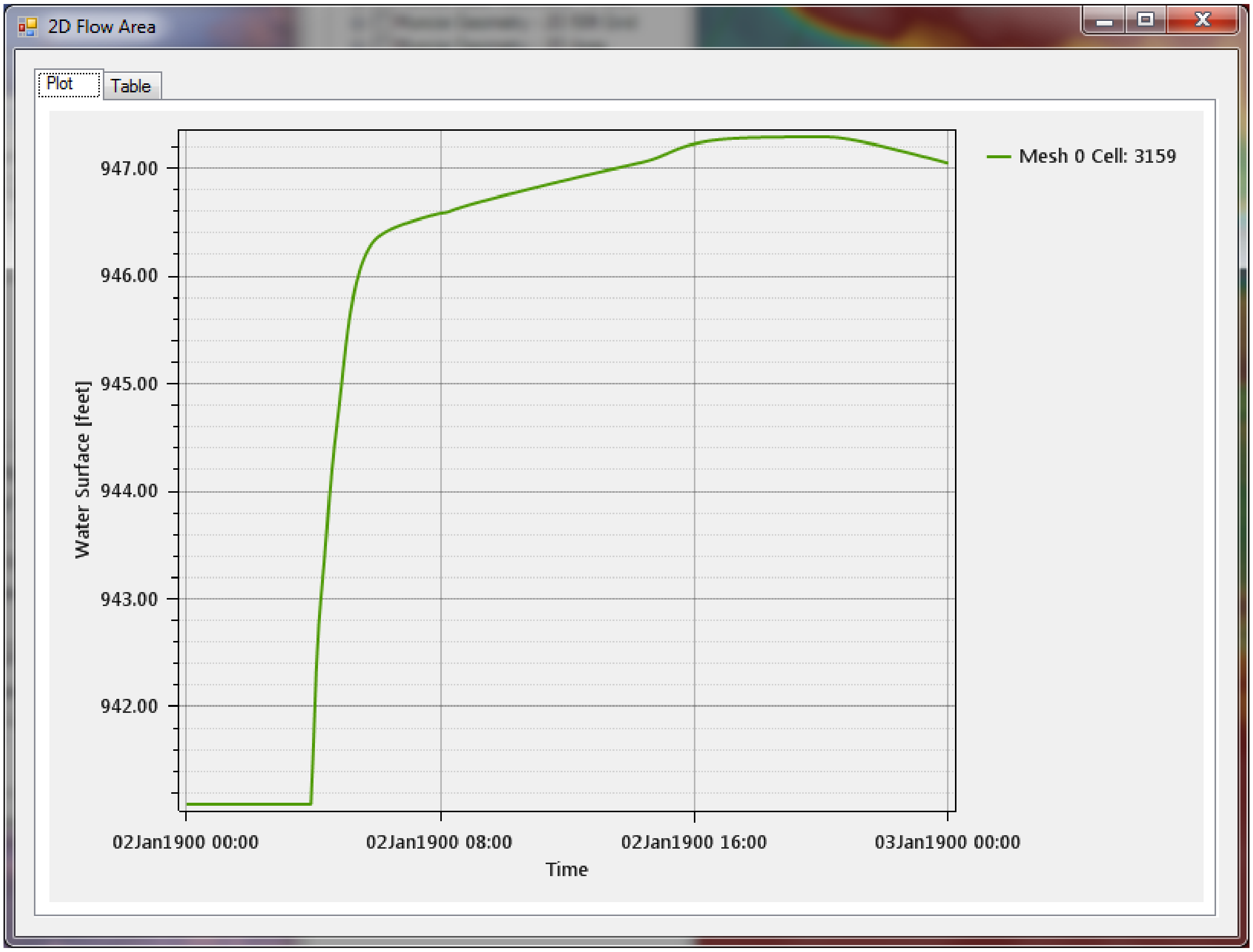


Figure 63. Example Time Series Hydrograph Plot from RAS Mapper.

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 **Cntrl-C** to copy the highlights 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 **Cntrl** 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.

G. Background Map Layers

HEC RAS Mapper has several options for bringing in other data layers/formats to be used as background maps below your 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 64 will appear.

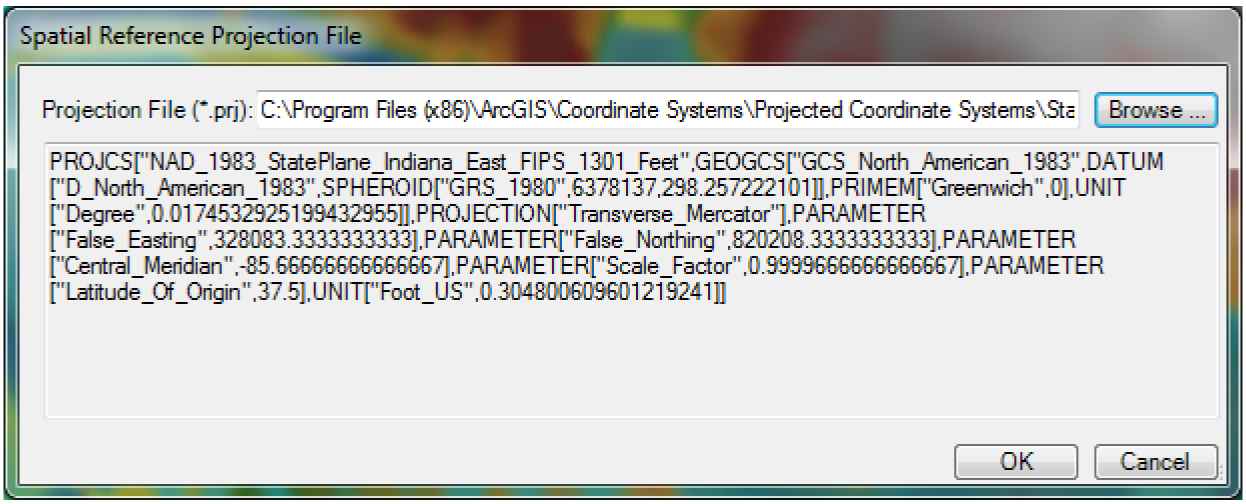


Figure 64. Editor to set the RAS project’s spatial reference system.

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 on your computer, you 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 lower stores a listing of all the available coordinate systems is listed in the “Projection File” text box, shown in Figure 64. For this example, “NAD 1983 State Plane Indiana East” was selected.

1. 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 65). 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 66.

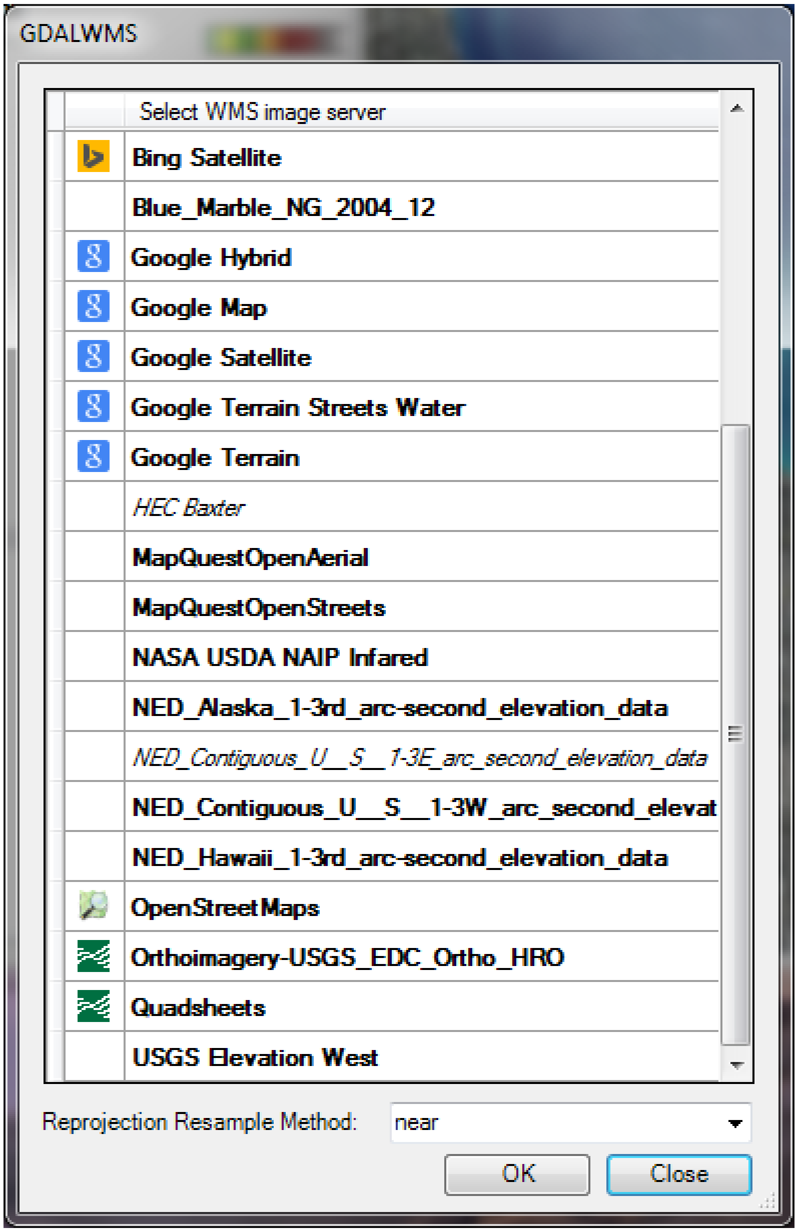


Figure 65. Web mapping services available in RAS Mapper.

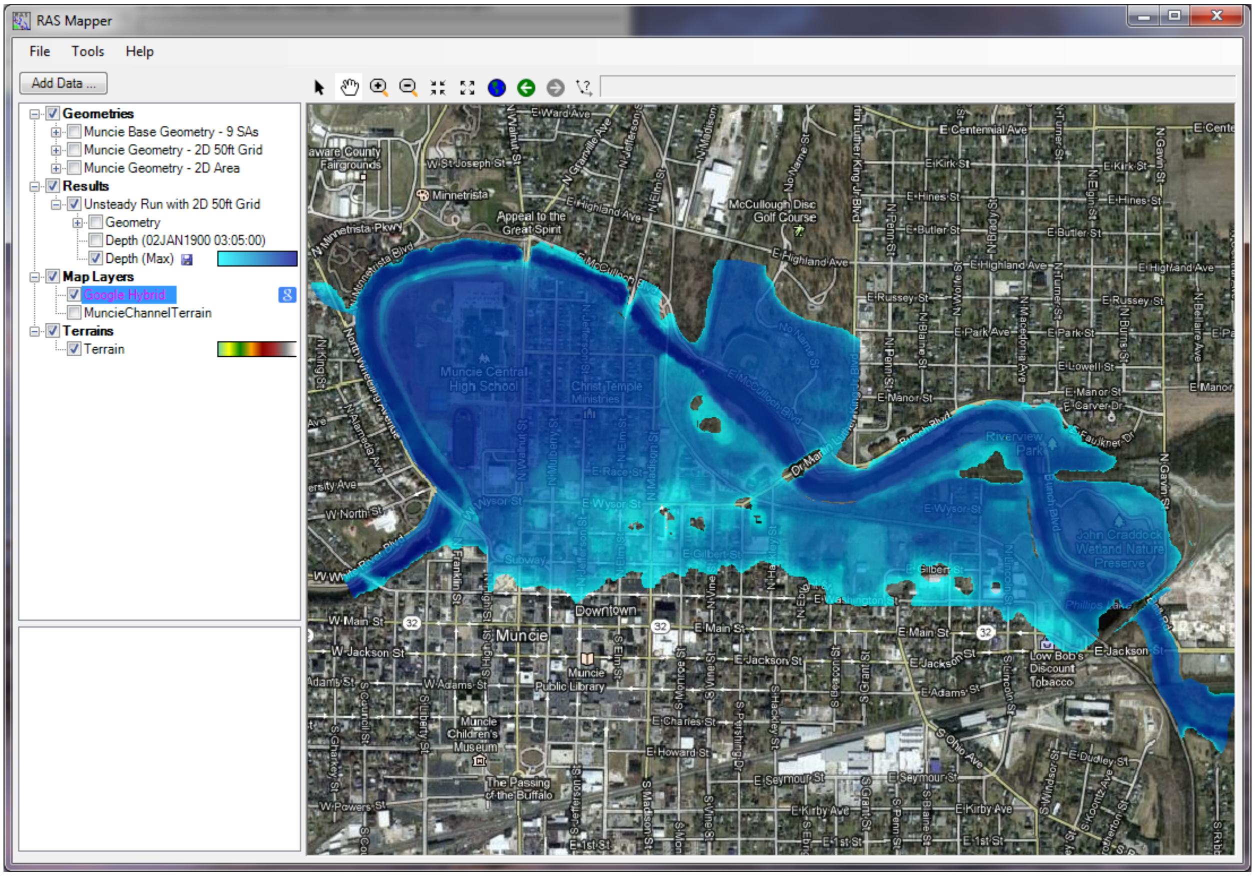


Figure 66. RAS Mapper with background Web imagery loaded with an inundation depth grid overlayed.

2. 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 Shape Files; 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 chooser window will appear, allowing you to navigate to the desired file and select it. See Figure 67 below:

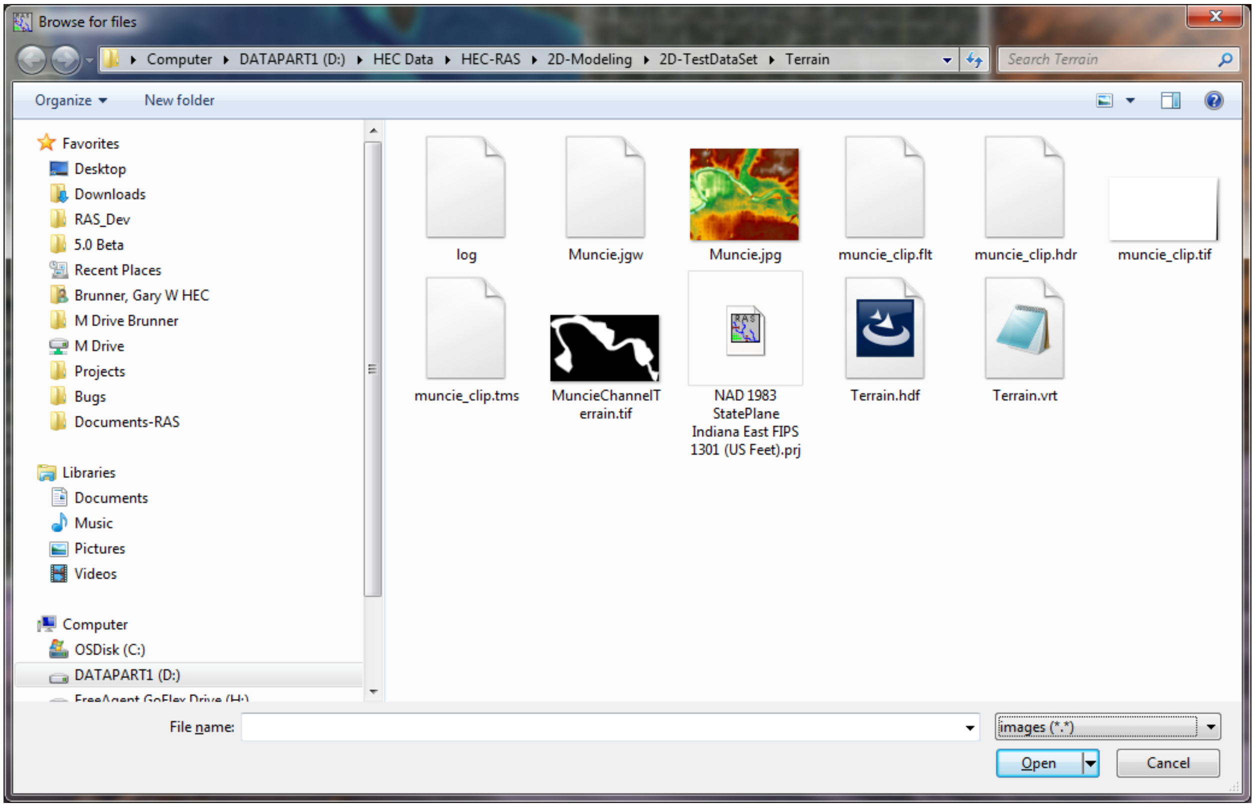


Figure 67. Example File chooser for bringing in Map Layers to be used for background display.

H. National Levee Database

The last tool to discuss is the link to the National Levee Database (NLD). If you select **Import NLD** from the RAS Mapper **Tools** menu, the software will automatically call the NLD and request a list of all the levees and floodwalls that are within the project area (the area you see on the screen when fully zoomed out). The NLD will send a list back to HEC-RAS and a window will appear on the screen with that list of levees/floodwalls (see Figure 68).

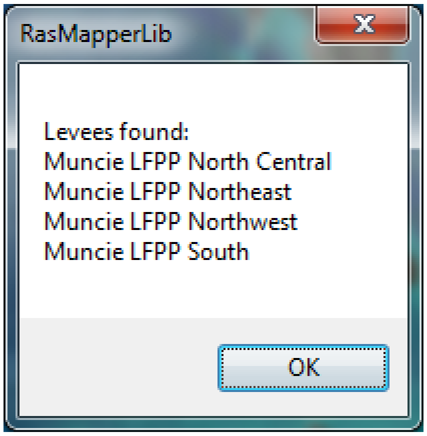


Figure 68. List of levees and floodwalls sent to RAS from the NLD.

If you press the **OK** button it will ask you 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 68. 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 yet (Sorry!!!).

VI. 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. 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).

There is a lot of output currently written to the HDF files that cannot currently be viewed spatially in RAS Mapper. For example, velocity information is computed for all the Cell Faces (Face Velocity), and the Face Points (Node X Vel, and Node Y Vel). To view and or use this additional output, 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/hdf-java-html/hdfview/index.html#download_hdfview

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 what is in there, display tabular data, and even plot results. Shown in Figure 69 is an example HDF file output from an HEC-RAS 1D/2D model run. As shown in Figure 69, the user can get to the Unsteady flow output for the 2D areas (as well as 1D objects) by drilling down through the **Results/Unsteady/Output/Output Blocks/Unsteady Time Series/2D Flow Areas/**, then click 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 what is available for 2D Flow Area is:

1. Depth: Depth of water in each of the cells (Feet or meters)
2. Face Shear: Average shear stress over the face (lb/ft^2 or $Newtons/m^2$)
3. Face Velocity: Average velocity at each cell face (ft/s or m/s)
4. Node X Vel: The X component of the velocity vector at a Face Point (ft/s or m/s)
5. Node Y Vel: The Y component of the velocity vector at a Face Point (ft/s or m/s)
6. Water Surface: Water surface elevation for each cell (feet or meters)

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 this 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).

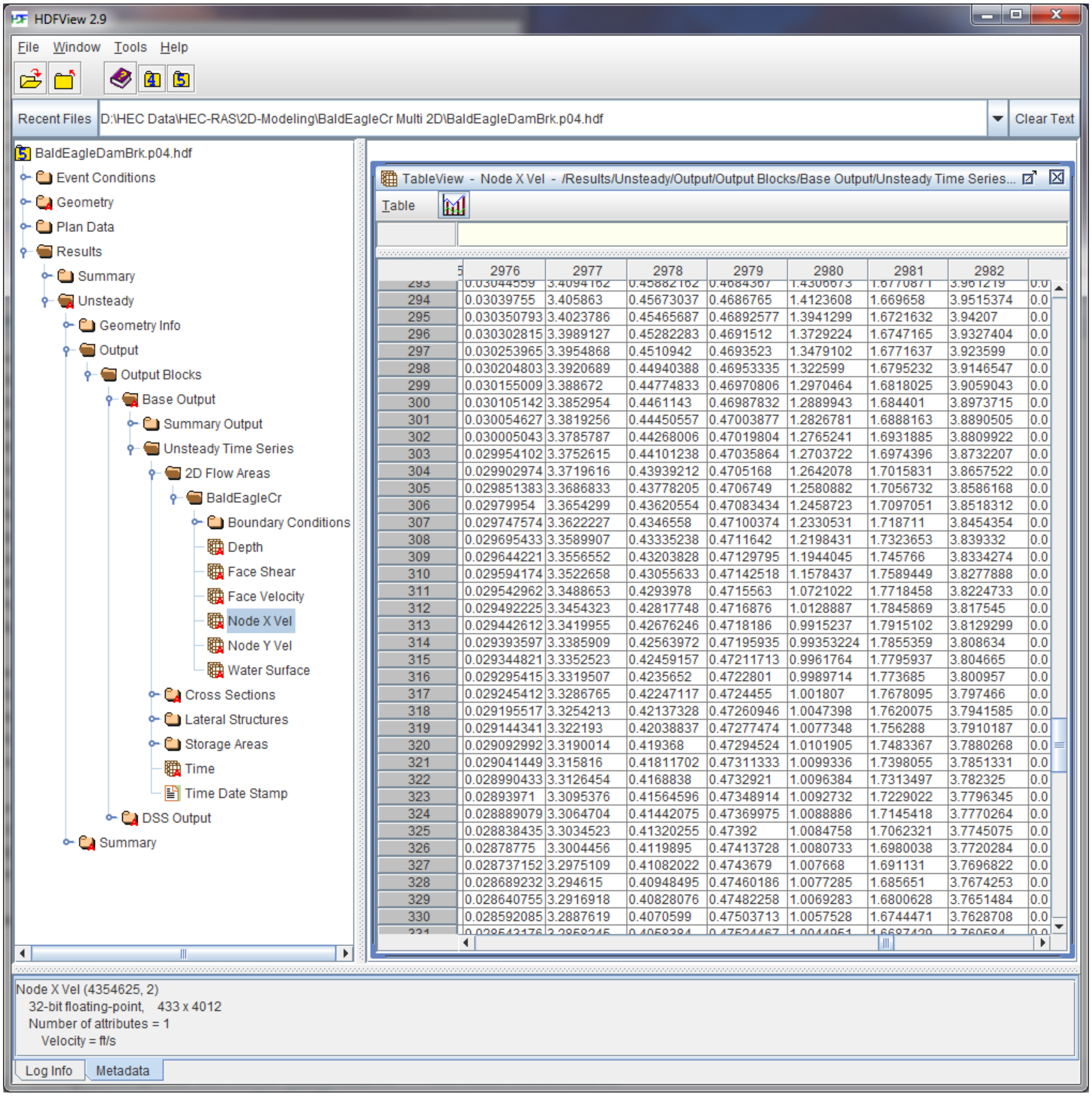


Figure 69. Example HDF File Output from HEC-RAS 1D/2D Model Run

Appendices

A. 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)