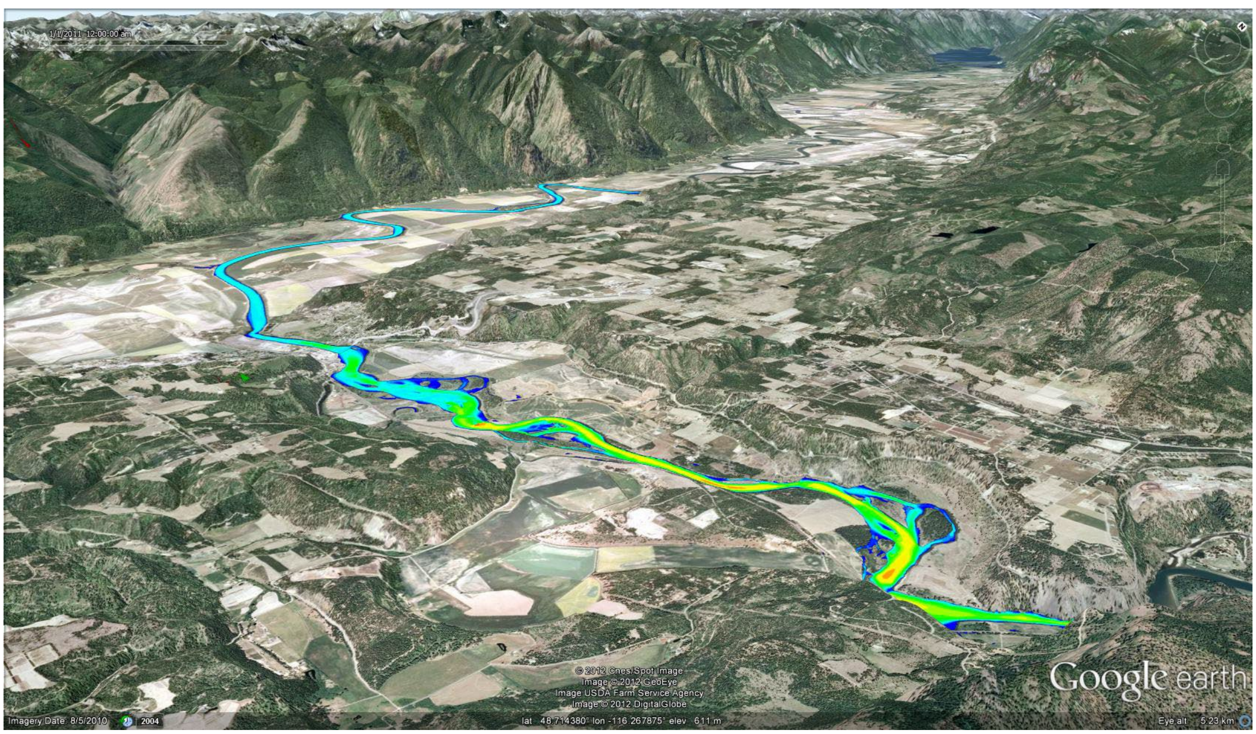
iRIC Software

Changing River Science

FaSTMECH Tutorial 1



Kootenai River Valley, Idaho, USA

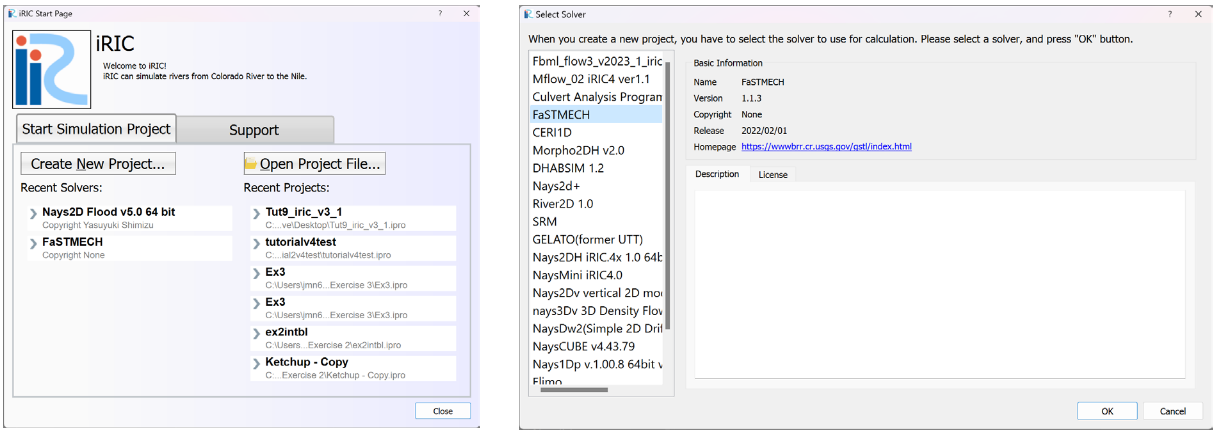
Contents

Introduction to the iRIC Graphical User Interface

Launch iRIC by selecting iRIC from the Program Menu list or double-clicking on the iRIC icon ( ) on the desktop. The iRIC Start Page (Figure 1A) opens and displays several options to start a project under the “Start Simulation Project” tab:

> **Create New Project**—Allows you to select the solver you wish to use from a list of solvers currently available in the application or to select the solver from a list of recently used solvers.
> **Open Project File**—Allows you to open an existing project using a browser window or to select from a list of recent projects.

The “Support” tab on the right provides links to the Home Page, Terms of Use, and Contact Information on the iRIC website.
Select the “Create New Project” button which opens the Select Solver dialog window (Figure 1B). Highlight FaSTMECH and click the OK button. An example of the resulting window, the main iRIC interface, is shown in Figure 2.



A B

Figure 1. Shows the iRIC Start Page (A) and the Select Solver dialog (B).

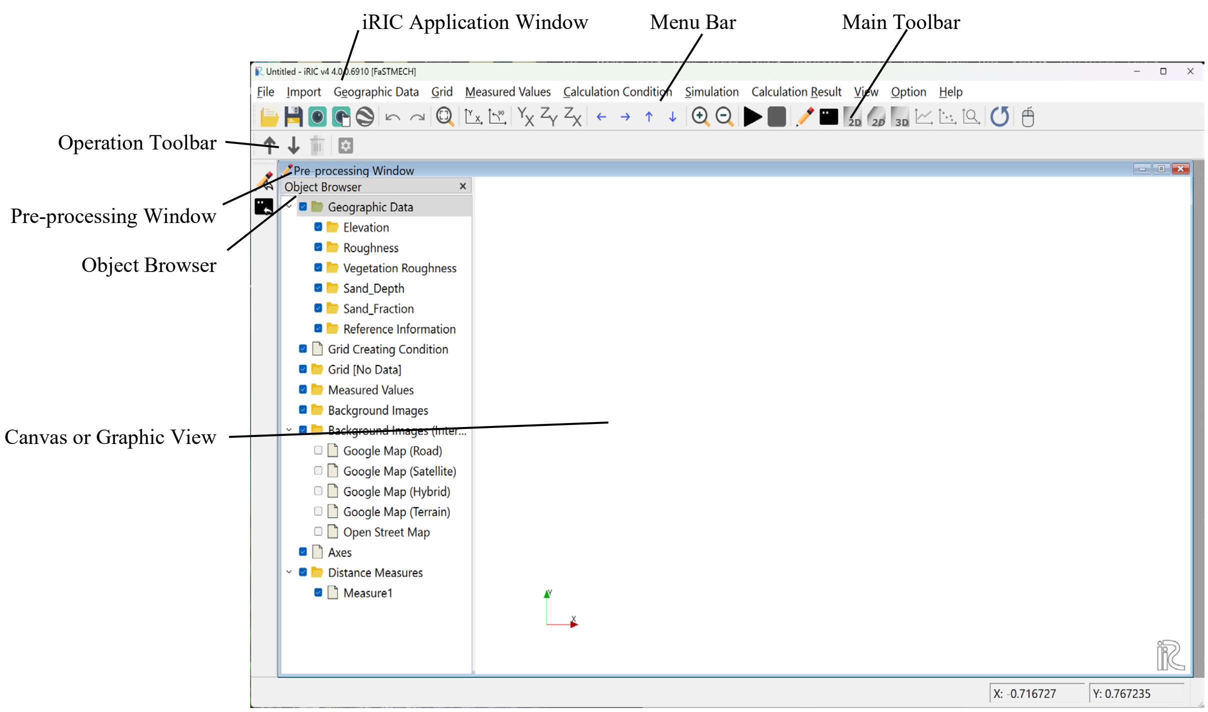


Figure 2. Example of the main iRIC interface.

The iRIC interface is operated using a variety of menus, toolbars, windows, and mouse operations. The **Menu Bar** at the top of the window provides access to the primary functions necessary to build a flow model including File Management, Data Import, Geographic Data, Grid, Measured Values, Calculation Conditions, Simulations, Calculation Results, View, Options, and Help. Additional items are added to the **Menu Bar** depending on whether a Pre-processing or Post-processing Window is active.

The **Main Toolbar** provides standard buttons to handle many of the same features such as opening and closing files, controlling the display screen, and opening other windows and graphs which will be explained in the section on

Post-processing. Some features are not enabled until a flow calculation is performed. See the table below for an overview of the features that are always enabled:

Main Toolbar Buttons:

- Opens an existing project
- Saves the project
- Saves a snapshot
- Undoes the action (the maximum number of undos can be set)
- Redoes the previous action (the maximum number of redos can be set)
- Zooms to the full extent of the data
- Resets rotation in the active canvas to original x-y orientation
- Rotates the active canvas 90 degrees counter-clockwise
- Shifts the active canvas display left
- Shifts the active canvas display right
- Shifts the active canvas display up
- Shifts the active canvas display down
- Zooms in
- Zooms out

The Pre-processing Window opens by default when iRIC is launched, or by selecting the Pre-processing icon ( ). The Pre-processing Window is used for features and functions related to importing data, editing data, generating computational grids, and editing computational grids. The Pre-processing Window has two parts: an Object Browser on the left and a graphic view or canvas on the right. The Object Browser in the Pre-processing Window allows you to control the display of geographic information such as elevation or topographic data as well as other datasets that may be necessary for your application. The Pre-processing Window also displays the computational grid and background images. The canvas displays the data selected in the Object Browser.

The Operation Toolbar provides a unique set of tools that are specific to the different branches of the Object Browser. The suite of tools available in the Operation Toolbar are loaded and become active depending on the branch of the Object Browser that is currently selected.

Mouse options allow you to pan (ctrl + left mouse button), zoom in and out (ctrl + mouse wheel), and rotate (ctrl + right mouse button) the canvas display. If you have a two-button mouse, often pressing and holding both buttons will allow you to scale the images.

iRIC provides a suite of tools for visualizing and Post-processing 2D model results. Map visualizations of model calculated flow characteristics are viewed in a 2D Post-processing Window (2D). Graphs of calculated flow characteristics along different grid dimensions or through time can also be generated using the Graph Window tool ( ). The Post-processing Windows and tools only become available when a simulation has been completed.

You will become familiar with the basic operation and workflow in the following three exercises. The exercises focus on:

- **Exercise 1:** The mechanics of Importing, Editing and Viewing measured data that will be used to initialize the model grid/mesh and can be used for model verification.
- **Exercise 2:** Building Grids and mapping/interpolating your measured data onto the grid.
- **Exercise 3:** Creating a simulation and editing the solver calculation conditions (parameters), running the model, and visualizing the results.

Following the exercises are a set of Tutorials that will lead you through practical applications of the FaSTMECH solver.

Exercise 1: Importing Data

This exercise provides familiarity with the mechanics of importing data that can be used to initialize the grid or for model calibration and verification. You will also learn how to visualize the data and edit the data if necessary. All the data and image files are in the *iRIC Tutorials\FaSTMECH\Tutorial 1* directory.

Import Topography

- The topography file is the most important piece of information required to build a numerical model of the river reach of interest. The topography can be imported by selecting **Menu Bar** *Import* → *Geographic Data* → *Elevation.*
- In the Select File to Import dialog, navigate to the following folder: *iRIC Tutorials\FaSTMECH\Tutorial 1\Exercise 1*. iRIC can import several different file formats; for this tutorial the file is a .tpo file which is on the default file list, so just select the following file: *r5finpt2m114_shifted.tpo* and click OPEN. This will open a Import Setting dialog that allows you to filter or reduce the number of points imported into iRIC, select the delimiter in the file, change the order of x,y, and z, and so forth. The skip feature can be useful if your data set is extremely large, but for our purposes leave the default setting at 1 and select OK to import the entire dataset.
- The Pre-processing Window now displays the topography data on the canvas and the data you imported appears in the **Object Browser** under *Geographic Data* **|** *Elevation* **|** *Point Cloud Data1* (Figure 3). In the **Object Browser** the topography can be visible or not by checking or unchecking the box next to *Elevation*.
- To adjust how the elevation points are displayed, select in the **Object Browser** *Geographic Data* **|** *Elevation* **|** *Point Cloud Data1* and then right-click to access a dialog that allows you to edit the data name in the Object Browser, Export the data, Delete the data, and adjust Properties. Select "Property", select "Points" and change the point size to 3 (Figure 4) to display the data as an array of points. Note that the tabs on this dialog provide access to color map and other graphical features (Figure 5).
- To change the range and intervals displayed in the data legend return to the properties menu under elevation as above, uncheck "Automatic" and change the maximum to 840. Select OK.
- Explore the Pre-processing window controls using the buttons on the **Main Toolbar** to zoom in/out and pan (See the Introduction). Try the mouse options that allow you to pan (ctrl + left mouse button), zoom in/out (ctrl + mouse wheel), and rotate (ctrl + right mouse button) the canvas display. Select to center the data in the Pre-processing window and, if you rotated the view, select to restore the original orientation.
- iRIC automatically generates a triangular irregular network (TIN) of the elevation data set. In the **Object Browser** right click on *Geographic Data* | *Elevation* | *Point Cloud Data1* and in the resulting pop-up menu select "Property" to open the Display Setting dialog that allows you to view the elevation data set as points, wireframe, or a surface (Figure 4). Experiment with viewing the data as wireframe and as a surface.
- Save by selecting *File* → *Save As File (*.ipro)* from the **Menu Bar**.

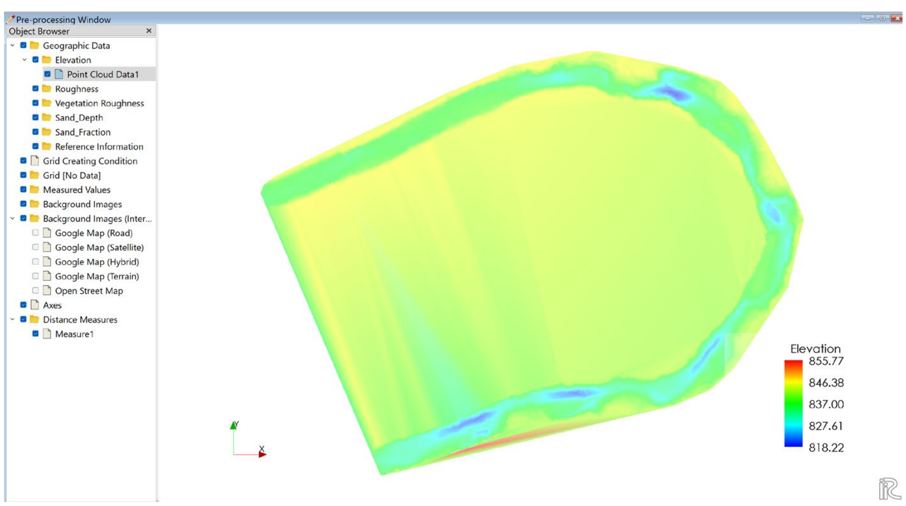


Figure 3. Pre-processing Window display of the elevation data.

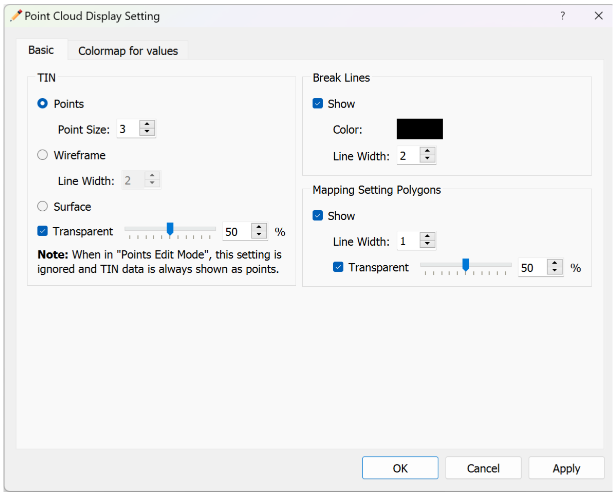


Figure 4. Display properties options for Elevation Data.

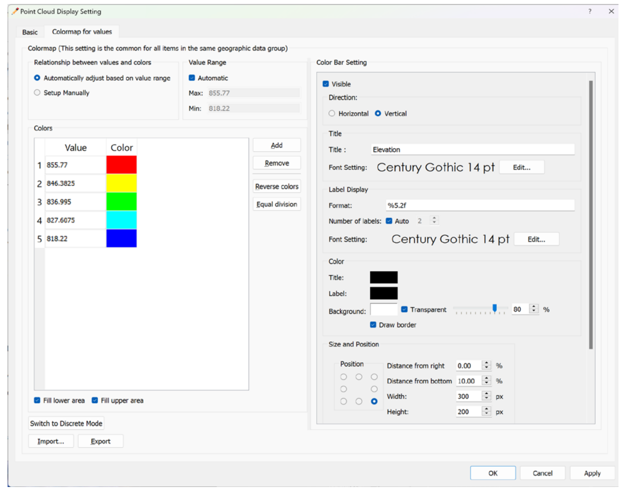


Figure 5. Display properties options for Elevation Color Setting.

Import Images

- Import images to place in the background of the data. Background images can be imported from the **Menu Bar** by selecting *Import* → *Background Image.* In the resulting Open Image file dialog select the file *r5img9.jpg* and then select OPEN. An error message may appear warning about the discretization, which can be ignored. To ignore, click OK to show the image in the Pre-processing window. Repeat this process and import a second image called *r5img10.jpg*, your screen should look like Figure 6. Save the file.
- Ideally any image you import will have a corresponding world file that positions the image correctly in space. If this is not the case you can adjust the image manually by rotating and scaling the image. See the User's Guide for more information.

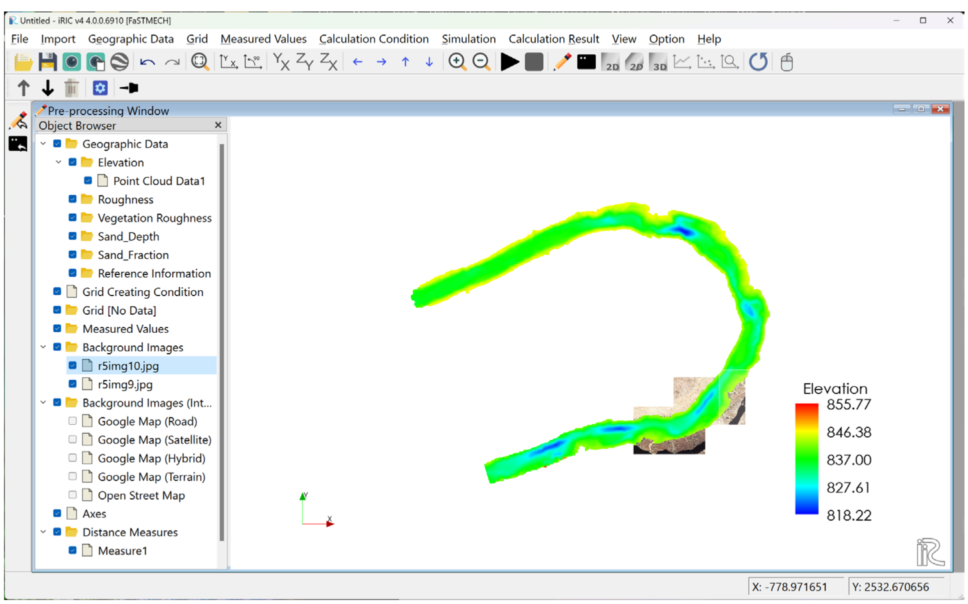
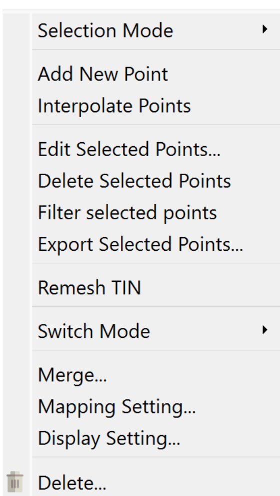


Figure 6. Surveyed topography and background image.

Simple Data Editing

- We would like to focus on the region of the data set encompassed by the two imported images. iRIC provides some simple tools for editing the data which we will introduce here. To delete the data outside of the two images, select in the **Object Browser** *Geographic Data | Elevation | Points Cloud Data1*. In order to edit points, right click in the Preprocessing window, scroll down to Switch Mode and select points editing mode. This will allow direct editing of the points data in the window.
- To select a rectangle of points, right click in the Preprocessing window and make sure the Select Points with Rectangle Region is selected (selected by default), then left-click the mouse and drag a rectangle while holding the mouse down, releasing the mouse to finish. When points are selected, right-click anywhere on the screen to bring up a pop-up menu of actions to apply to the selected points (Figure 7). The selected points can be deleted, deleted above, or deleted below a user specified threshold. In this case we want to delete the points outside of the two imported images to end up with points and images as displayed in Figure 9. Thus, you'll need to draw a rectangle with your mouse to encompass the points outside the images and then select "delete selected points". This will take more than one selection. Remember to use undo if you make a mistake while editing your data points. There is another editing menu for TINs, as shown in Figure 8.
- More information on the data editing tools can be found in the User's Guide.



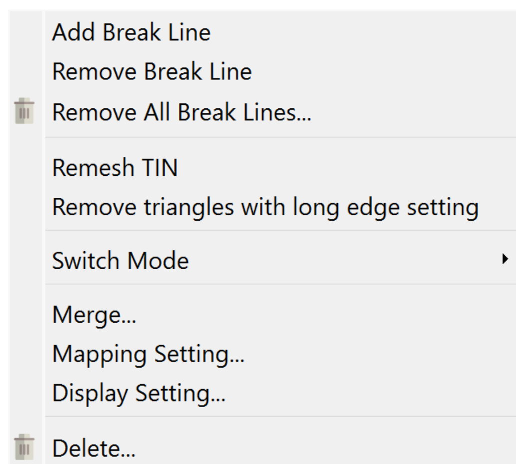


Figure 8. Menu for actions applied to Tin

Figure 7. Menu for data editing tools available for point data sets.

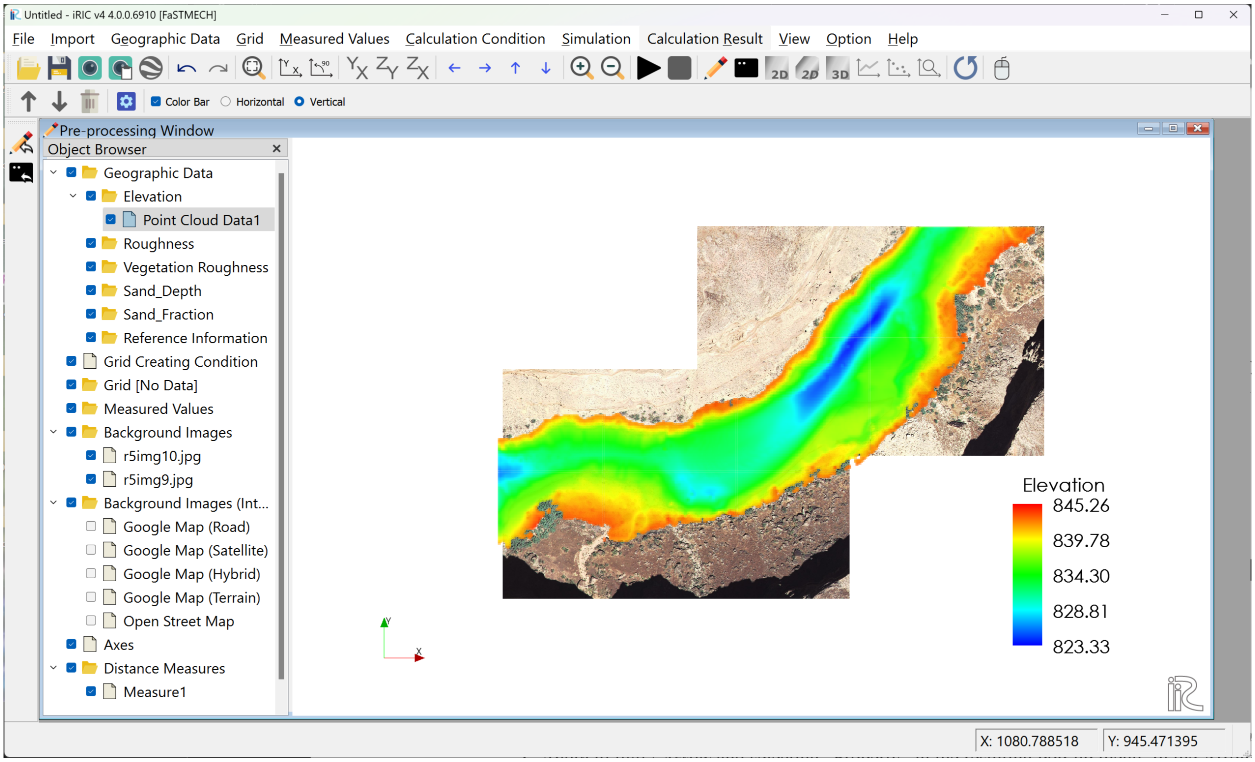


Figure 9. Image and elevation points following data point editing.

Import Data for Model Calibration and Verification

Import Measured Velocity Data

- Import data that can be used to calibrate or verify model predictions. Import measured velocity data from the **Menu Bar** by selecting *Import* → *Measured Values.* Select *EM_mar7a_adcp shift.csv* and click OPEN.
- Adjust the length of the imported vectors in the **Object Browser** by right-clicking on *Measured Values* | *C:\(path to file)* | *Arrow* and selecting "Property" in the resulting pop-up menu. In the Arrow Setting dialog set the values to those in Figure 10 below.

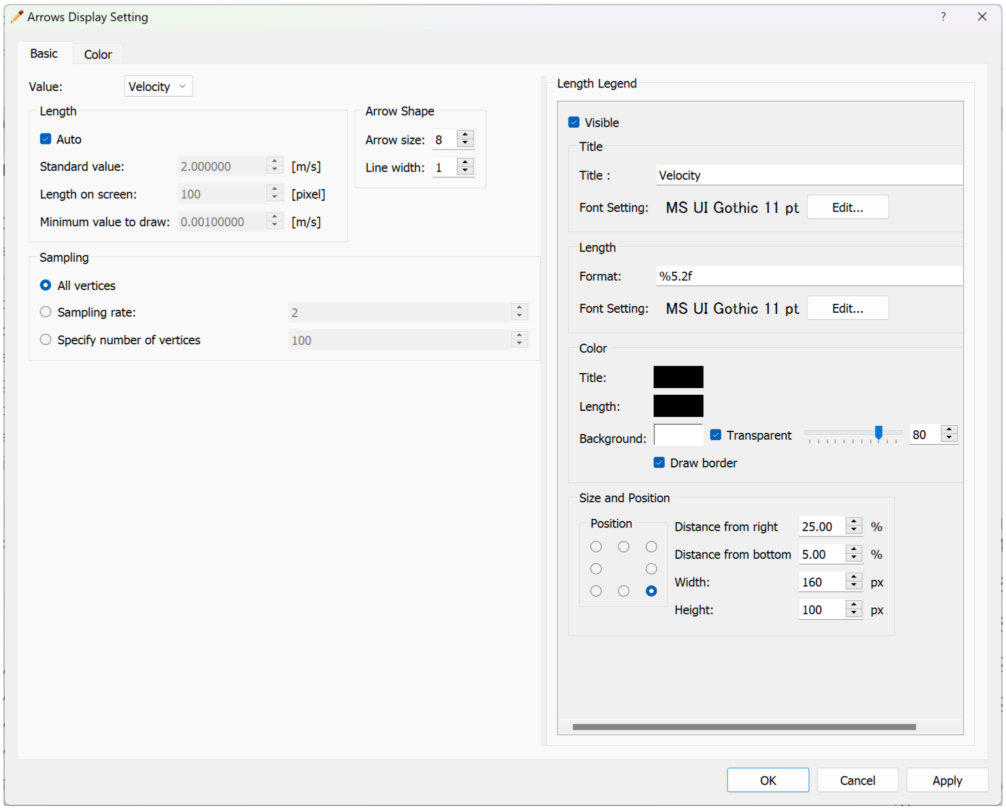


Figure 10. Arrow setting to set measured vector properties

Import Measured Water-Surface Elevations

- Import measured water-surface elevation data from the **Menu Bar** select *Import* → *Measured Values.* Select *3_10_08_wse.csv* and click OPEN.
- Adjust the size of the points used to represent the water-surface elevations by right-clicking in the **Object Browser** *Measured Values* | *C:\(path to file)* | *Scalar* and selecting "Property" in the resulting pop-up menu. In the Scalar Setting dialog set the Point Size property to 5 and then select OK.
- The elevation scalarbar can be hidden by right-clicking *Geographic Data* | *Elevation* and selecting "Set Up Scalarbar…", then unchecking the "Visible" box. Click OK.
- The resulting view with images in the background and measured water-surface elevations and flow velocities are shown in Figure 11.
- Save the project.

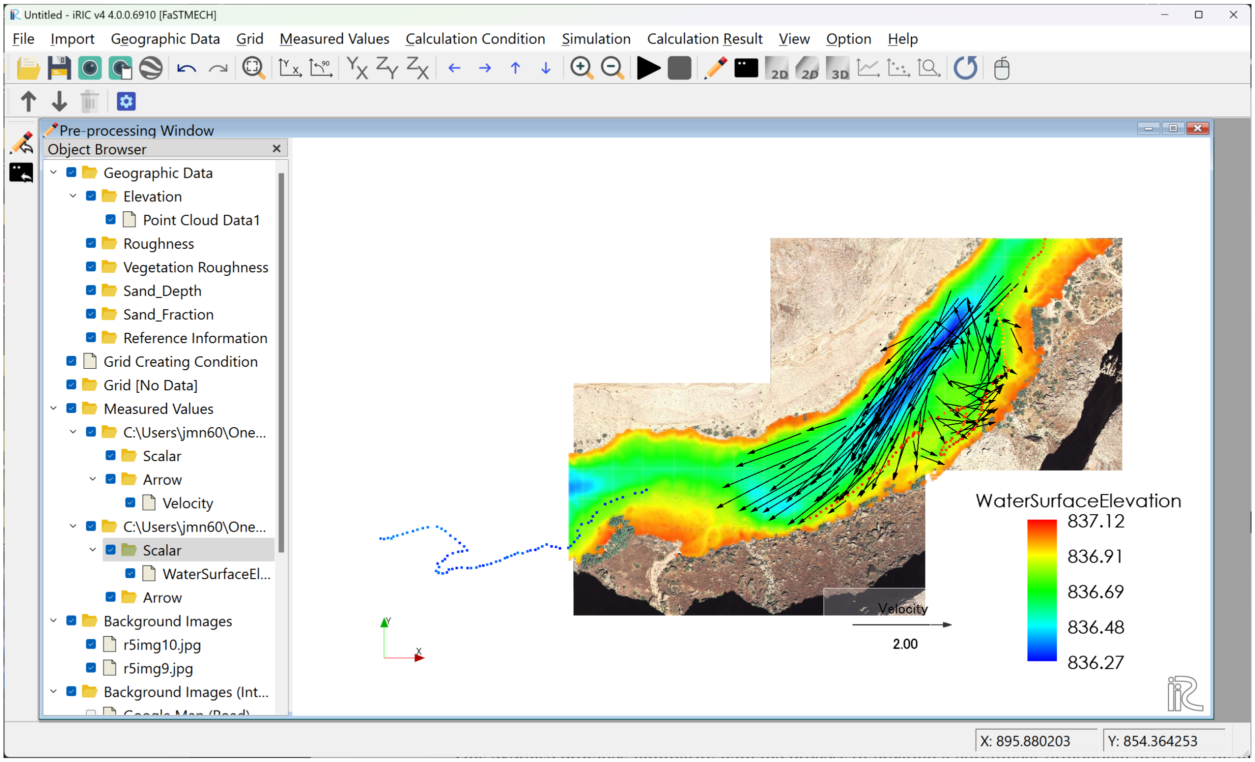


Figure 11. Measured velocity vectors and water-surface elevations scalar values shown on top of georeferenced images.

Exercise 2: Grids

This exercise provides familiarity with the process of creating a curvilinear orthogonal grid used by the FaSTMECH solver in iRIC. This exercise assumes Exercise 1 has been completed and you have some basic skills importing data and working with the **Object Browser** to view data.

- First create a new FaSTMECH project and import the *KootMeanderShift_filtered3.tpo* topography file in the *Tutorial 1\Exercise 2 folder*. This is a large file of 2.1 million points. If you have a computer with 4GB or less memory or you have an integrated graphics board rather than a separate graphics board, you may want to filter the data. A good choice for filtering the data is 10, importing every 10th point in the data set. You'll also need to use the Import Setting dialog to set the delimiter to "space" and skip one header line. Check the sample screen in the dialog to judge when the importation is correct.
- Import the *Meander2.jpg* file (Figure 12).
- Take a look at the data to familiarize yourself with the reach. The Kootenai River is located in Northern Idaho, USA. The flow direction is from bottom to top. The mean width of the river is approximately 200 meters. However, there is an island towards the top of the reach and the width of the two channels plus the island increases to approximately 800 meters.

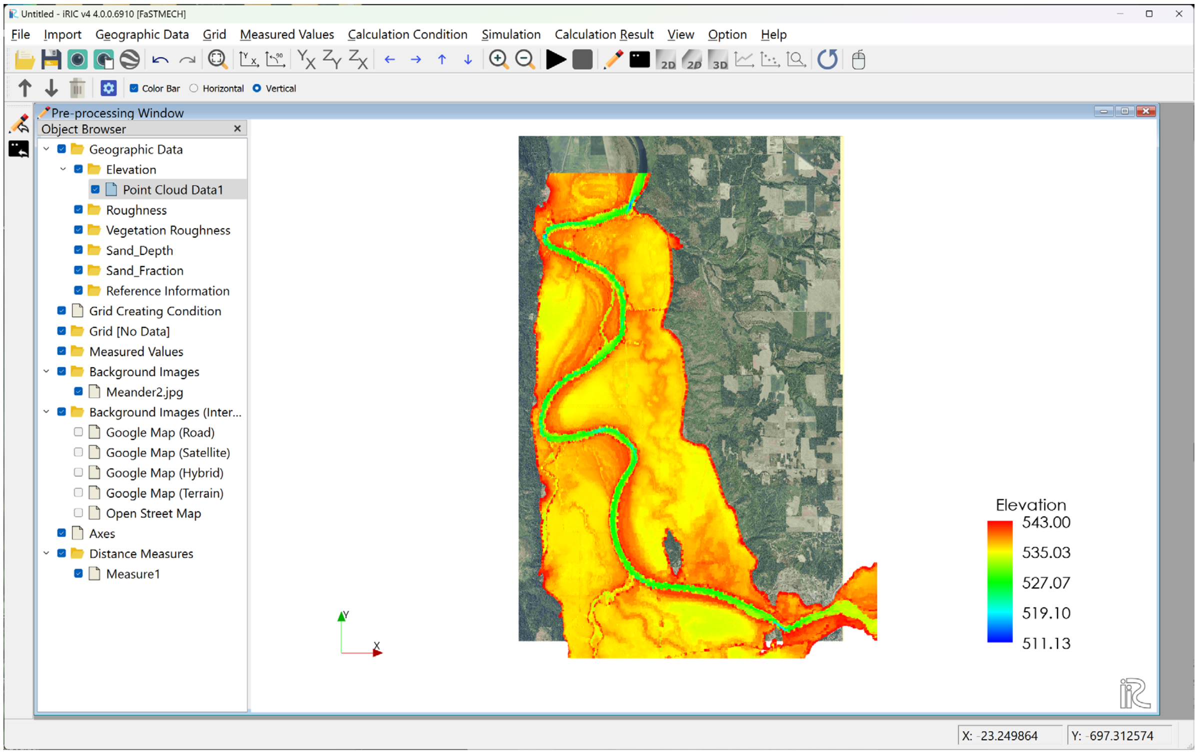


Figure 12. Kootenai River topography and image.

Creating a Numerical Grid

iRIC contains a number of different methods to generate grids for the many different iRIC solvers. The FaSTMECH solver uses a structured curvilinear orthogonal grid. This type of grid can be created in three basic steps: define the grid centerline, specify the width and density of points in the grid, and refine the curvature and location of the grid until a satisfactory result is achieved. The result of the process will be a channel-following grid that looks something like that in Figure 13.

Create the Curvilinear Grid:

- In the **Menu Bar** select *Grid* → *Select Algorithm to Create Grid.* In the Select Grid Creating Algorithm dialog (Figure 14), select “Create grid from polygonal line and width”. Note that a brief description or instructions for creating the grid are given in the Description pane. Click OK. Another dialog will open providing further instructions on using this function. Click OK.
- To draw the centerline, click the left mouse button in the desired locations starting at the upstream most point of interest and ending at the downstream end of the reach. Figure 15 shows the start of this process. Note that the centerline is always drawn from upstream to downstream, and Upstream and Downstream labels are affixed to those points as they are drawn. In this case flow is from lower right to upper left. At any point in the process of drawing the centerline you can use the Pan (Ctrl+Left Mouse) and Zoom (Ctrl+Middle Mouse) functions without affecting the placement of the grid centerline. In other words, using the Ctrl key to affect a Pan or Zoom will interrupt the centerline process but not break it. When finished press “Enter” on the keyboard.
- The Grid Creation dialog allows you to specify the number of nodes in the streamwise direction, n_I, the number of nodes in the cross-stream direction, n_J, and the width of the grid, W (Figure 16). Set the grid width to 800 meters and define the number of points in the streamwise and cross-stream dimension to give corresponding increments of about 20 meters (displayed by d_I and d_J)). Use the “Apply” button on the

dialog to dynamically view of the result of your grid parameters incrementally to find the desired spacing of nodes in the streamwise and stream-normal directions. Select Create Grid when you are done.

- In this case it is likely that you will get the following warning, "Grid shape is invalid. Modify grid creating condition, and try again." Select OK to dismiss the warning. This occurs when the streamline curvature of the grid centerline is high relative to the width of the grid and results in the grid overlapping on itself as in Figure 17.
- A Confirmation message "Do you want to map geographic data to grid attributes now?" follows. In this case we will decline by selecting "No". We want to modify the location and curvature of the grid which is likely in this case to take many iterations. To disable the automatic mapping of geographic information from the **Menu Bar** select *Grid* → *Attributes Mapping* → *Setting* and in the resulting Grid Attribute Mapping Setting dialog, select "Manual" for the Execute Mapping property. Select OK when done.
- To adjust the centerline to better fit the grid to the data or as in the case here, adjust the curvature of the grid to remove overlapping nodes. In the **Object Browser** select *Grid Creating Condition*. This is necessary to edit the grid. The centerline and points defining the centerline should be visible. When the mouse is placed over the centerline or over a centerline point, the mouse cursor changes to an open hand, and if the left mouse is clicked and dragged, it will move the centerline or centerline point with the cursor as a closed hand. Experiment with adjusting centerline points to remove the overlapping grid nodes. If you make a mistake Ctrl+z will undo and Ctrl+y will redo the previous action. Continue adjusting the centerline until there is no overlap of the grid, as seen in Figure 18. Additional utilities to add or remove points are also available and will be discussed in Tutorial 1.
- Save the project.

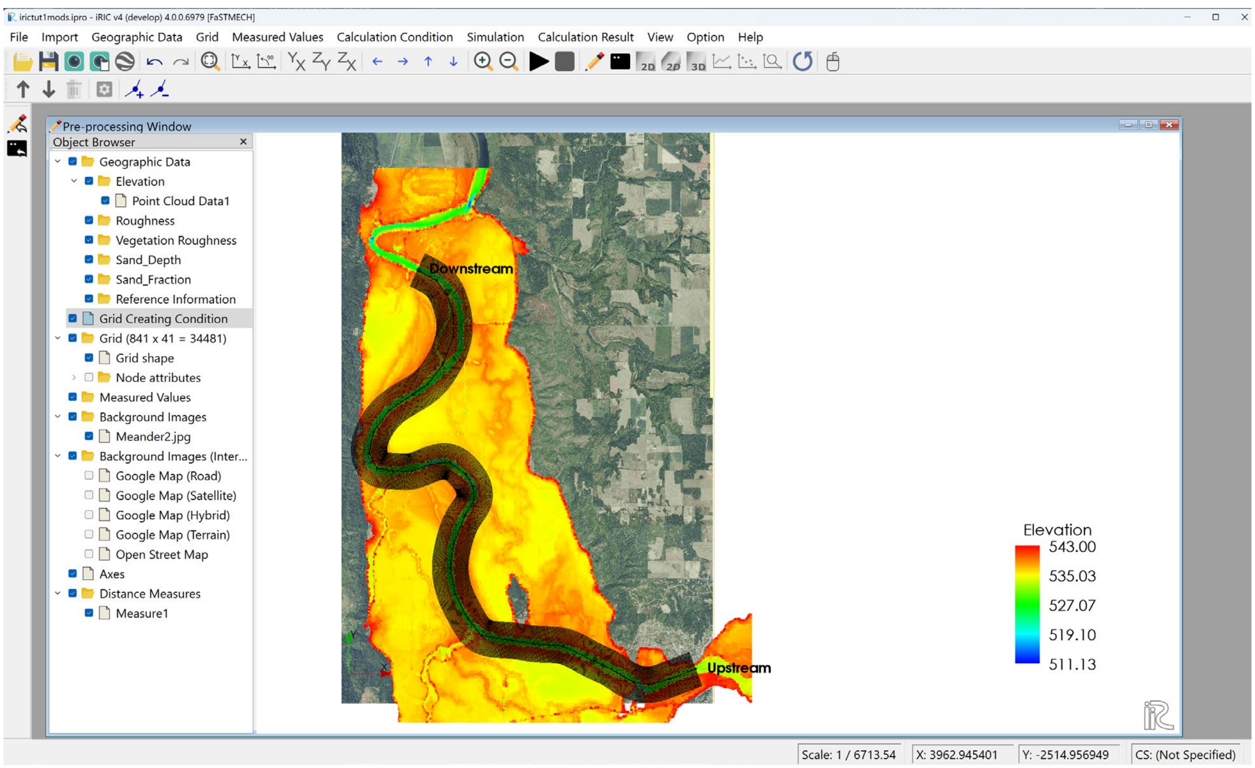


Figure 13. A user-defined grid following the channel of the Kootenai River.

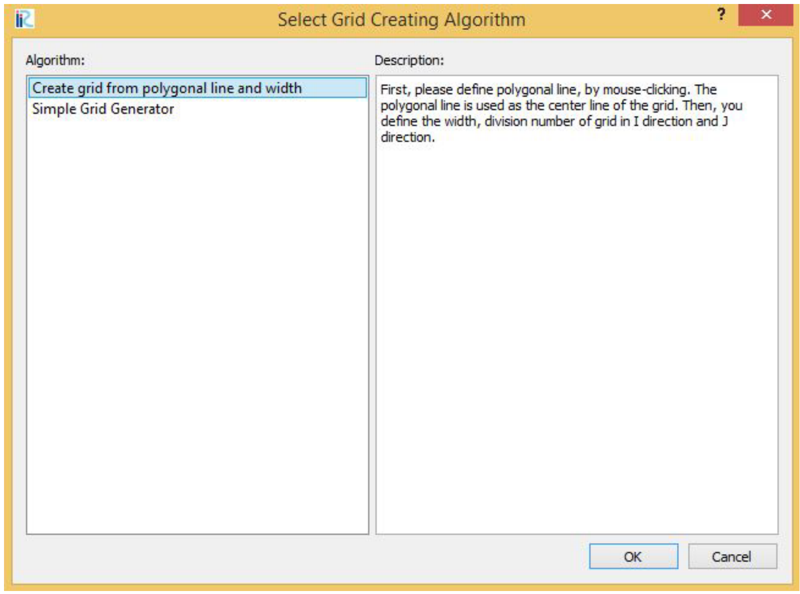


Figure 14. Select Grid Creating Algorithm dialog.

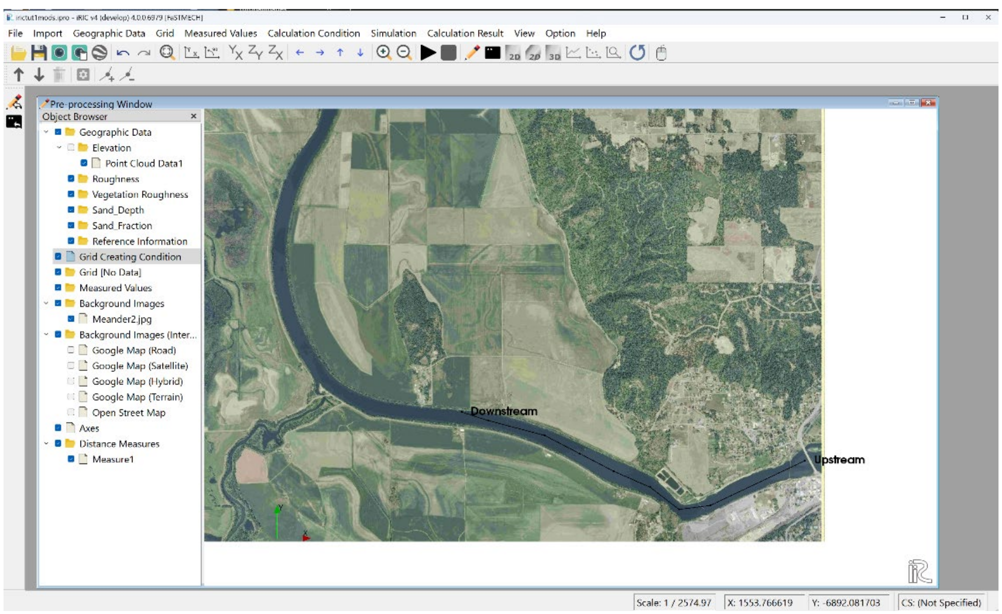


Figure 15. Process to draw the grid centerline. Note the Upstream and Downstream labels are defined.

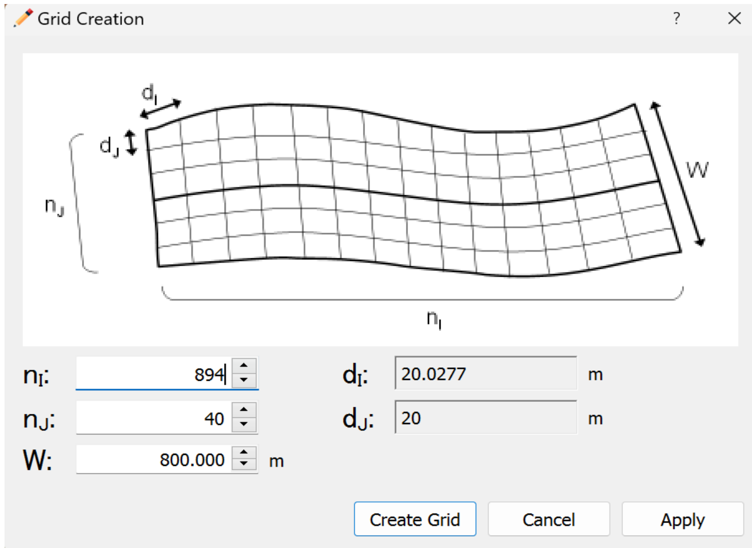


Figure 16. The Grid Creation dialogue.

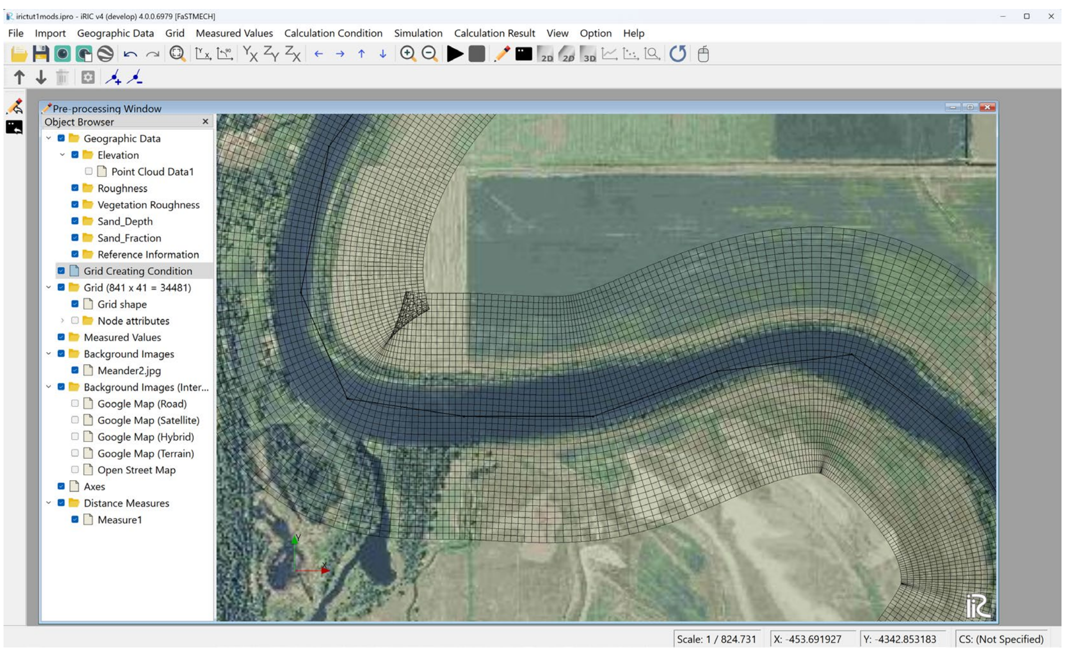


Figure 17. Curvilinear grid with grid overlap.

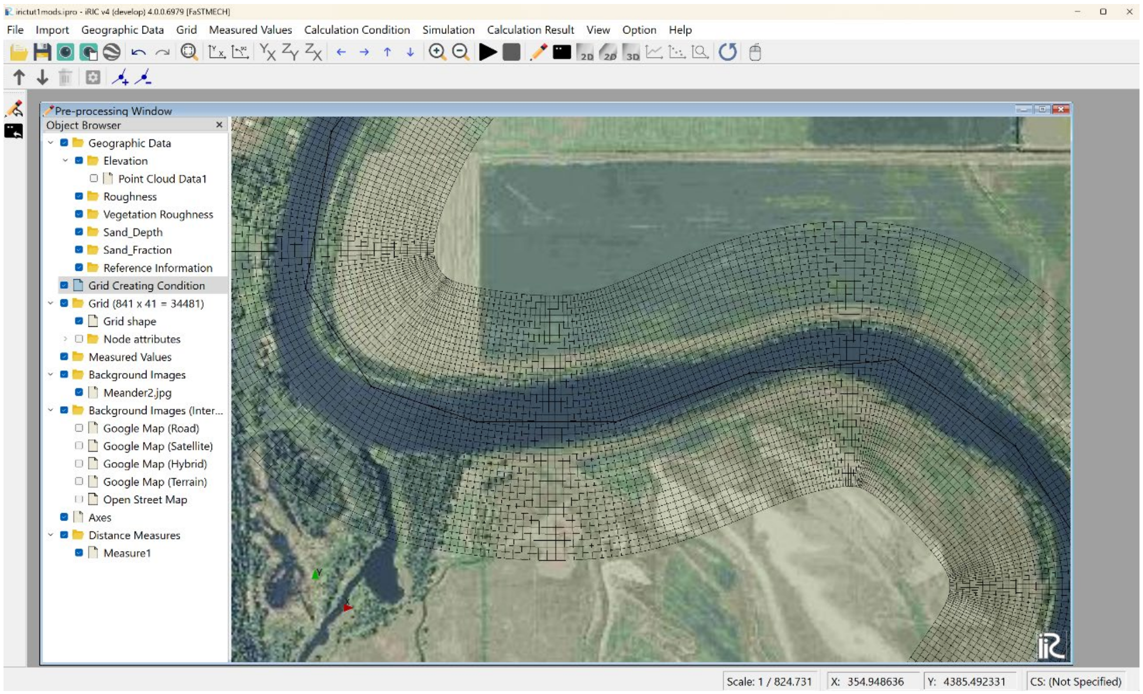


Figure 18. Grid after editing the location of centerline points to refine the curvature and eliminate overlapping grid nodes as shown in Figure 17.

Mapping Geographic Data To The Grid Attributes

In the **Object Browser**, most branches in the geographic data tree have a corresponding branch in the *Grid | Node attributes*. Imported data into each branch of the Geographic Data, such as *Elevation* in this case, is interpolated to the grid by one of several methods described below. The FaSTMECH solver has several branches under the Geographic Data but only Elevation is required. The possible use of the other branches is discussed throughout the Tutorials. Once you are satisfied with your computational grid, you can map or interpolate measured elevations to each node of the grid. There are two algorithms to do this. See the User's Guide for a more detailed explanation of each.

1. The first uses a triangular-irregular network (TIN), a surface defined by a set of contiguous, non-overlapping triangles generated by a Delaunay Triangulation of the imported data. The value at each node of the grid is determined by finding the triangle that contains the grid node and linearly interpolating the value based on the values of the three vertices of the triangle.
2. The second is based on a nearest neighbor approach that utilizes a template with a user-defined width and length, where the length follows the local curvature of the grid. Interpolated values are assigned the inverse distance weighted average of all measured points in the template.

Mapping With TINs

- In a previous step above we set the *Grid* → *Attributes Mapping* → *Setting* to "Manual". A setting of "Auto" would result in the application mapping the geographic data after every edit to the grid. The "Manual" setting now allows us to choose when to map the geographic data. Do that now by selecting from the **Menu Bar** *Grid* → *Attributes Mapping* → *Execute*. Check "Elevation" and select OK. Select OK when notified that mapping is complete. Node attributes will be added to the Object Browser in *Grid | Node attributes*. Expand the Node attributes and make sure that *Elevation* is selected so you can view the results (Figure 19).

- Make sure in the **Object Browser** that *Geographic Data | Elevation, Grid Creating Condition,* and *Grid() | Grid shape* are all unselected to view the results as shown in Figure 19.
- Save the Project.

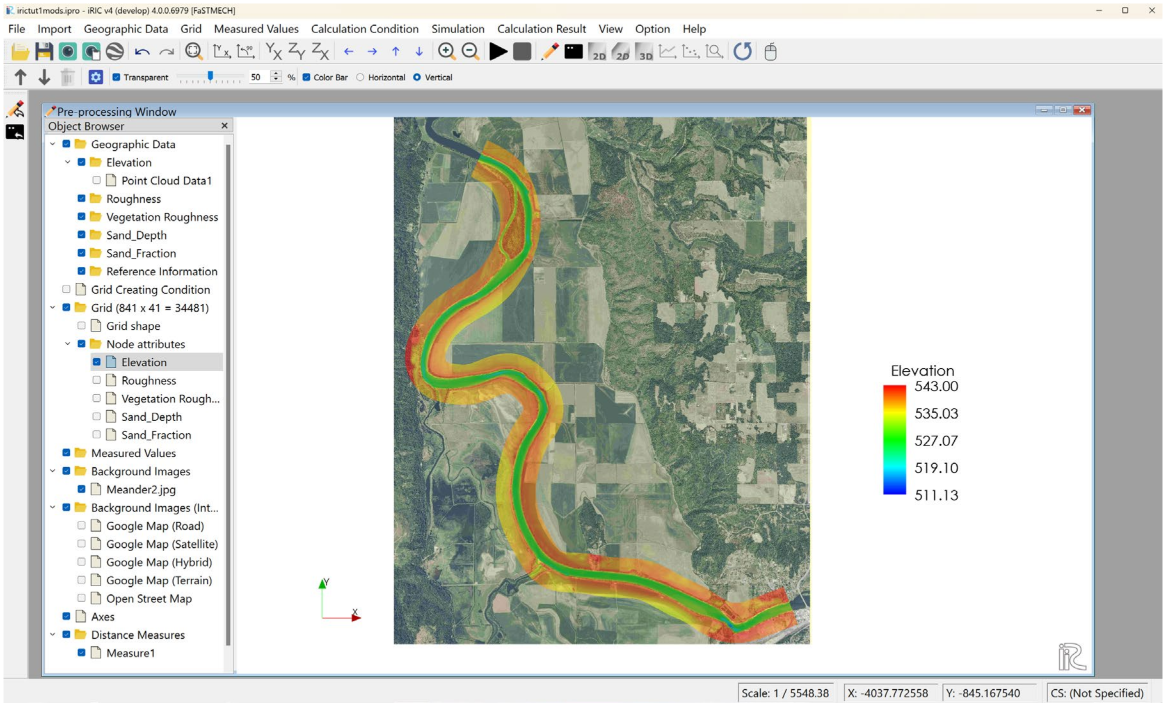


Figure 19. Elevation mapped to the grid using the TIN algorithm

Mapping With the Template Method

To illustrate the template method, we'll open a new FaSTMECH project with elevation data that was dominantly collected as cross-sections. In this section you will also learn how to import a non-georeferenced image and stretch, rotate, and translate the image so that it fits the measured topographic data

For this exercise you have two options. Option 1) Start from Part 1 below and learn how to pan, stretch, and rotate an image to fit your data or Option 2) Open an existing project with topography and background image already fit to the data.

Option 1 (Fitting a background image to the data set and mapping with a template)

1. From the **Menu Bar** select *File* → *New Project*. Select the FaSTMECH solver.
2. In the **Object Browser** left-click then right-click on the *Geographic Data* | *Elevation* and in the resulting pop-up menu select "Import > Point Cloud Data", choose ketchupisl.tpo, set the delimiter to Tab, and skip one header line in the Import Settings dialog. As always, make sure to check that the sample screen for the data looks correct before selecting "Ok" to import the data set. Right click on *Point Cloud Data1* and set the graphics display to points so you can see the actual data points, which are primarily cross-sections along with some ground surface and edge of water data near the banks of the main channel and islands.
3. From the **Object Browser** right-click *Background Images* and select "Add Image" in the resulting pop-up menu and open *ketchup2.jpg* (Figure 20).

4. To adjust the location of the image, *select Background Images | Ketchup2.jpg*. In iRIC, whenever you want to edit data whether it is Geographic Data, Grid Creating, or Images, the object that is selected in the **Object Browser** is enabled for editing.
5. When *Background Images | Ketchup2.jpg* is selected, the left mouse button will pan the image, the middle mouse button scales the image (push and slide the mouse), and the right mouse button rotates the image. If you have a two-button mouse, often pressing and holding both buttons will allow you to scale the images. Experiment with these tools to fit the image to the data (Figure 21).
6. Continue with Option 2 – step 2.

Option 2 (Importing an image already fit to the data and mapping with the template).

1. From the **Menu Bar** select *File* → *Open* and open the following project file: *iRICTutorials\FaSTMECH\Tutorial 1\Exercise 2\ Ketchup.ipro.*
2. From the **Object Browser** right-click *Grid Creating Condition* and in the resulting pop-up menu click “Select Algorithm for Creating Grid”, and then select “Create grid from polygonal line and width”. Flow direction is from bottom to top. Draw a centerline and create a grid with a width of 450 meters and discretization along the centerline of 10 meters (Figure 22).
3. Select *Point Cloud Data 1* in the Object Browser, then right click in the Preprocessor Window, choose Mapping Setting in the menu, select the “Template mapping” option and then select the “Detail…” button. Set up the following dialog similar to Figure 23 and click OK. Click OK again.
4. From the **Menu Bar** select *Grid* → *Attributes Mapping* → *Execute*. Be sure Elevation is selected and select OK. View the results and experiment with different template dimensions to see the differences in the mapped topography (Figure 24). Save your best effort to compare with TIN mapping below. Click on the camera icon in the **Main Toolbar** to save an image.
5. Compare the template mapping to the TIN mapping. Select *Point Cloud Data 1* on the Object Browser, then right click in the Preprocessor Window, select Mapping Setting and set the mapping algorithm to Mapping with TIN. Select OK. Execute the Tin Mapping by selecting from the **Menu Bar** *Grid* → *Attributes Mapping* → *Execute*. Once again be sure *Elevation* is selected and choose OK. Click on the camera icon in the **Main Toolbar** to save an image and compare with the image taken of the Template Mapping above (See Figures 24 and 25).
6. Save the file.

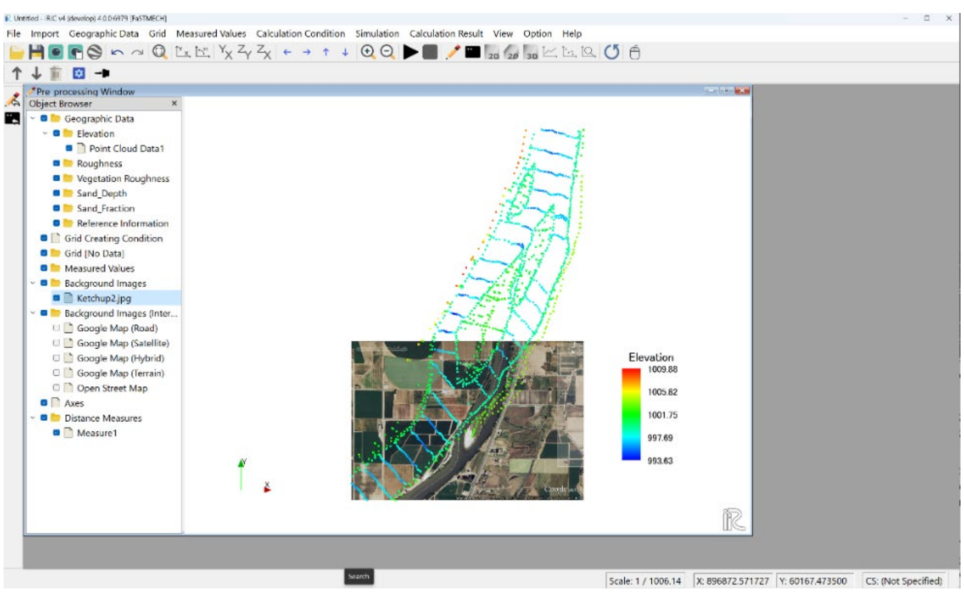


Figure 20. A non-georeferenced image imported into iRIC.

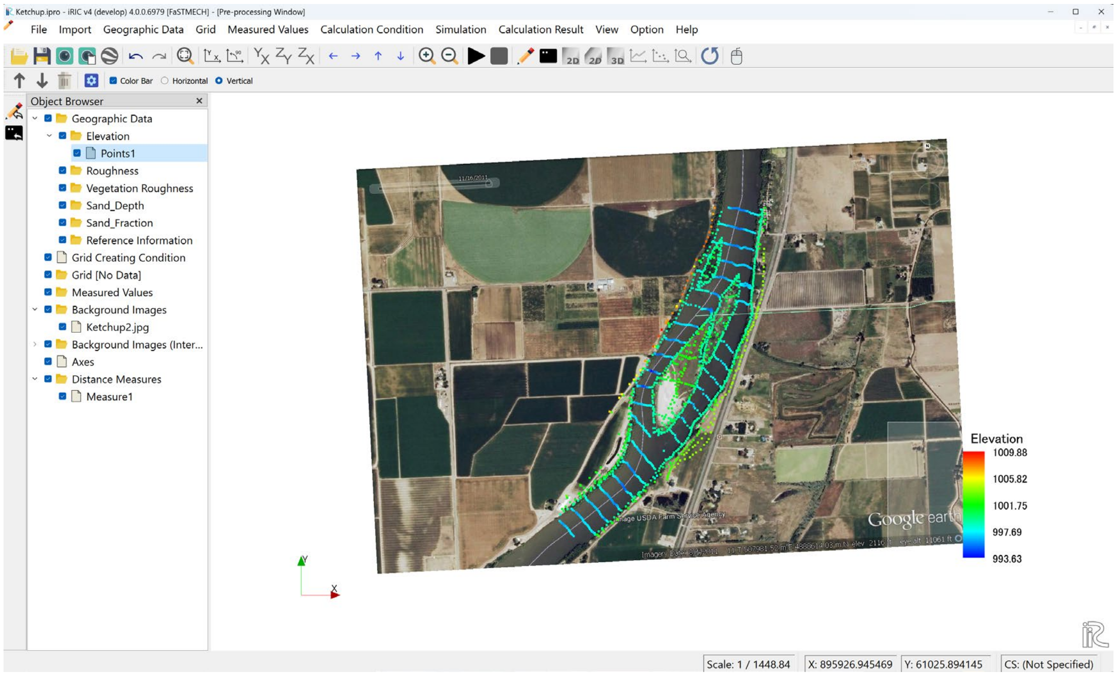


Figure 21. The background image fit to the data.

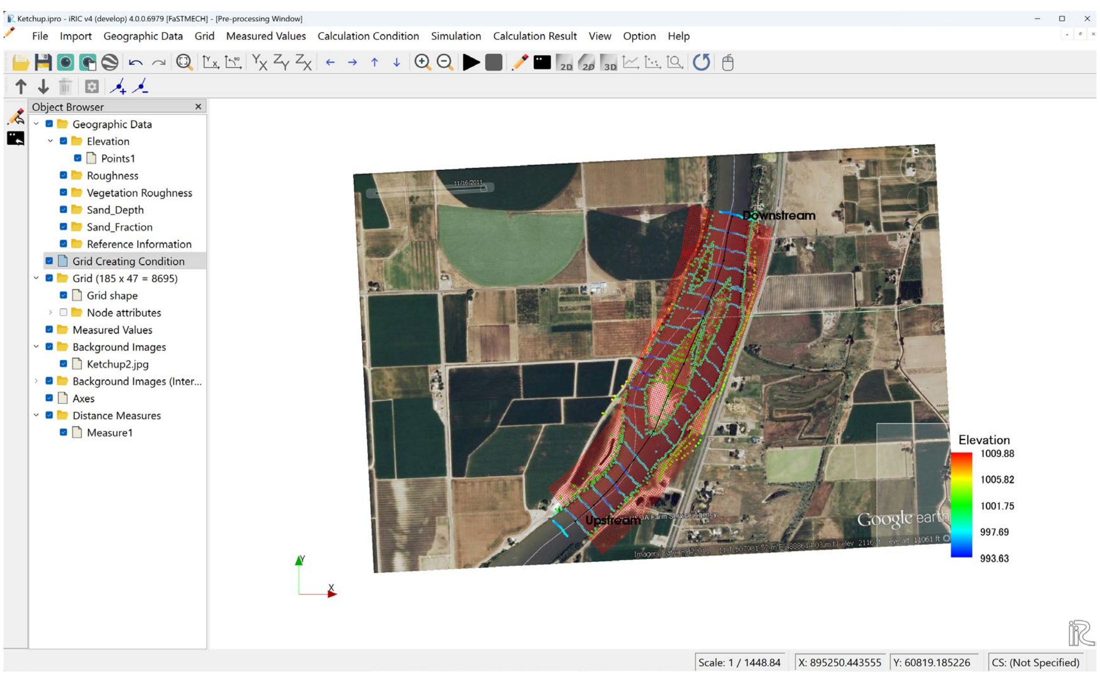


Figure 22. Location of the grid.

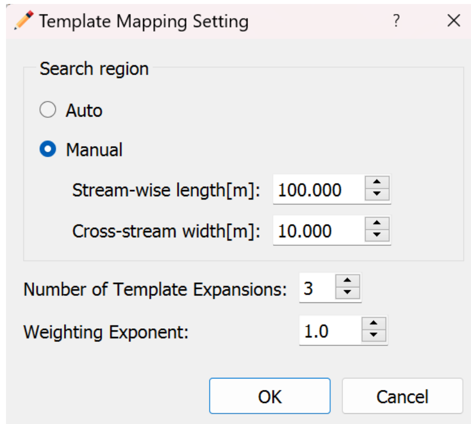


Figure 23.   Template mapping dialog

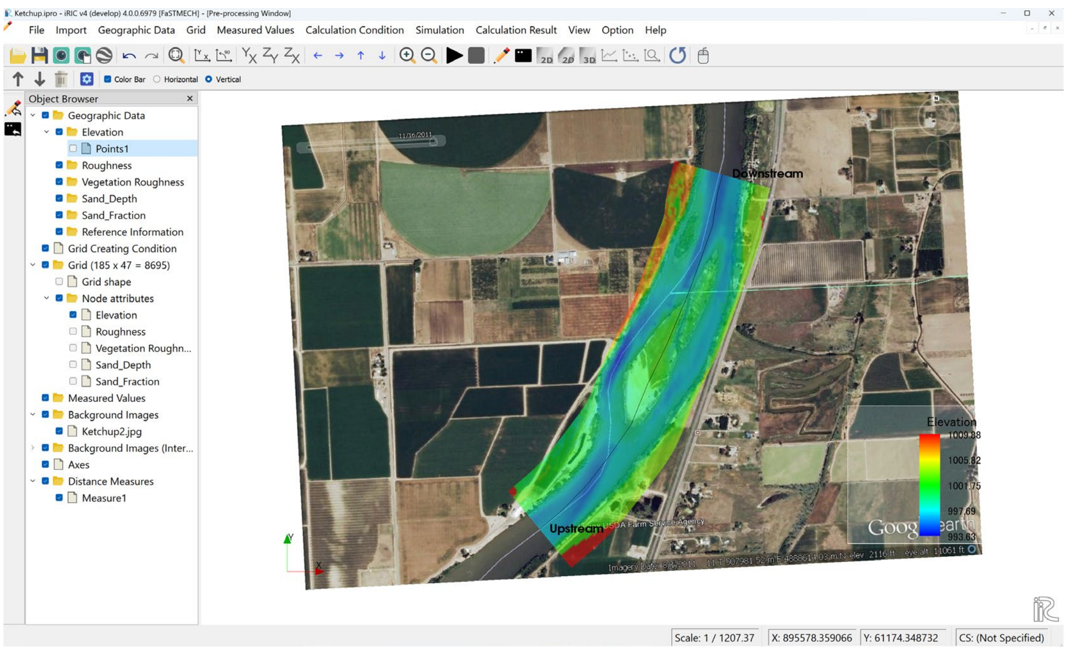


Figure 24.   Elevation mapped to the grid with the template algorithm.

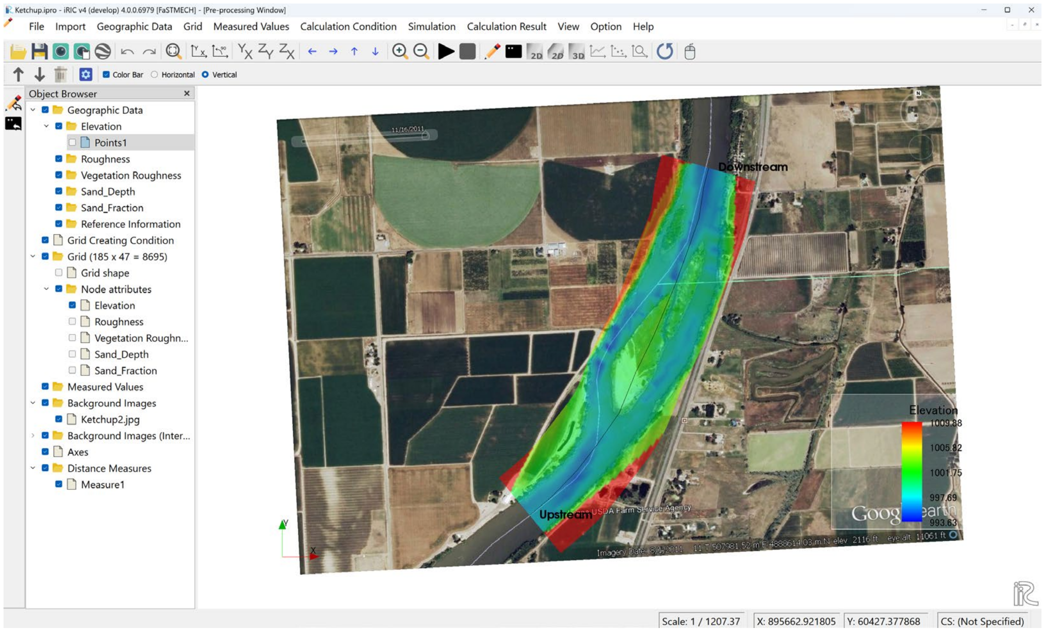


Figure 25. Elevation mapped to the grid with the TIN algorithm.

Edit Data to Improve Mapping

Topography collected in the field may not fully resolve the channel. iRIC provides a set of tools that can be used to improve the mapping. When any point data set is selected in the **Object Browser** under *Geographic Data*, right click in the Preprocessor Window and set Switch Mode to the Points Editing Mode. With this setting, right clicking in the Preprocessor Window will bring up a suite of point editing tools that can be used to add points using a simple point adding tool, various tools for deleting points, and interpolating between known points. In this part of the exercise we will briefly introduce these tools and provide an example that shows how they may be used to improve the mapping of Geographic Data to the *Grid | Node Attributes*.

- Now that you're an expert, start a new FaSTMECH project.
- Import the elevation data set *CotLower.tpo*, noting that this is space-delimited data and has one header line. Changing the graphics view to points.
- Import the georeferenced image file *output_mosaic.jpg*.
- Create a grid with a width of 200 meters, and an approximately 5 X 5 meter cell dimension along the centerline (flow is from left to right).
- Map the Grid using the TINs. Make sure that the Mapping Setting is set to TIN and that the "Execute mapping" setting in "Grid Attribute Mapping Setting" under **Menu Bar** *Grid* → *Attributes Mapping* → *Setting* is set to "Manual". Select "Execute" from the same menu list to create the TIN, checking "Elevation" in the Attribute Mapping dialog. Select OK. Select OK again.
- In the **Object Browser** uncheck *Grid () | Grid Shape* and check both *Grid () | Node attributes* and *Grid () | Node attributes | Elevation.*

- In the **Object Browser** select *Grid () | Node attributes | Elevation* and right-click *Elevation*. In the resulting pop-up menu select "Property" change the selection to "Discrete Mode". You may also adjust the transparency of the figure at the bottom of the dialog. Select OK when done.
- The result should look similar to that shown in Figure 26.
- Figure 27 shows a zoomed-in image of the downstream, river-right section of the grid in Figure 26. Because of the way the TIN algorithm creates triangles, cusps can be formed along the banks of the channel as shown in Figure 27. Zoom in to your result to find these cusps.
- To provide some insight into how these cusps are formed in the **Object Browser** turn off *Grid | Node Attributes | Elevation*. Also in the **Object Browser** right-click on *Geographic Data | Elevation | Points Cloud Data1* and in the resulting pop-up menu select "Property" and then in the Display Setting dialog select "Wireframe" as the Display Method (Figure 28). It is clear that the cusping originates from the elevation values mapped onto the grid from the TIN which has triangles with vertices located on the bank and channel essentially pulling high bank topography into the channel.
- We will use two techniques to improve the TIN and thus the mapping of elevation to the computational grid.

  1. **Breaklines**: A breakline is drawn between two or more points. When the TIN is re-meshed, edges of the triangle are enforced to conform to the breaklines. Thus, the value along the breakline is a linear interpolation of the bounding points. We will be editing *Geographic Data | Elevation | Point Cloud Data1* so make sure that is selected in the **Object Browser**. We want to separate the channel from the bank so we will create a breakline along the base of the bank following the points with lower topography or in this case cooler colors (blue as opposed to green or yellow).
     - With the Mapping Setting set to TIN and Point Cloud Data1 selected in the Object Browser, right click in the Preprocessor Window and choose Add Breakline and follow the instructions shown to draw a breakline between the four points shown in Figure 29. Note that as you move the mouse the point to be selected is highlighted. Make sure to select points at the bottom of the bank.
     - Re-mesh the TIN to reflect the breakline by right-clicking anywhere in the **Graphic View** and in the resulting pop-up menu selecting "Remesh TINs". Select OK. Note that the edges of the triangles in the TIN now conform to the breakline.
     - Remap the Geographic Data to the Grid by selecting from the **Menu Bar** *Grid* → *Attributes Mapping* → *Execute*. The result of the mapping is shown in Figure 30.
  2. **Interpolation:** The interpolation tool can be used in one of two ways. First, to interpolate linearly between known points at a specified interval and second, to interpolate between known points by a user defined path. In both cases, select known points with a left mouse click and define a path between existing points by holding the Ctrl-key while left-clicking to define the path between points.
     - In the **Object Browser** select *Geographic Data | Elevation | Point Cloud Data1*.
     - With the Graphics Setting set to points, right click in the Preprocessor Window and set the Mode to Points Edit Mode then right click in the Preprocessor Window and select the interpolation tool. An information dialog box will open, Select "Yes". Select the same points as selected above for the breakline and select "Enter" when finished. In the resulting dialog enter "3" for the Set Interpolation Increment. You should see an image similar to Figure 31A. Select OK.
     - Right-click anywhere in the **Graphics View** and in the resulting pop-up menu select "Remesh TINs". Right-click on *Geographic Data | Elevation | Points1* in the **Object Browser** and select "Property". Change the Display method to "Wireframe" and view the results (Figure 31B).
     - From the **Menu Bar** select *Grid* → *Attribute Mapping* → *Execute*. View the result in Figure 32 and compare with Figures 27 and 30.
- Save the project.

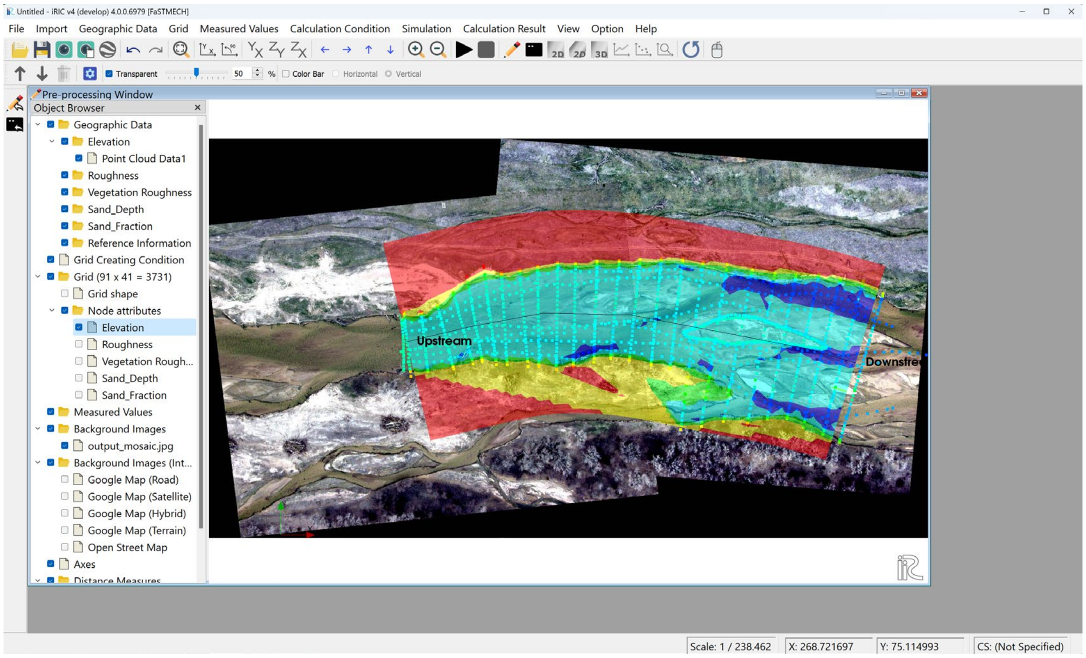


Figure 26. Mapping of reach showing cusps along lower right bank.

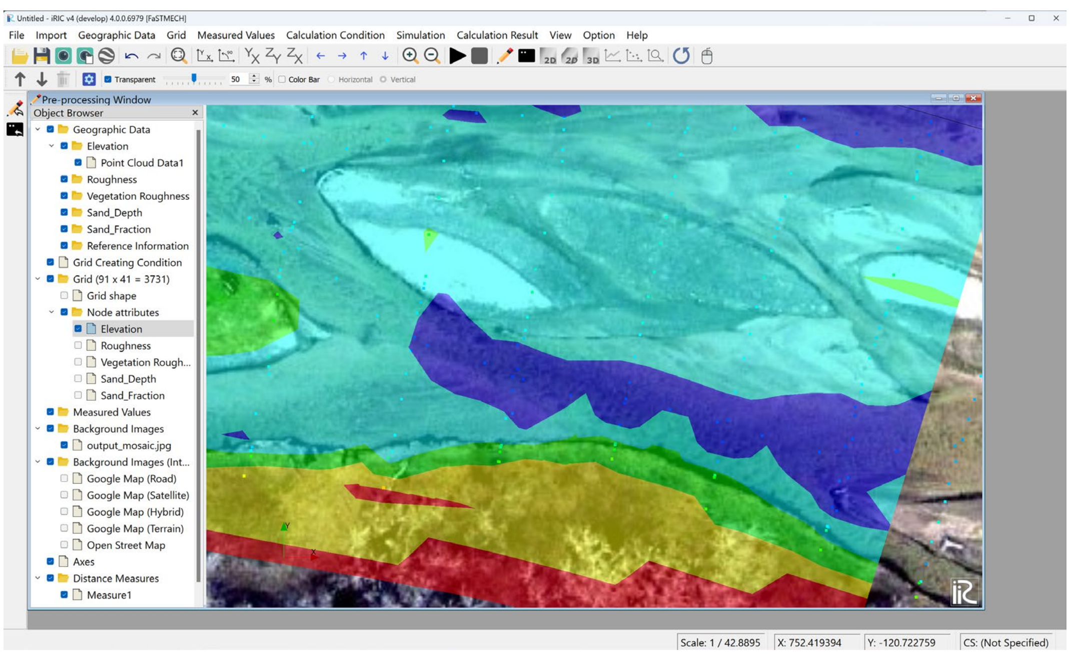


Figure 27. Cusps along the bank from the TIN algorithm.

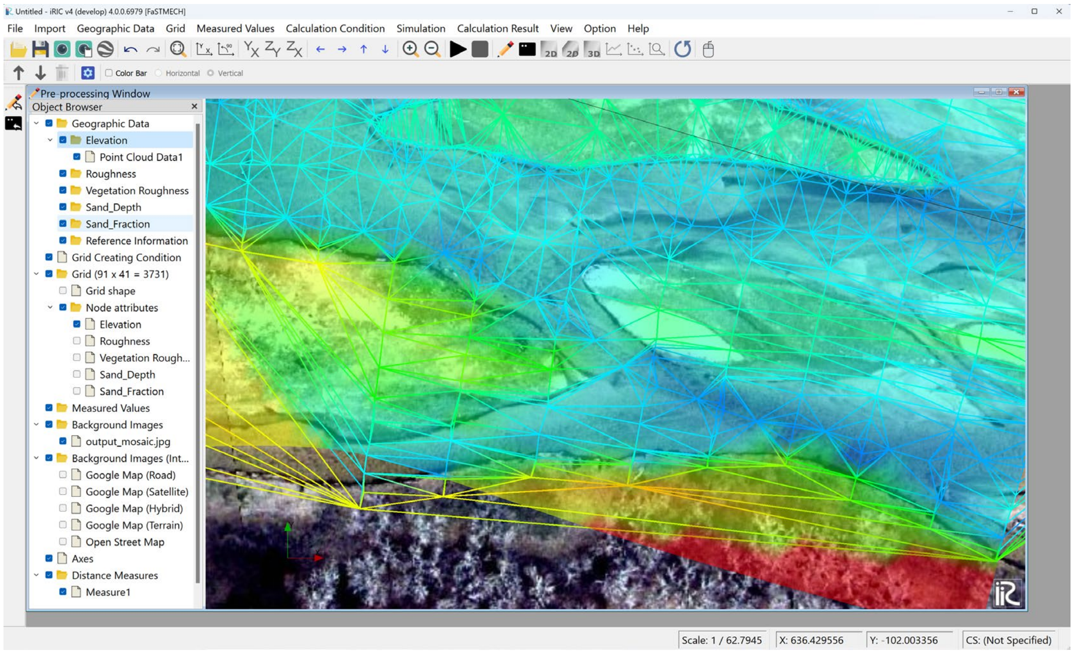


Figure 28. The TIN and location of the cusps.

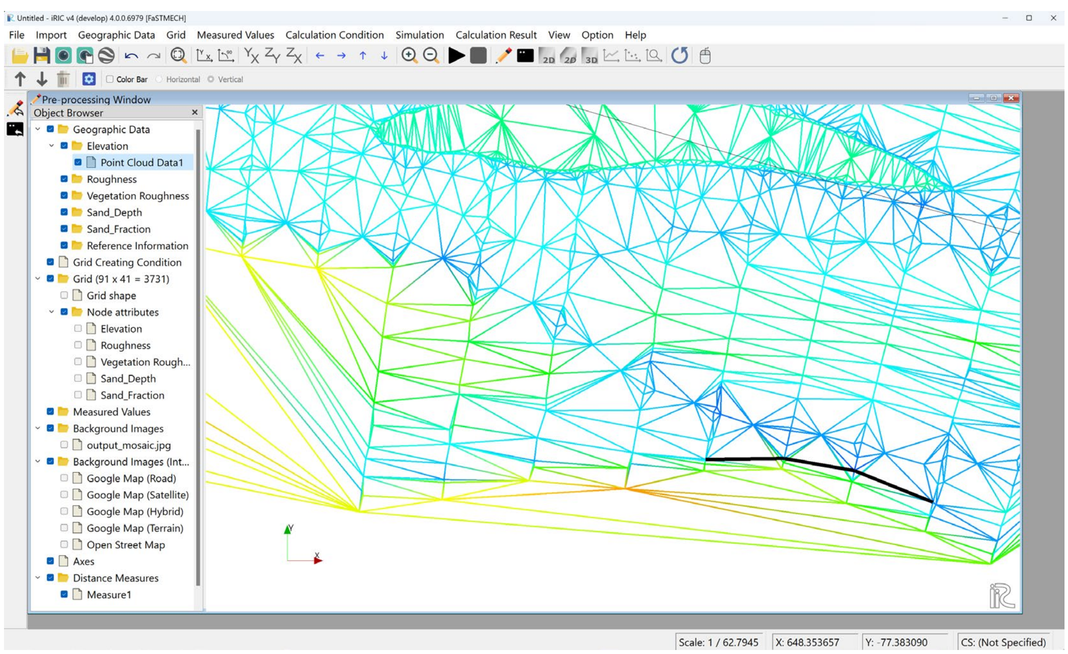


Figure 29. Location of breaklines (in black)

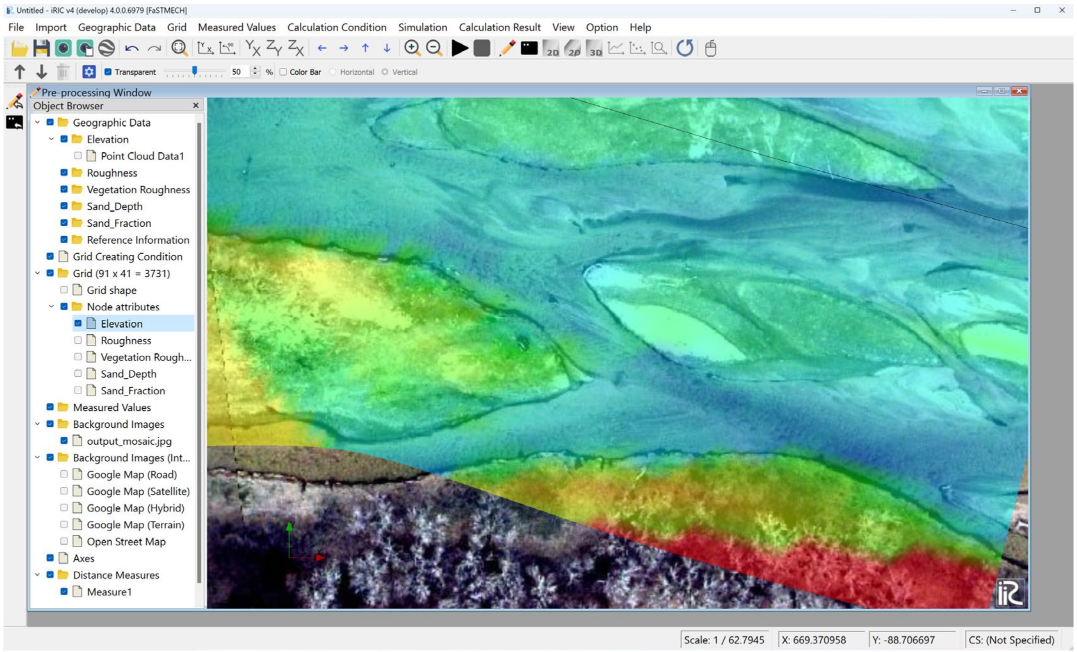
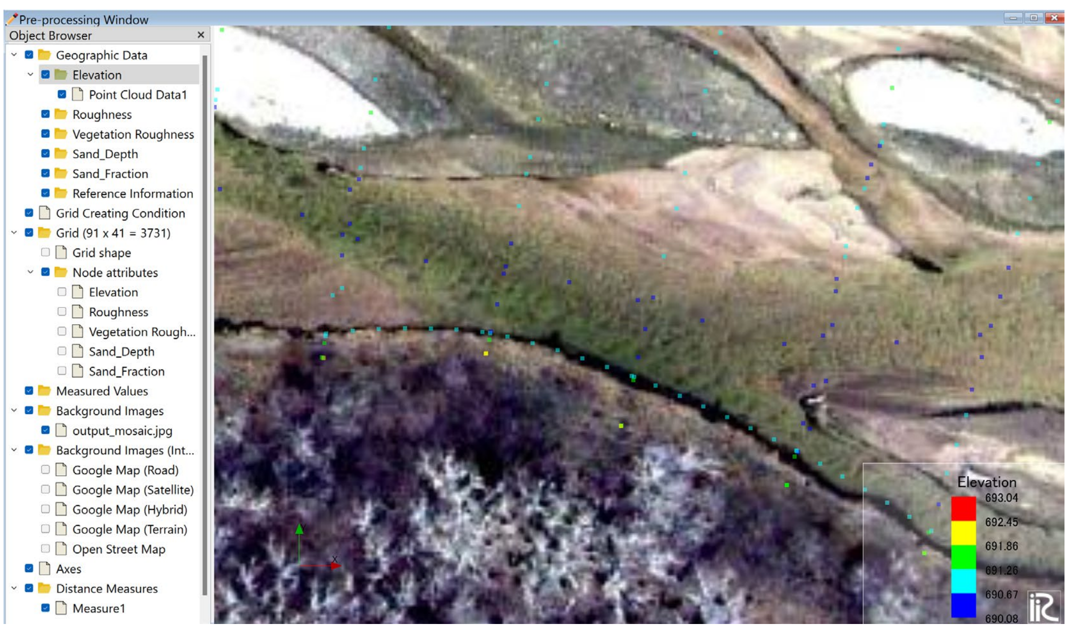
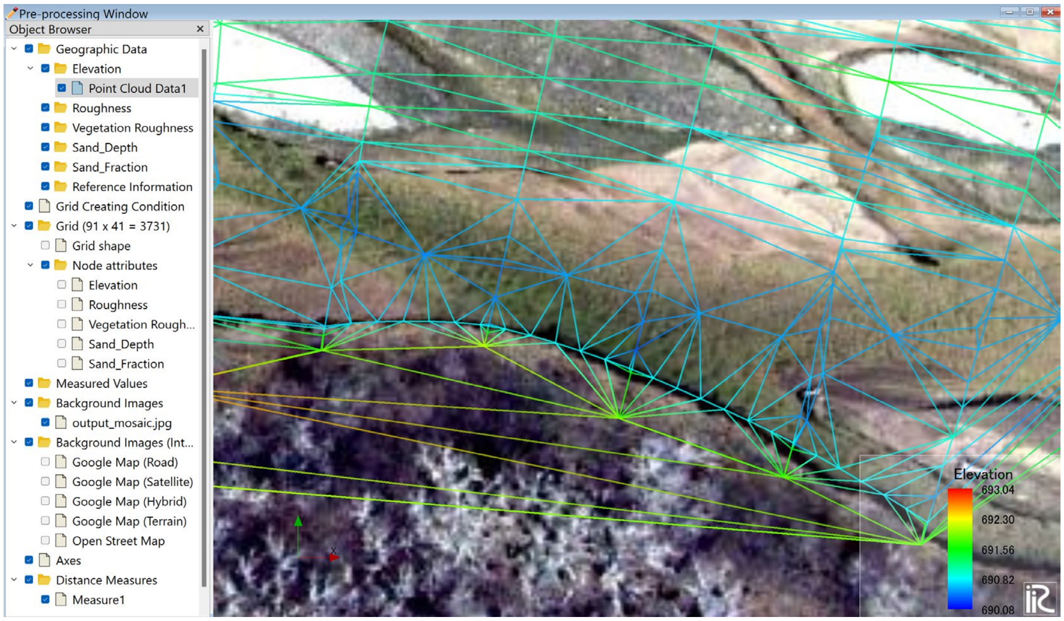


Figure 30. Result of mapping with the TIN and breaklines. Note that the cusping is gone. Compare to Figure 27.



A



B

Figure 31. Interpolation line (A) and remeshed TIN including the interpolated points (B). Compare with Figures 27 and 30.

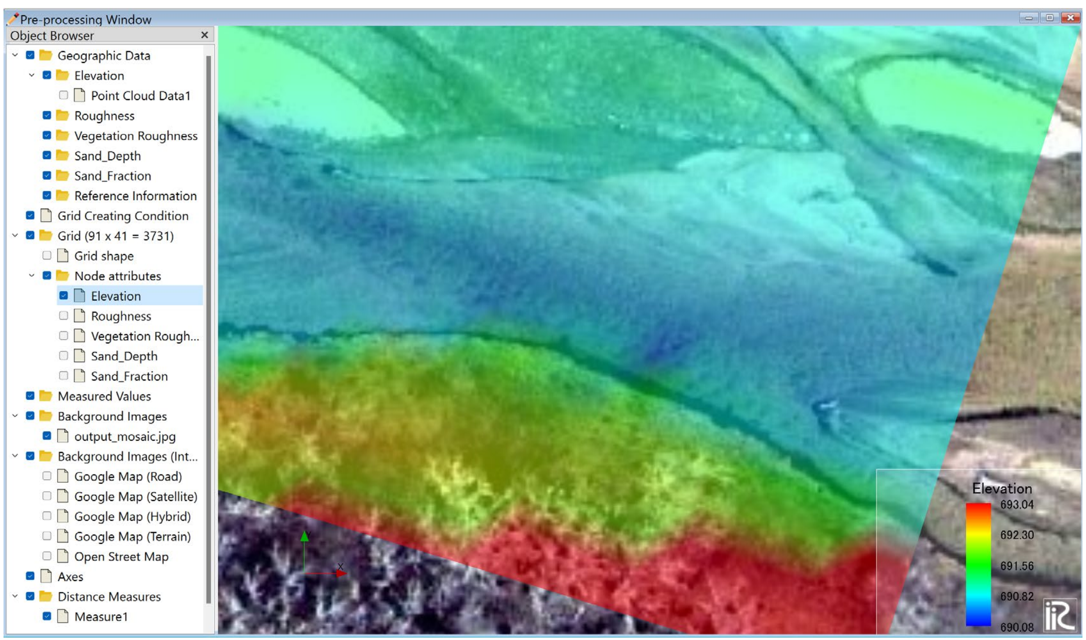


Figure 32. Remapped elevation after remeshing with interpolation line. Compare to Figures 27 and 30.

Exercise 3: Defining Calculation Conditions and Post-processing

The last exercise illustrates the process of creating a simulation and provides an introduction to post-processing the results. You will set the calculation conditions for the simulation, run the simulation, and view the results in a variety of ways. We will look at a reach of the Colorado River in Grand Canyon National Park. The simulation includes a large lateral recirculation zone on river left. Flow is from upper right to lower left. Note that there are measured water-surface elevations and velocities in the project and a grid has already been created for you.

Defining Calculation Conditions

- Open an existing project from **Menu Bar** by selecting *File* → *Open* and select the *Ex3.ipro project* file in the *Tutorial 1\Exercise 3* folder (Figure 33).
- Confirm that you are in the Pre-processing window and that your screen looks similar to Figure 33.

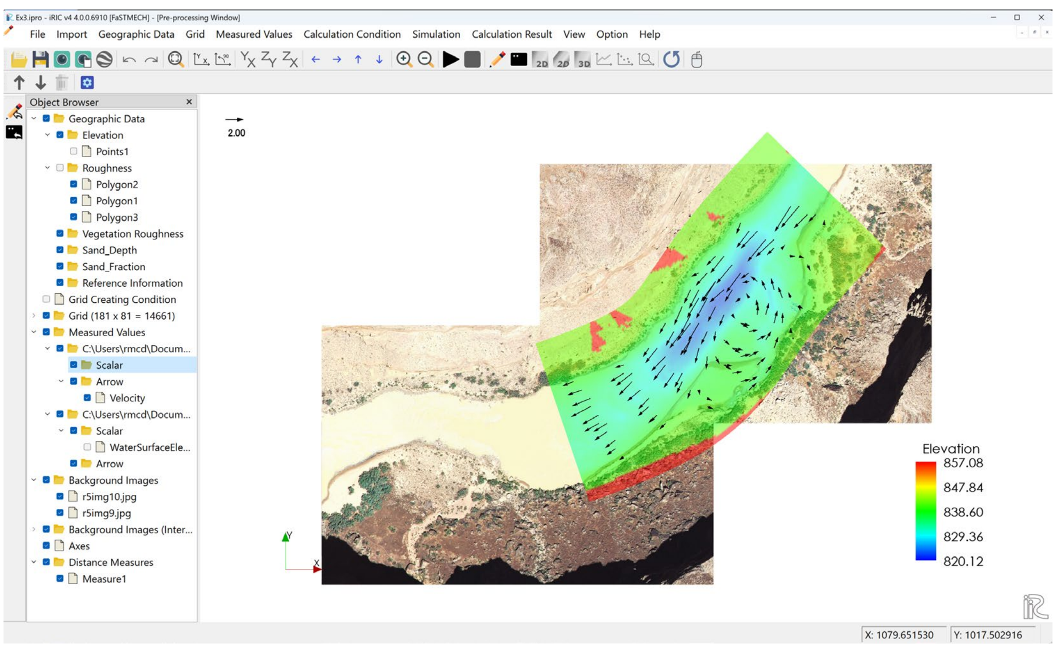
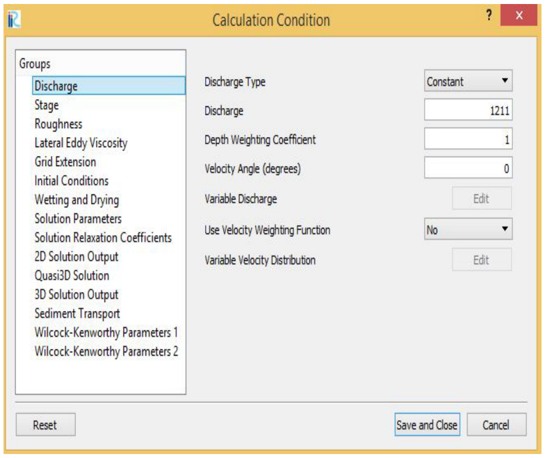
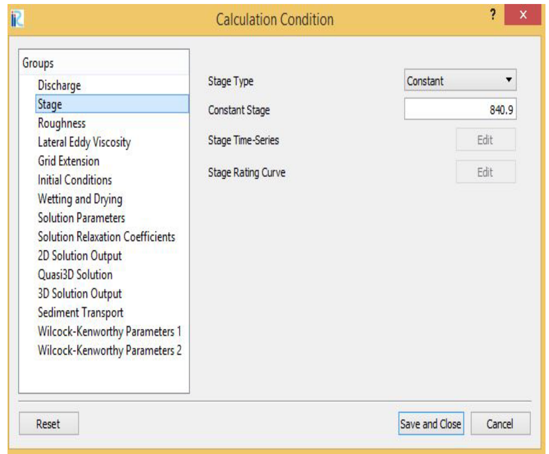


Figure 33.   Domain for Exercise 3 including the measured velocity and gridded elevation.

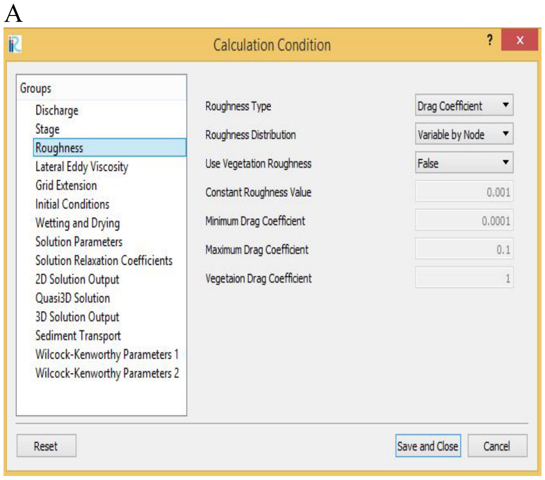
- To define the calculation conditions in the **Menu Bar** select *Calculation Condition* → *Setting.* Enter the parameters as shown in Figure 34. Select "Save and Close" when you are done.


Calculation Condition
Groups
Discharge
Stage
Roughness
Lateral Eddy Viscosity
Grid Extension
Initial Conditions
Wetting and Drying
Solution Parameters
Solution Relaxation Coefficients
2D Solution Output
Quasi3D Solution
3D Solution Output
Sediment Transport
Wilcock-Kenworthy Parameters 1
Wilcock-Kenworthy Parameters 2
Discharge Type
Constant
Discharge
1211
Depth Weighting Coefficient
1
Velocity Angle (degrees)
0
Variable Discharge
Edit
Use Velocity Weighting Function
No
Variable Velocity Distribution
Edit
Reset
Save and Close
Cancel

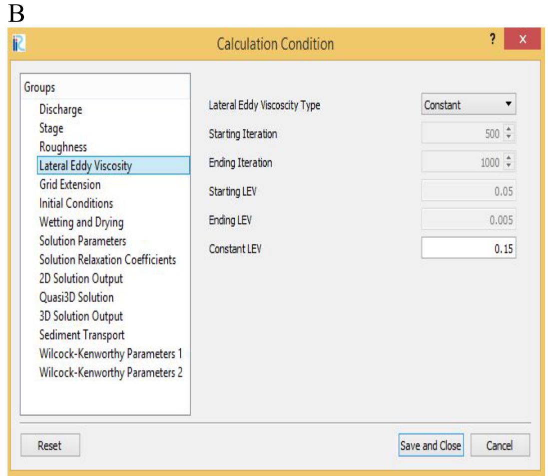
A


Calculation Condition
Groups
Discharge
Stage
Roughness
Lateral Eddy Viscosity
Grid Extension
Initial Conditions
Wetting and Drying
Solution Parameters
Solution Relaxation Coefficients
2D Solution Output
Quasi3D Solution
3D Solution Output
Sediment Transport
Wilcock-Kenworthy Parameters 1
Wilcock-Kenworthy Parameters 2
Stage Type
Constant
Constant Stage
840.9
Stage Time-Series
Edit
Stage Rating Curve
Edit
Reset
Save and Close
Cancel

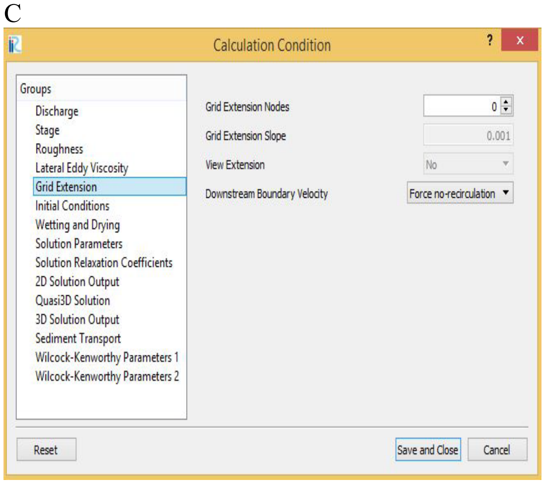
B


Calculation Condition
Groups
Discharge
Stage
Roughness
Lateral Eddy Viscosity
Grid Extension
Initial Conditions
Wetting and Drying
Solution Parameters
Solution Relaxation Coefficients
2D Solution Output
Quasi3D Solution
3D Solution Output
Sediment Transport
Wilcock-Kenworthy Parameters 1
Wilcock-Kenworthy Parameters 2
Roughness Type
Drag Coefficient
Roughness Distribution
Variable by Node
Use Vegetation Roughness
False
Constant Roughness Value
0.001
Minimum Drag Coefficient
0.0001
Maximum Drag Coefficient
0.1
Vegetaion Drag Coefficient
1
Reset
Save and Close
Cancel

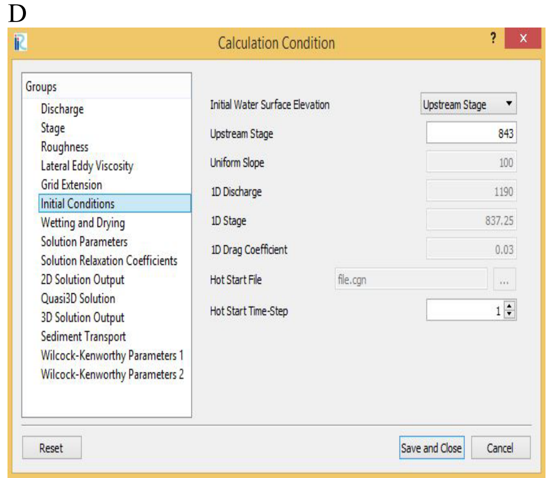
C


Calculation Condition
Groups
Discharge
Stage
Roughness
Lateral Eddy Viscosity
Grid Extension
Initial Conditions
Wetting and Drying
Solution Parameters
Solution Relaxation Coefficients
2D Solution Output
Quasi3D Solution
3D Solution Output
Sediment Transport
Wilcock-Kenworthy Parameters 1
Wilcock-Kenworthy Parameters 2
Lateral Eddy Viscoscity Type
Constant
Starting Iteration
500
Ending Iteration
1000
Starting LEV
0.05
Ending LEV
0.005
Constant LEV
0.15
Reset
Save and Close
Cancel

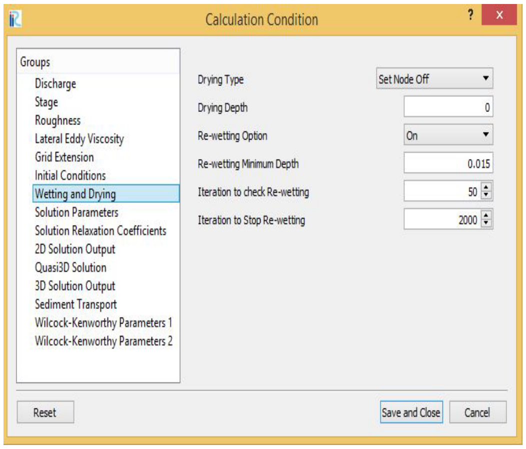
D


Calculation Condition
Groups
Discharge
Stage
Roughness
Lateral Eddy Viscosity
Grid Extension
Initial Conditions
Wetting and Drying
Solution Parameters
Solution Relaxation Coefficients
2D Solution Output
Quasi3D Solution
3D Solution Output
Sediment Transport
Wilcock-Kenworthy Parameters 1
Wilcock-Kenworthy Parameters 2
Grid Extension Nodes
0
Grid Extension Slope
0.001
View Extension
No
Downstream Boundary Velocity
Force no-recirculation
Reset
Save and Close
Cancel

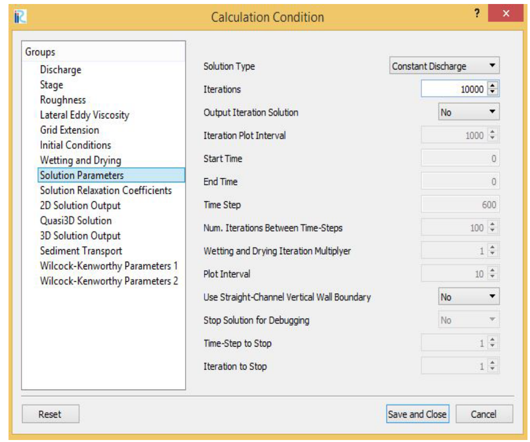
E


Calculation Condition
Groups
Discharge
Stage
Roughness
Lateral Eddy Viscosity
Grid Extension
Initial Conditions
Wetting and Drying
Solution Parameters
Solution Relaxation Coefficients
2D Solution Output
Quasi3D Solution
3D Solution Output
Sediment Transport
Wilcock-Kenworthy Parameters 1
Wilcock-Kenworthy Parameters 2
Initial Water Surface Elevation
Upstream Stage
Upstream Stage
843
Uniform Slope
100
1D Discharge
1190
1D Stage
837.25
1D Drag Coefficient
0.03
Hot Start File
file.cgn
Hot Start Time-Step
1
Reset
Save and Close
Cancel

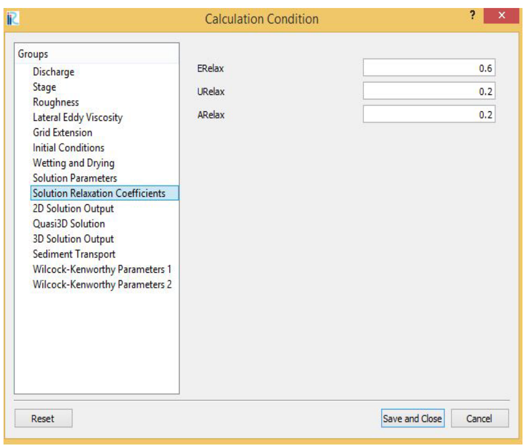
F



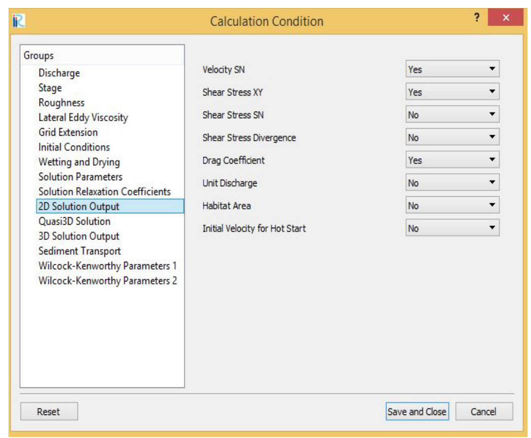
G



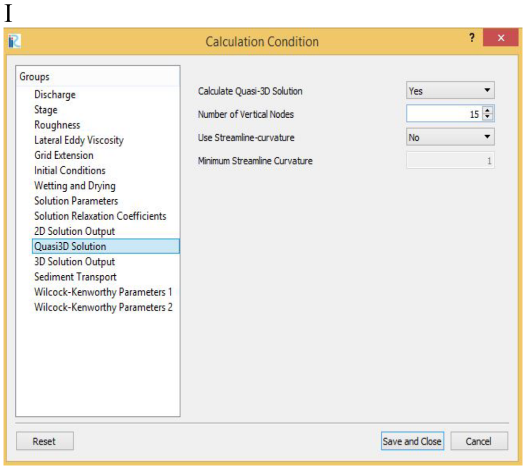
H



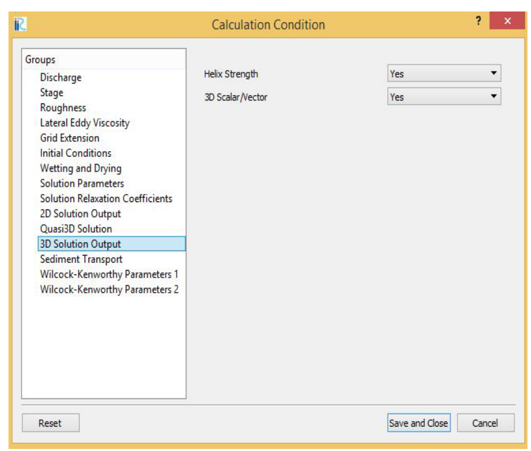
I



J



K



L

Figure 34. A. Discharge, B) Stage, C) Roughness, D) Lateral Eddy Viscosity, E) Grid Extension, F) Initial Condition, G) Wetting and Drying, H) Solution Parameters, I) Solution Relaxation Coefficients, J) 2D Solution Output, K) Quasi 3D Solution, L) 3D Solution Output.

- To run the simulation from the **Menu Bar** select *Simulation* → *Run* or click on the button. The warning dialog will open to ask whether you wish to save the current project. Select "Yes". After the project saves a Solver Console will open. This Console shows information about the simulation as the calculation is running. A dialog opens to notify you when the calculation is complete. Click OK to close the dialog.

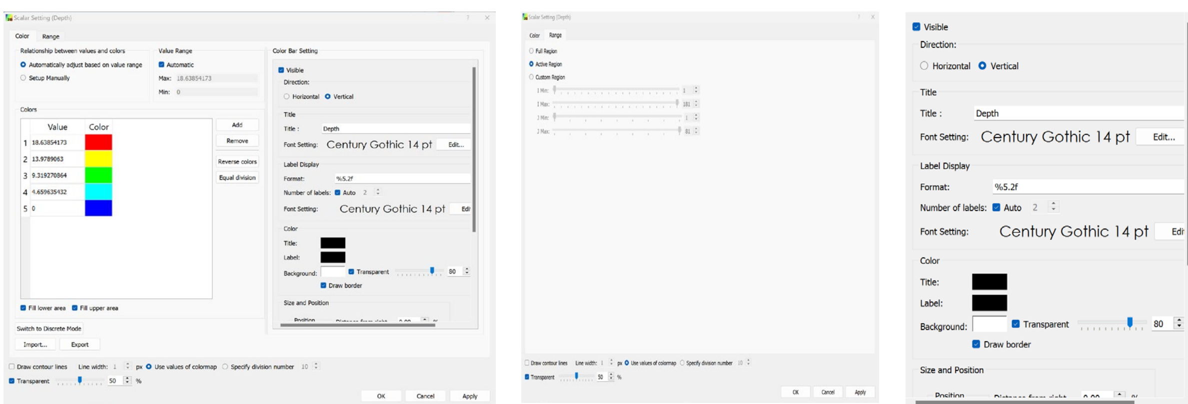
2D Post-processing

Calculation results can be viewed by opening a new 2D Post-processing Window by selecting **Menu Bar** *Calculation Results* → *Open new 2D Post-processing Window* from the **Menu Bar** or by selecting the button on the **Main Toolbar**. The Post-processing Window is organized in a similar way as the Pre-processing Window with an Object Browser and Canvas. The Object Browser in a Post-processing Window allows you to control the display of calculated flow characteristics such as depth, water-surface elevation, and velocity as well as to display arrows (vectors).

Displaying Scalar Results

Scalar results show the magnitude of various flow characteristic through contour plots.

- In the **Object Browser** turn off *Measured Values*.
- In the **Object Browser** select the check box next to *FaSTMECH Grids | iRICZone | Scalar | Depth*.
- Notice that the entire grid is contoured. To mask the contour to the nodes of the grid that are wet, from the **Object Browser** right-click on *FaSTMECH Grids | iRICZone | Scalar | Depth* and in the pop-up menu select "Property", and in the resulting Scalar Setting dialog (Figure 35A) select the Range tab (Figure 35B), then select the "Active Region" radio button and then select OK.
- In the same Scalar Setting dialog click the "Switch to Discrete Mode" button and take note of other attributes that can be set from this dialog including the min, max, and number of intervals for the contour. Change the display of the legend or scalebar using the "Color Bar Setting" subdialog in the Scalar setting dialog (Figure 35C). Select OK when finished.
- In the **Object Browser** turn the *Background Images* on. The result should look like Figure 36.



A B C

Figure 35. A) Scalar Setting dialog and associated B) Region Setting, and C) Color Legend Setting dialogs.

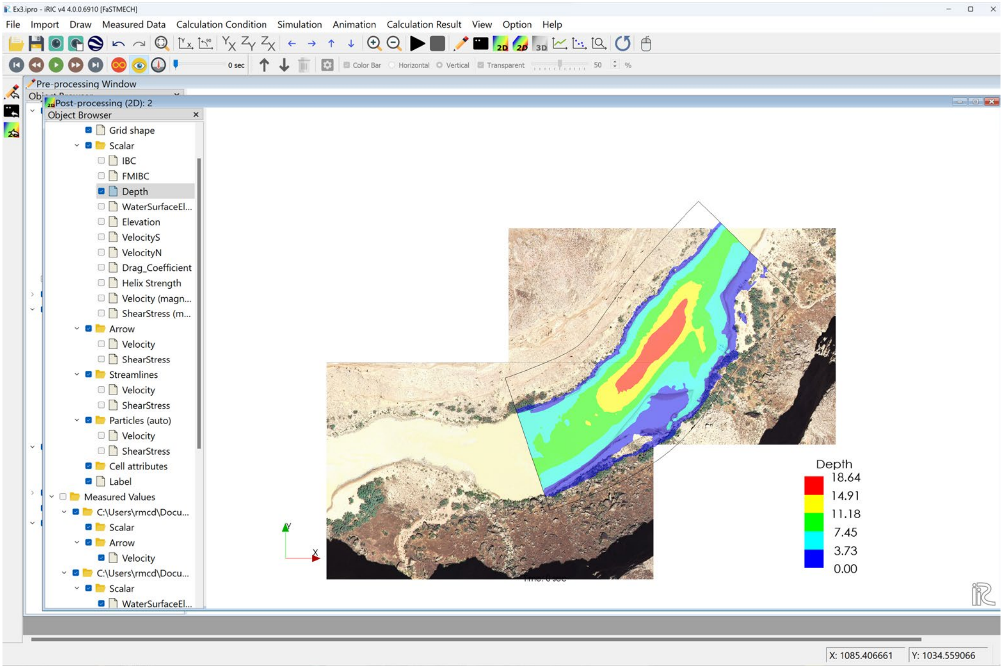


Figure 36. Scalar solution of Depth.

Displaying 2D Vector Results

- In the **Object Browser** turn on *FaSTMECH Grids | iRICZone | Arrow | Velocity*. At first the vectors will be hard to discern so we'll adjust their size, then adjust their plotting increment to make them easier to see.
- Access the Arrow properties by right-clicking on *Arrow* in the **Object Browser** and selecting "Property" in the resulting pop-up menu.
- In the resulting Arrow Setting dialog (Figure 37A) in the "Length" attribute deselect the "Auto" property and set the "Length" on screen property to 25 pixels. In the "Sampling" attribute select the "Sampling rate" radio button and set the "Sampling rate" property to 2 in both the I and J directions. Select OK. The result is shown in Figure 37B.
- Alternatively, turn off the Depth, set the "Arrow Setting" attributes as in Figure 37C, and use the Color tab to map the vector colors using the velocity magnitude with a result as shown in Figure 37D.

Compare Measured Velocity Vectors to Solution Vectors

- In the previous step you set the Arrow Setting attributes as in Figure 37D. Using the Color tab on the Arrow Setting attributes, change the vector color attribute back to black. To compare the measured velocity vectors with the solution vectors we want to scale the measure vectors the same as the solution vectors. In the **Object Browser** select *Measured Values | C\:(path) | Arrow* and then right-click and in the resulting pop-up menu select "Property". Set the "length" property attributes to the same ones in Figure 37C by unchecking "Auto" and changing the values, such that both the measured and solution vectors have the same scale. Using the Color tab, change the vector color attribute to red. Select OK when done.
- Confirm that *Measured Values* is now turned on in the **Object Browser**.
- The result is shown in Figure 38.

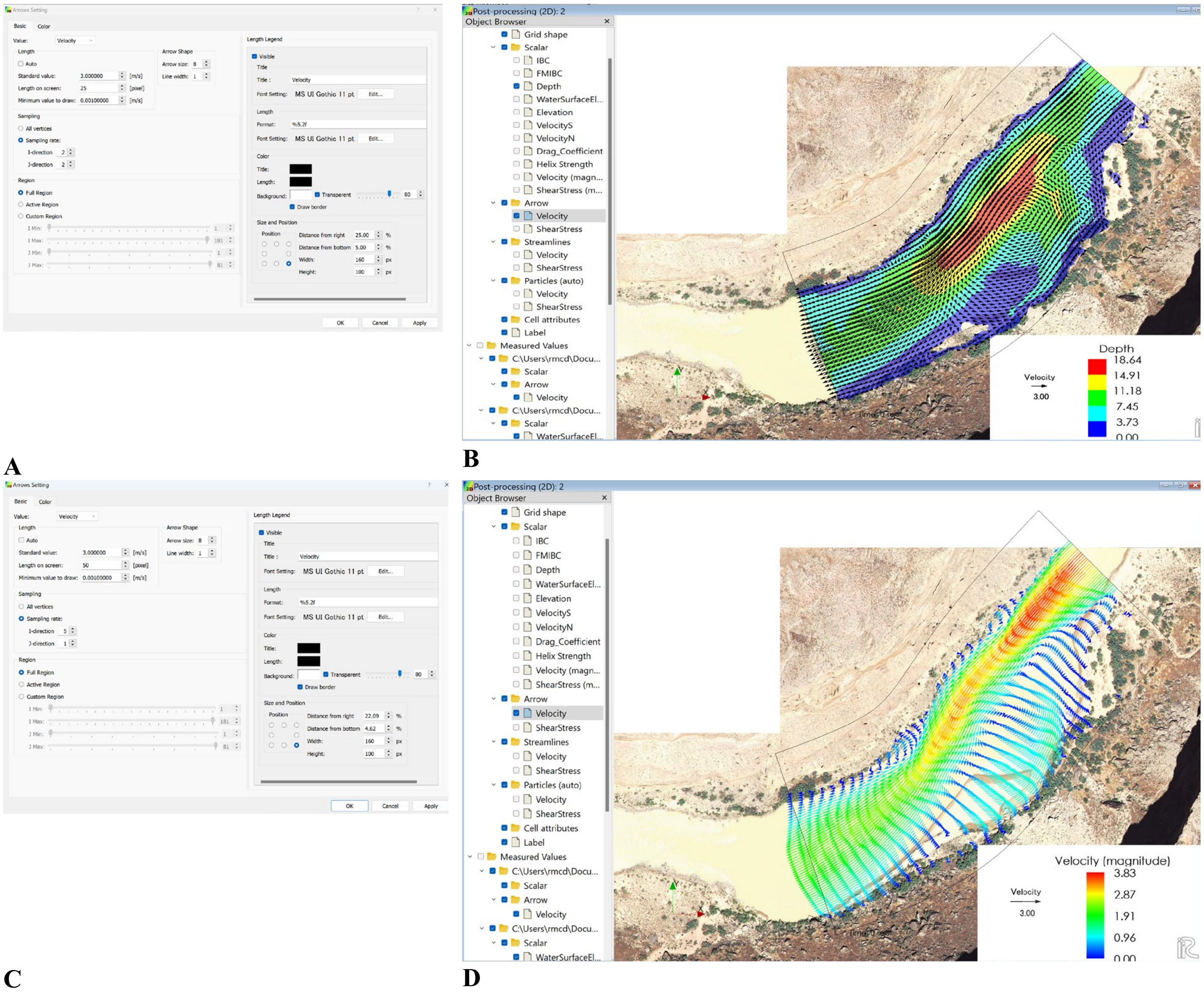


Figure 37. Arrow setting attributes and resulting plot of vectors.

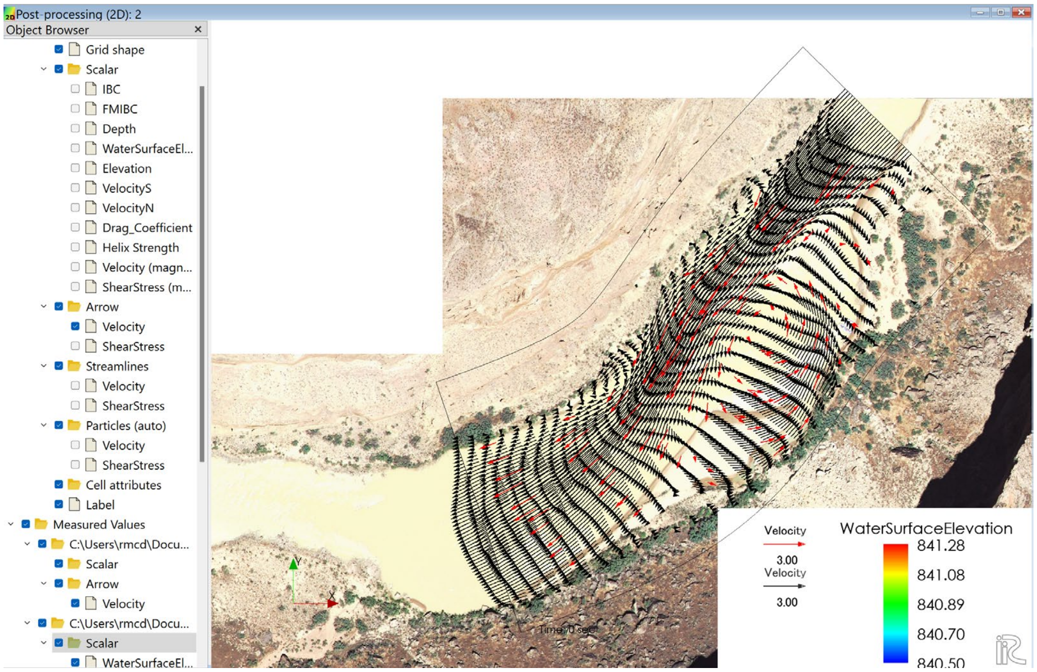


Figure 38. Solution vectors (Black) and measured vectors (Red).