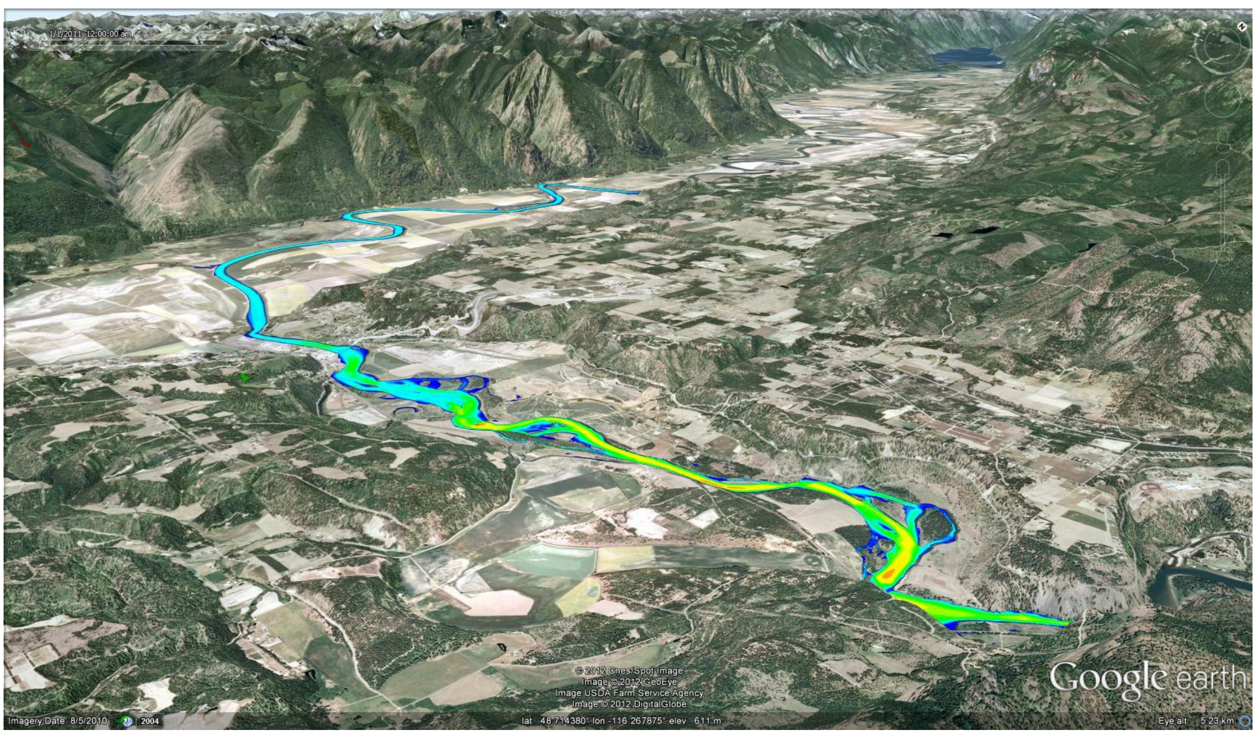
0

•

iRIC Software

Changing River Science

FaSTMECH Tutorial 1



Kootenai River Valley, Idaho, USA

Contents

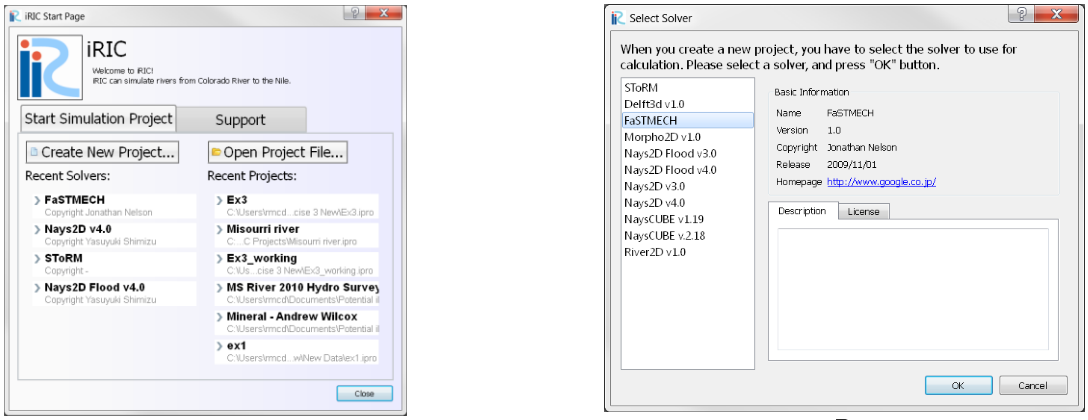
Introduction to the iRIC Graphical User Interface

Launch iRIC 2.0 by selecting iRIC 2.0 from the Program Menu list or click 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.



A B

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

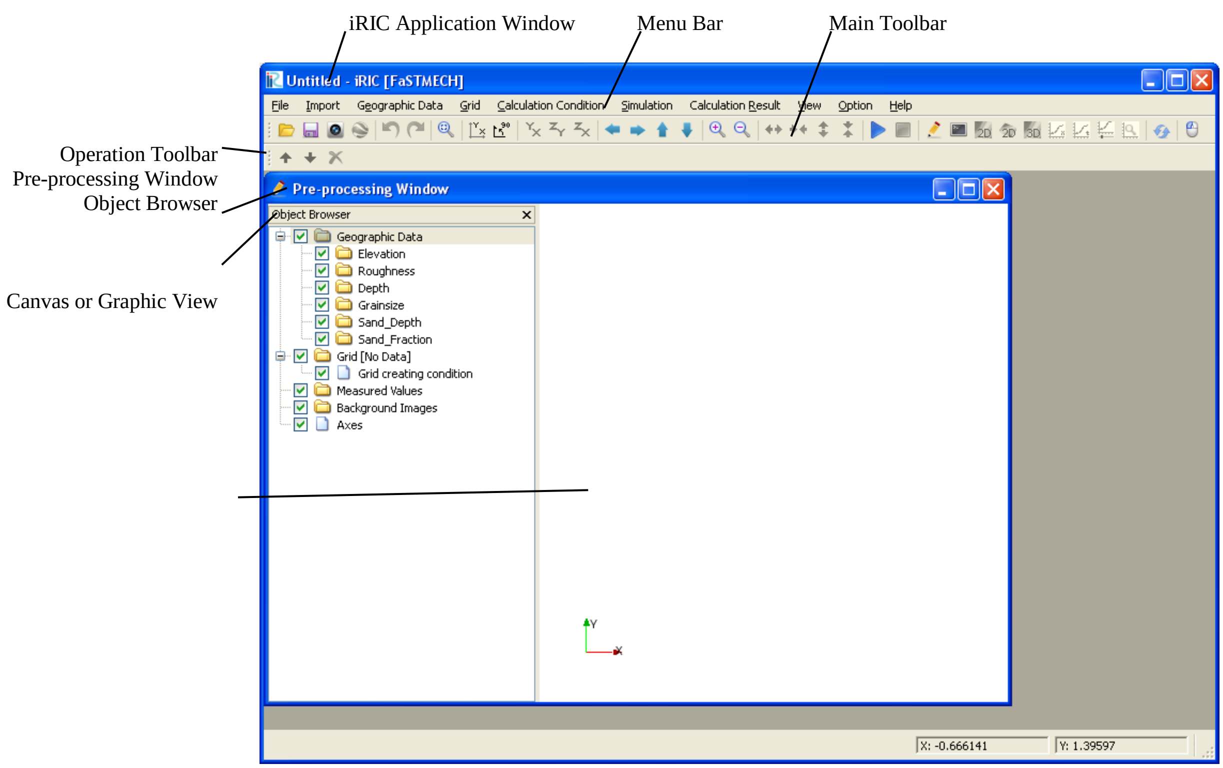


Figure 2. Example of the main iRIC interface

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, Calculation Conditions, Solutions, Calculation Results, View, Options, and Help. Addition 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
- Zooms to the full extent of the data
- 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 display screen 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 Tree. 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.

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 tools (, ,). 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 select ".tpo" in the Files of Type drop down menu and select the following file: r5finpt2m114_shifted.tpo. This will open a dialog that allows you to filter or reduce the number of points imported into iRIC. This can be useful if your data set is extremely large, but for our purposes leave the default setting at 1 and select enter 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* **|** *Points1* (Figure 1). In the Object Browser the topography can be made visible or not visible 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* **|** *Points1* 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 and change the point size to 1 (Figure 2).

- Add a data legend for the elevation data by selecting in the **Object Browser** *Geographic Data* **|** *Elevation* and then right-clicking and in the resulting pop-up menu selecting Set up Scalarbar. Make sure Elevation is displayed in the drop down menu and check the "Visible" box. Select the "Edit…" button for additional features that control the legend display. Or simply left click on the legend and drag to a new location.

  To change the range and intervals displayed in the data legend select **Menu Bar ->**Geographic Data **->**Color setting **->**Elevation. This opens a dialog that allows you to change the minimum and maximum elevation colors. (Figure 3). Uncheck "Automatic" and change the maximum to 840 to see more detail in the channel.
- Explore the Pre-Processing window controls using the buttons on the **Main Toolbar** to zoom in and out and pan (See the Introduction for an overview). Also try the mouse options that allow you to pan (ctrl + left mouse button), zoom in and 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 or TIN of the elevation data set. In the **Object Browser** right click on *Elevation* | *Points1* 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 2). Experiment with viewing the data as wireframe and as a surface.

- Save by selecting File **->**Save as File (*.ipro) from the Menu Bar.

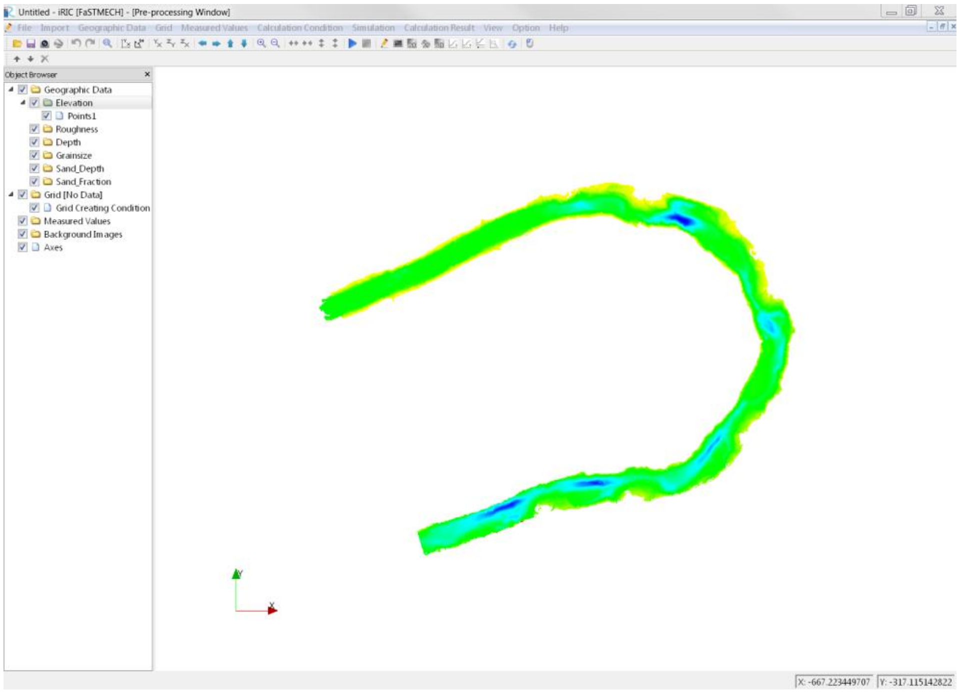


Figure 1. Pre-processing Window display of the elevation data.

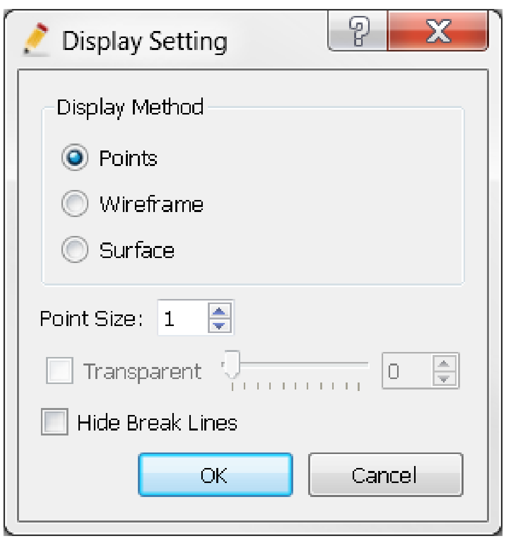


Figure 2. Display properties options for Elevation Data

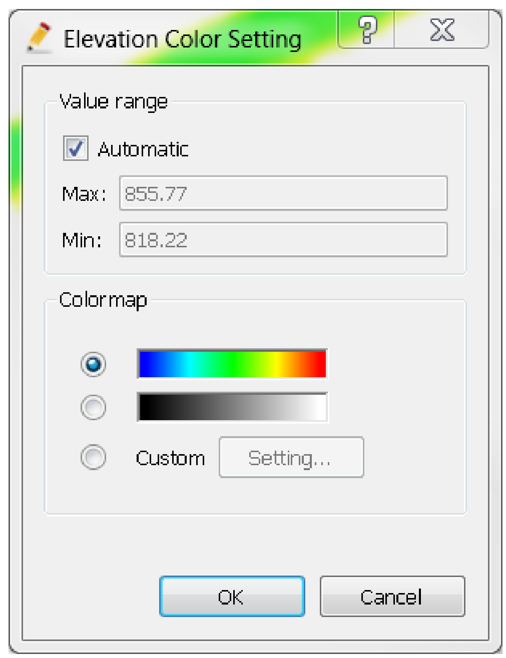


Figure 3. Display properties options for Elevation Color Setting

Import Images

- Import images to placed 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. Ignore this and click ok to show the image in the Pre-Processing window, your screen should look like Figure 4.
- Open a second image file through the **Object Browser** by right-clicking on *Background Image* and selecting Import Images in the resulting pop-up menu. Select the file r5img10.jpg. As with importing Geographic Data there are multiple ways to access the import functions by using the **Menu Bar** or by using the **Object Browser**.
- 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.
- Save the file

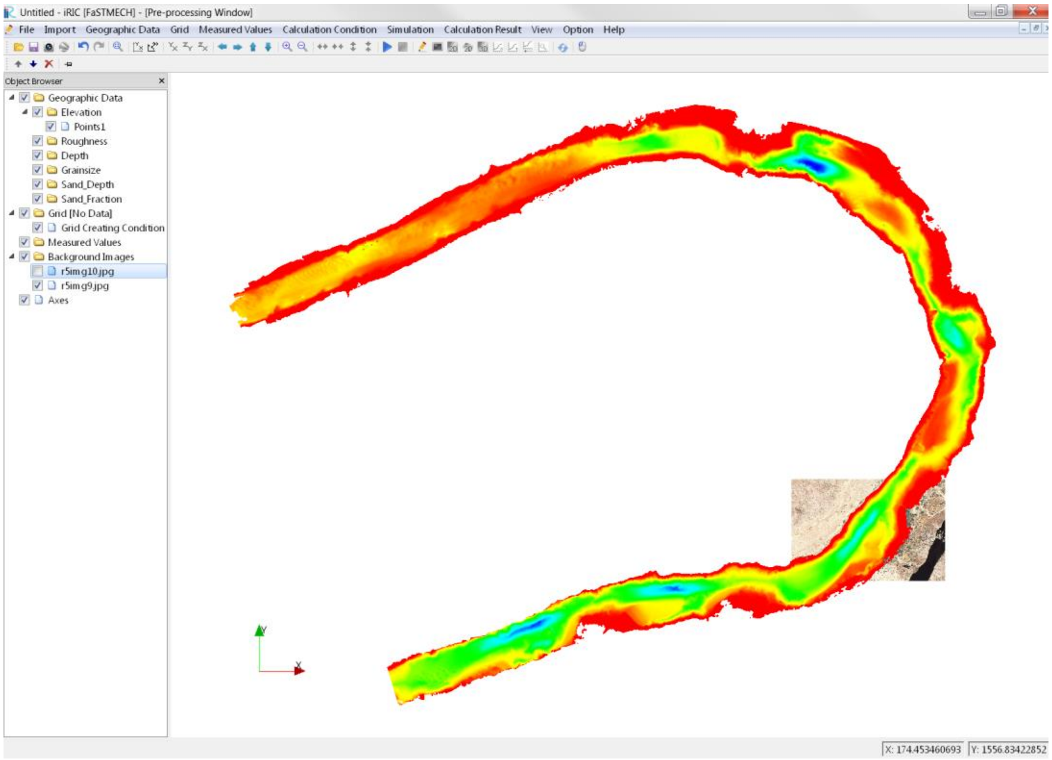


Figure 4. Surveyed cross-sections and background image.

Simple Data Editing

- We would like to focus on the region of the data set encompassed by the two imported images. iRIC 2.0 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 | Points1*. You will see a number of tool icons loaded into the Operation Toolbar as in Figure 5 below.
- To select a rectangle of points, make sure the icon is selected (selected by default) and 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 6). 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 7. Thus, you'll need to draw a rectangle with your mouse to encompass the points outside the images. This will take several selections. Remember to use undo if you make a mistake while editing your data points.

- More information on the data editing tools can be found in the User's Guide.

Figure 5. Data editing tools available for point data sets.

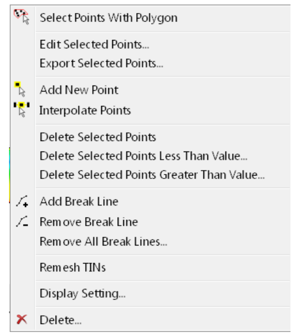


Figure 6. Pop-up menu for actions applied to selected points

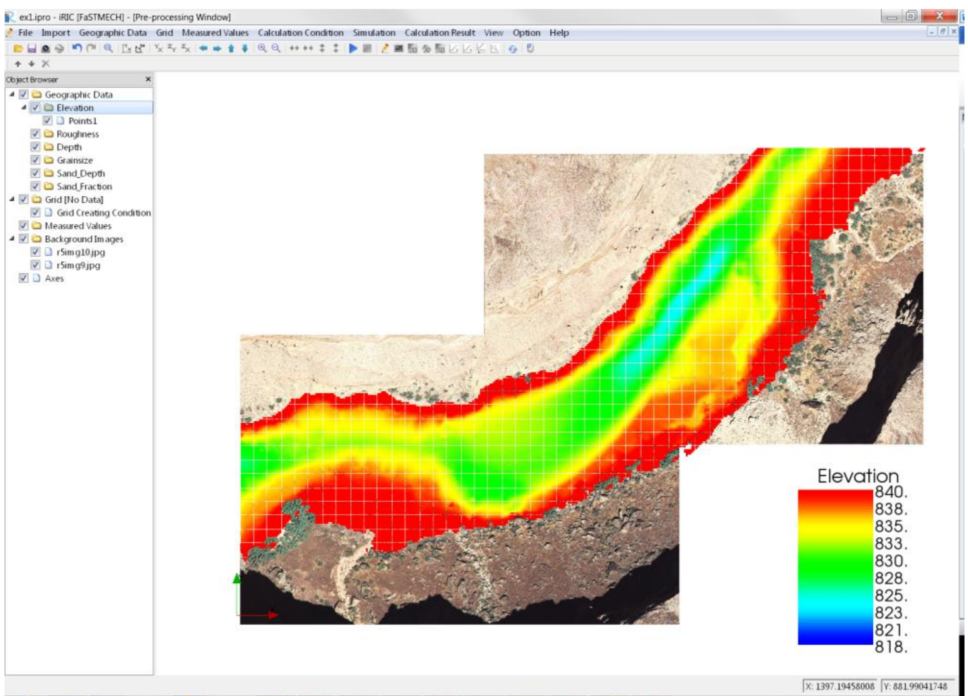


Figure 7. Image and elevation points following data point editing.

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** select Import **->**measured values. Select EM_mar7a_adcp shift.csv.
- 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 8 below.

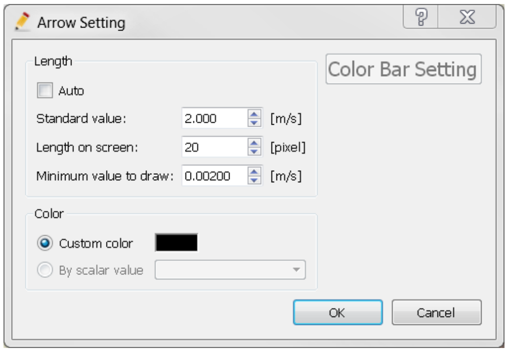


Figure 8. Arrow setting to set measure vector properties

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.
- 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.
- The resulting view with images in the background and measured water-surface elevations and flow velocities are shown in Figure 9.
- Save the project

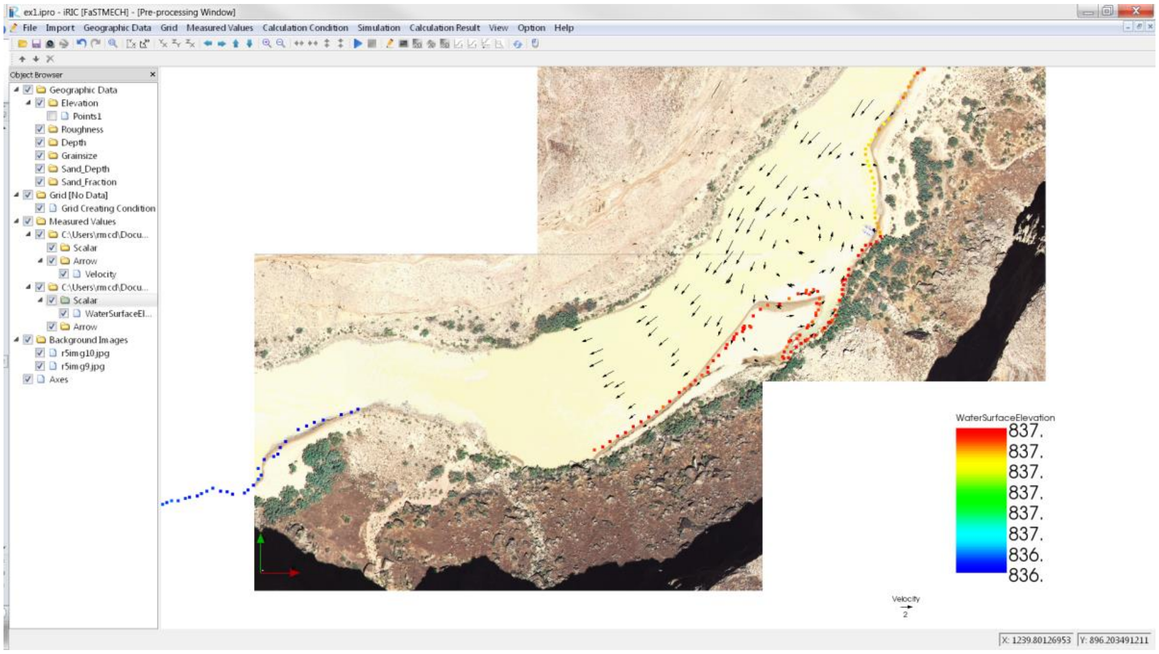


Figure 9. Measured velocity vectors and water-surface elevations scalar values shown on top of geo-referenced images.

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, or importing every 10th point in the data set.
- Import the Meander2.jpg file (Figure 1).
- 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.

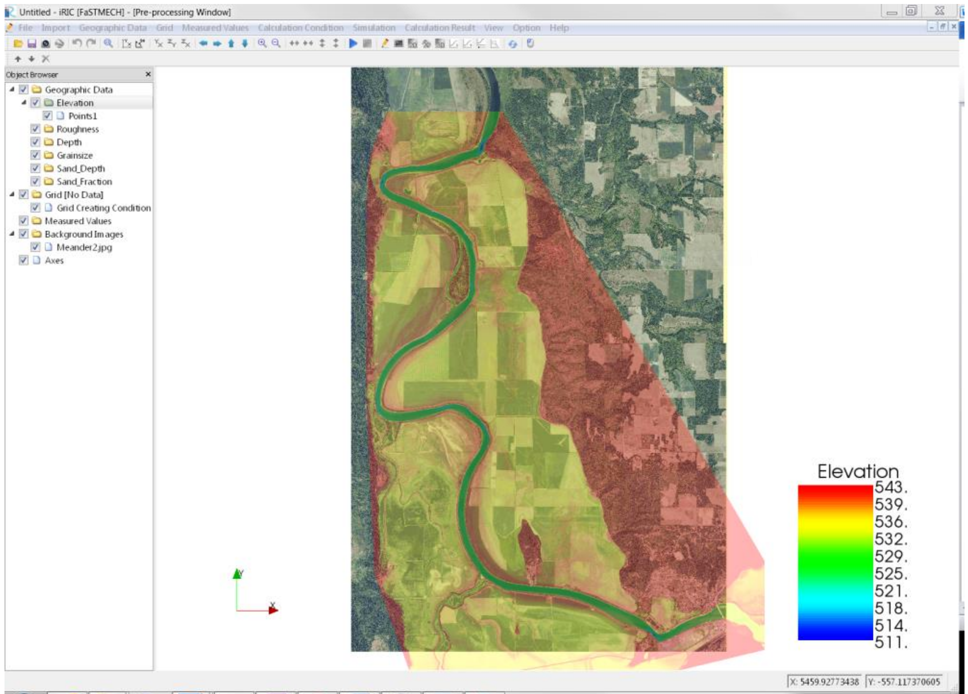


Figure 1. Kootenai River topography and image.

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 2.

Create the Curvilinear Grid:

- In the **Menu Bar** select Grid **->**Select grid creating algorithm. In the Select Grid Creating Algorithm dialog (figure 3), 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. Figure 4 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 (Cntrl+Middle Mouse) functions without affecting the placement of the grid center-line. In other words, using the Ctrl key to affect a Pan or Zoom will interrupt the center-line 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 stream-wise direction, n_I, the number of nodes in the cross-stream direction, n_J, and the width of the grid, W (Figure 5). 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 stream-wise and stream-normal directions. Select OK 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 6.

- 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.

- 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 () | Grid Creating Condition*. This is necessary to edit the grid. The center-line and points defining the center-line should be visible. When the mouse is placed over the center-line or over a center-line point, the mouse cursor changes to a closed hand, and if the left mouse is clicked and dragged, it will move the center-line or center-line point. Experiment with adjusting center-line 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 center-line until there is no overlap of the grid. Additional utilities to add or remove points are also available and will be discussed in Tutorial 1.

- Save the project.

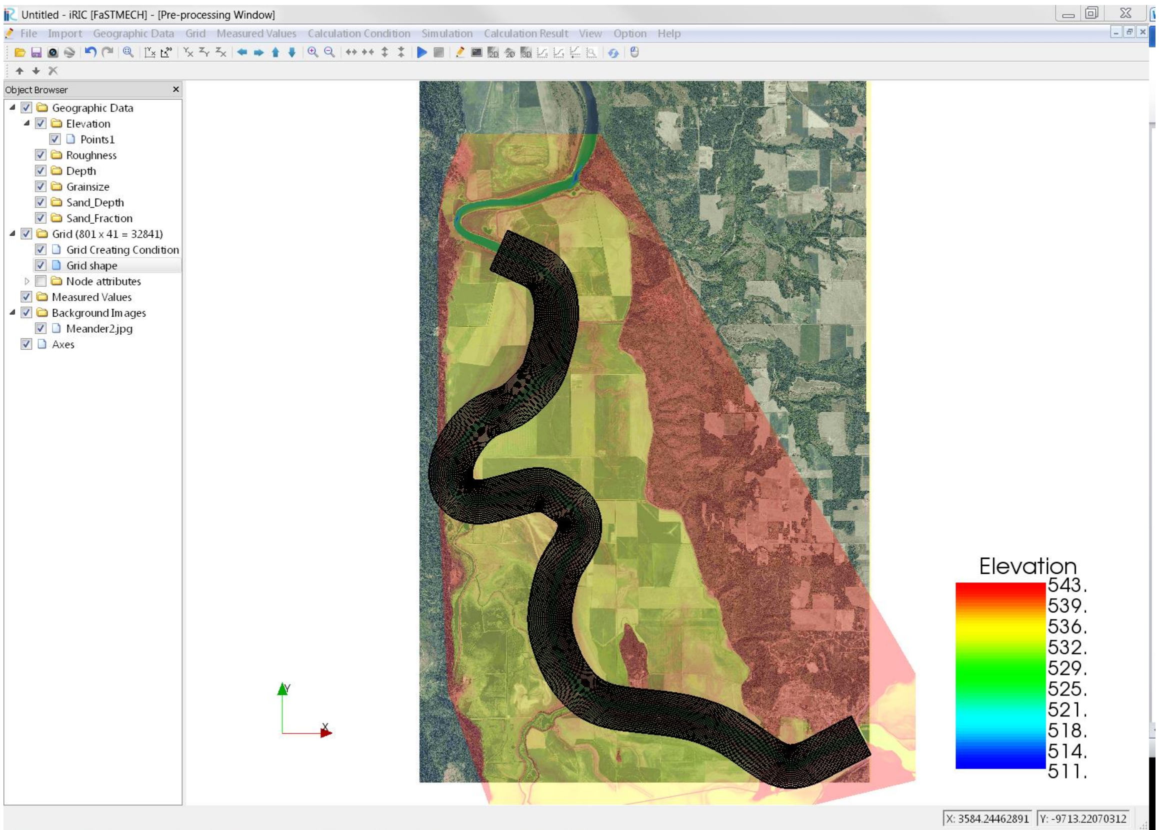


Figure 2. A user-defined grid following the channel of the Kootenai River.

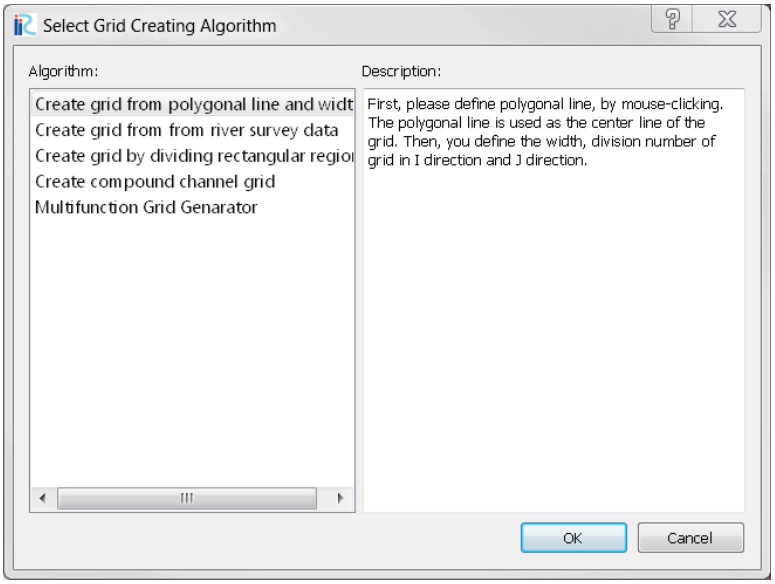


Figure 3. Select Grid Creating Algorithm dialog.

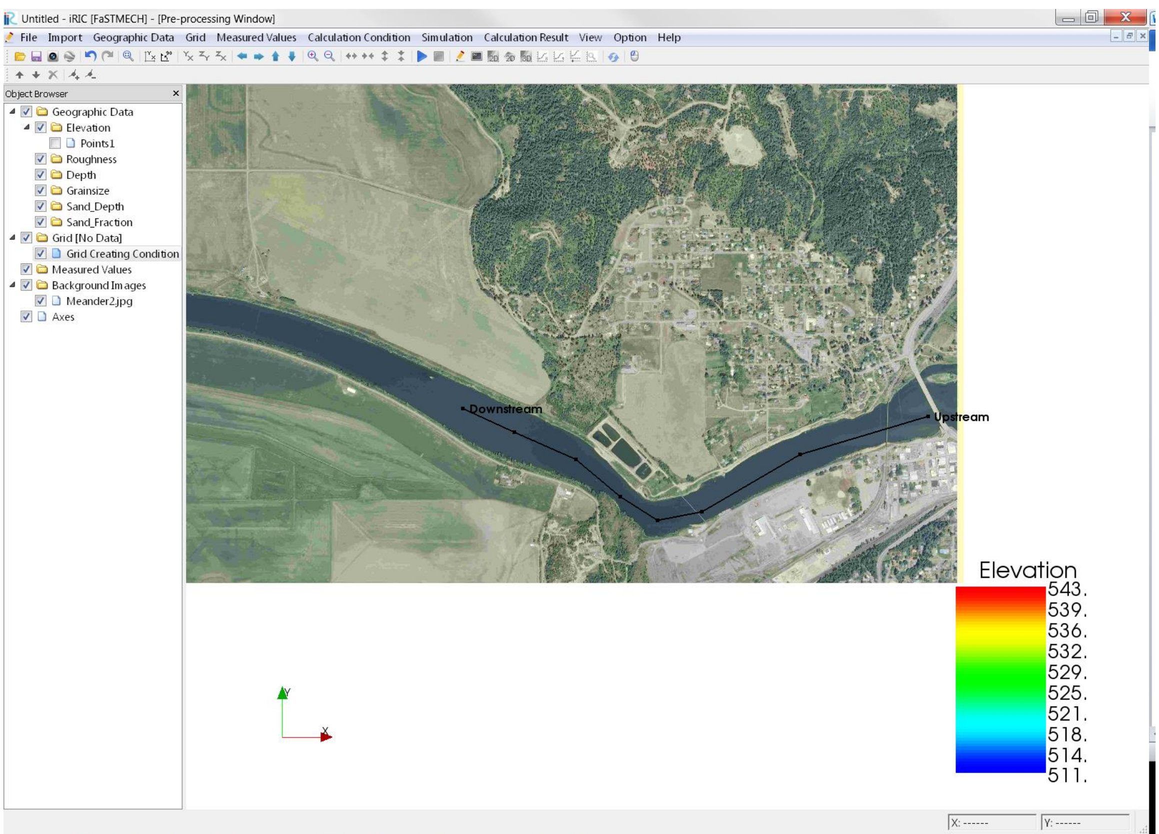


Figure 4. Process to draw the grid centerline. Note the Upstream and Downstream labels are defined.

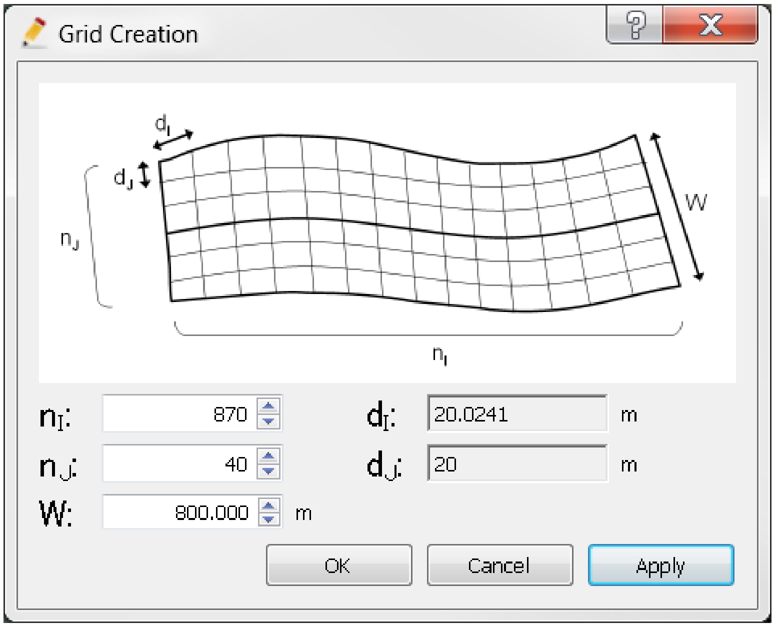


Figure 5. The Grid Creation dialog

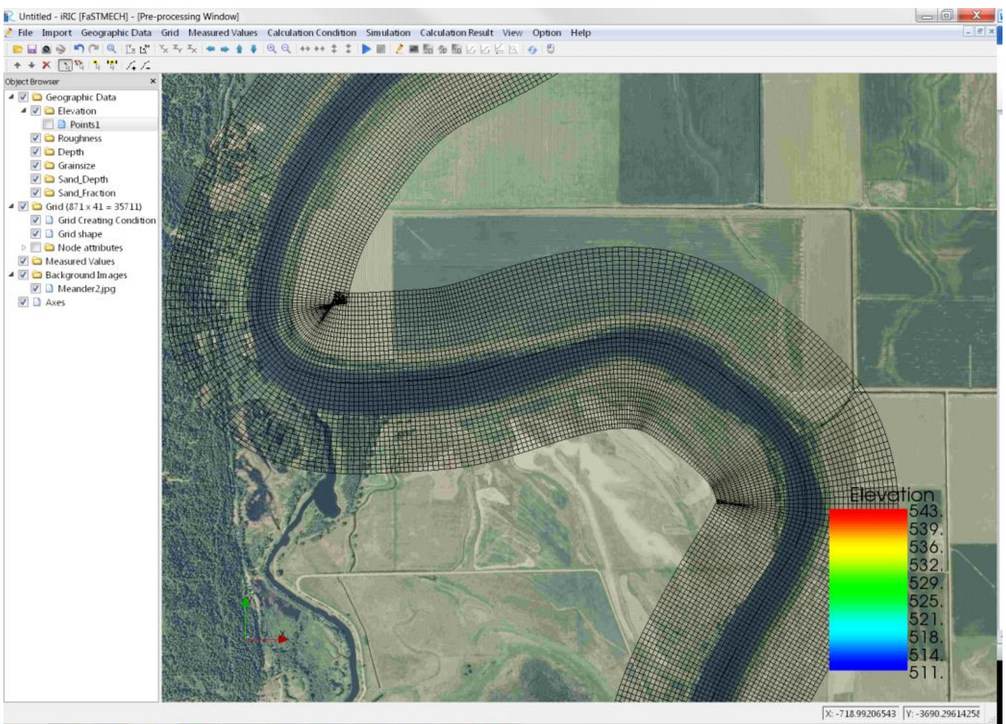


Figure 6. Curvilinear grid with grid overlap.

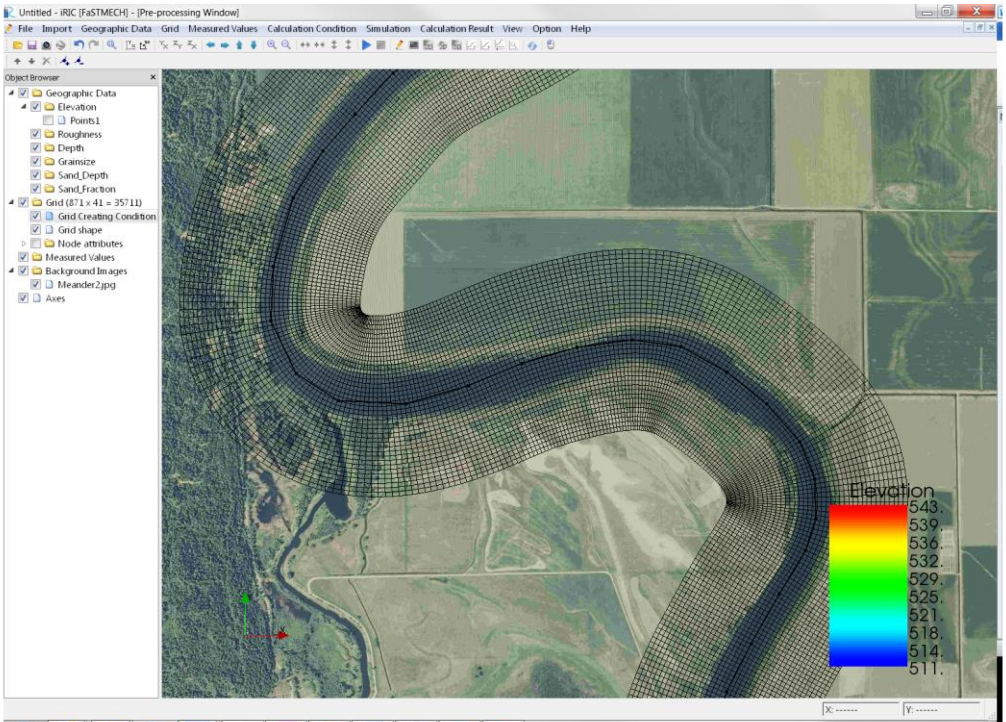


Figure 7. Grid after editing the location of centerline points to refine the curvature and eliminate overlapping grid nodes as shown in Figure 6.

Mapping Geographic Data To The Grid Attributes.

In the **Object Browser**, every branch in the *Geographic Data* tree has 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 however, 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 bin.

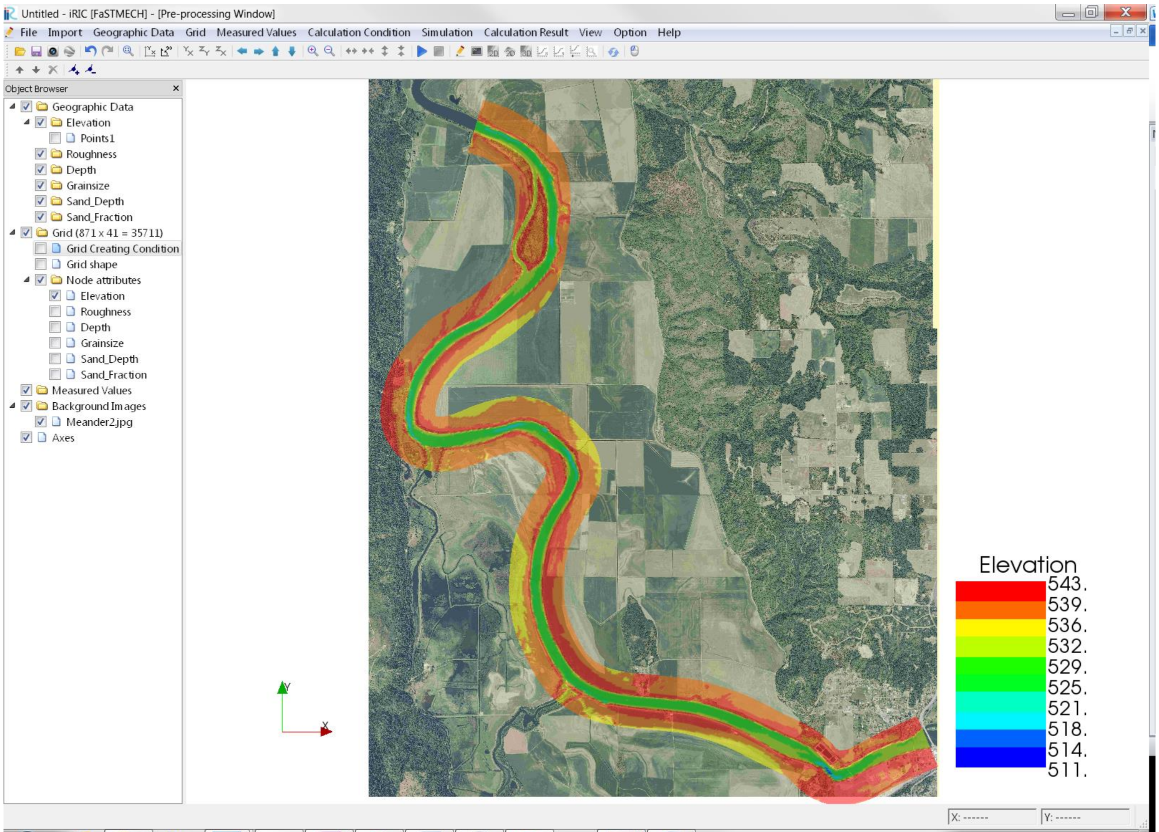


Figure 8. Elevation mapped to the grid using the TIN Algorithm.

Mapping With TINs

- In a previous step above we set the Grid **->**Attributes Mapping **->**Settings 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. 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 8).
- Save the Project.

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 geo-referenced image and stretch, rotate and translate the image so that it fits the measured topographic data

- From the Menu Bar select File **->**New Project. Select the FaSTMECH solver.
- In the Object Browser left-click then right-click on the Geographic Data | Elevation and in the resulting pop-up menu select Import and open the ketchupisl.tpo file.
- From the Object Browser right-click Background Images and select Add Image in the resulting pop-up menu and open ketchup2.jpg (Figure 9).

- To adjust the location of the image select Background Images | Ketchup2.jpg. In iRIC whenever you want to edit data whether it’s Geographic Data, Grid Creating, or Images, the object that is selected in the Object Browser is enabled for editing.
- When Background Images | Ketchup2.jpg is selected, the left mouse button will pan the image, the middle mouse button scales the image, and the left mouse button rotates the image. Experiment with these tools to fit the image to the data (Figure 10).
- From the Object Browser right-click Grid | Grid Creating Condition and in the resulting pop-up menu Select Algorithm for Creating Grid, and then select “Create grid from polygonal line and width”. Flow direction is from bottom to top. Draw a center line and create a grid with a width of 450 meters and discretization along the center line of 10 meters (Figure 11).
- From the **Menu Bar** select *Grid* **->***Attributes Mapping* **->***Setting*. In the Grid Attribute Mapping Setting dialog select the “Template mapping” option and then select the “Detail…” button. Set up the following dialog similar to Figure 12 and click ok.
- From the **Menu Bar** select *Grid* **->***Attributes Mapping* **->**Execute. View the results and experiment with different template dimensions to see the differences in the mapped topography (Figure 13). Save your best effort to compare with TIN mapping below. Click on in the Main Toolbar to save an image.
- Compare the template mapping to the TIN mapping. Go back to the Menu Bar and select *Grid* **->***Attributes Mapping* **->***Setting*, and set the mapping algorithm to TIN. Execute the Tin Mapping by selecting from the Menu Bar *Grid* **->***Attributes Mapping* **->**Execute. Click on in the Main Toolbar to save an image and compare with the image taken of the Template Mapping above (See Figure 13 and 14).
- Save the file.

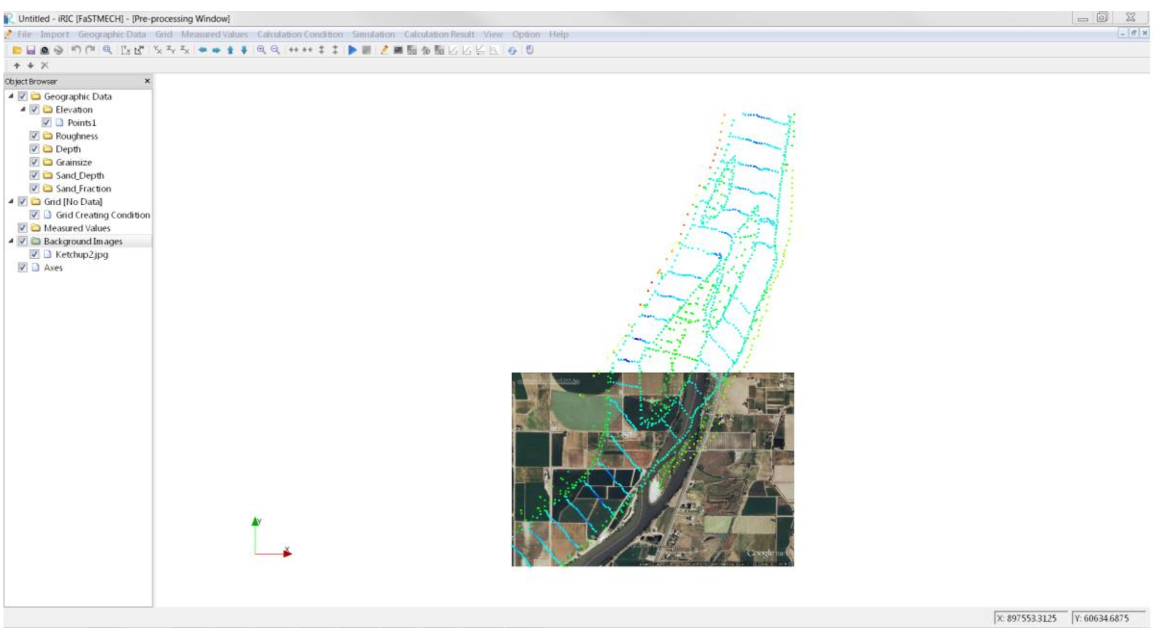


Figure 9. A non-geo-referenced image imported into iRIC.

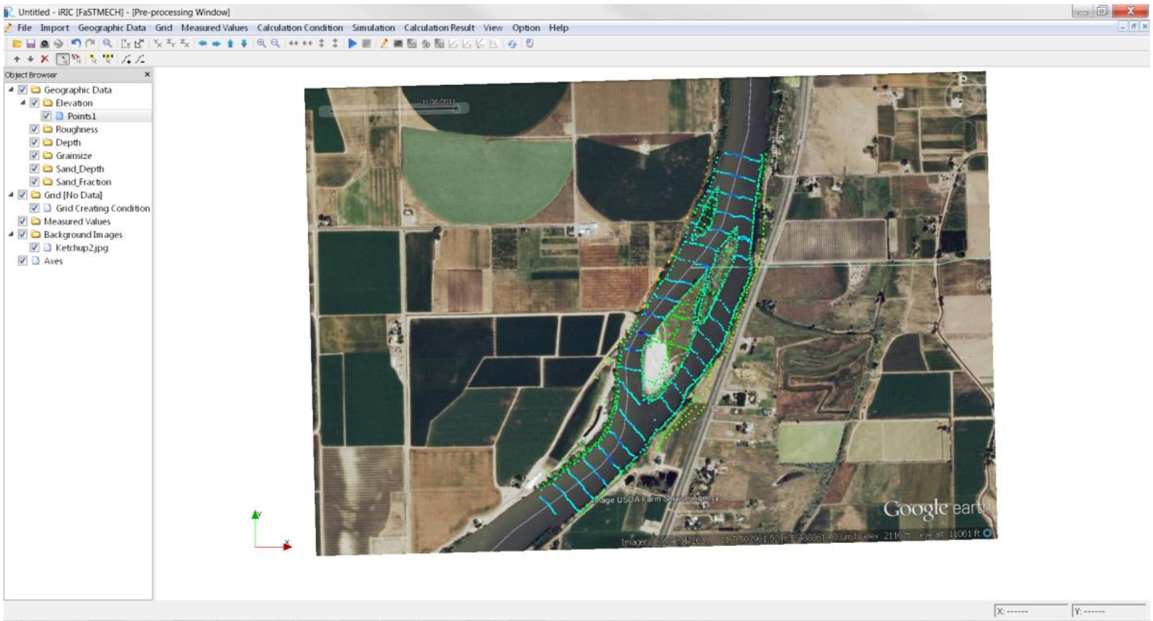


Figure 10. The background image fit to the data.

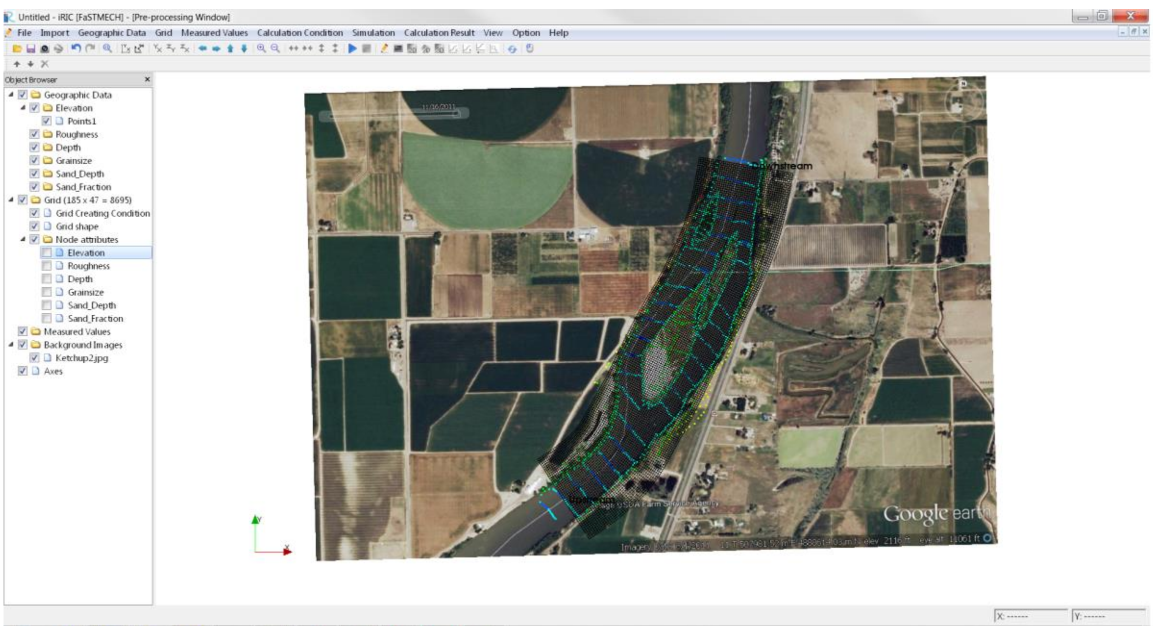


Figure 11. Location of the grid.

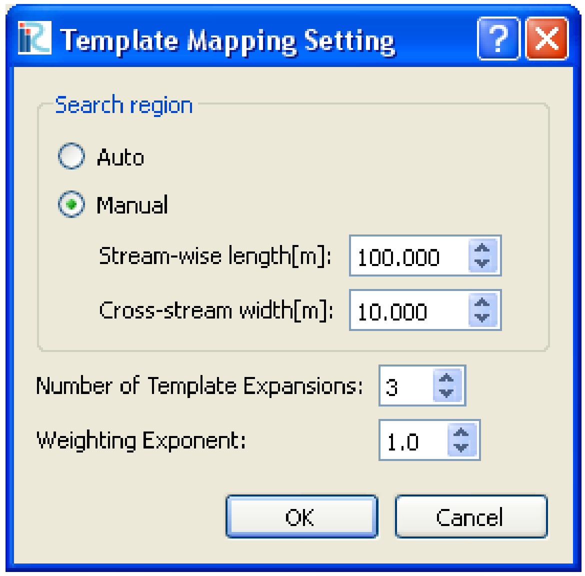


Figure 12. Template mapping dialog

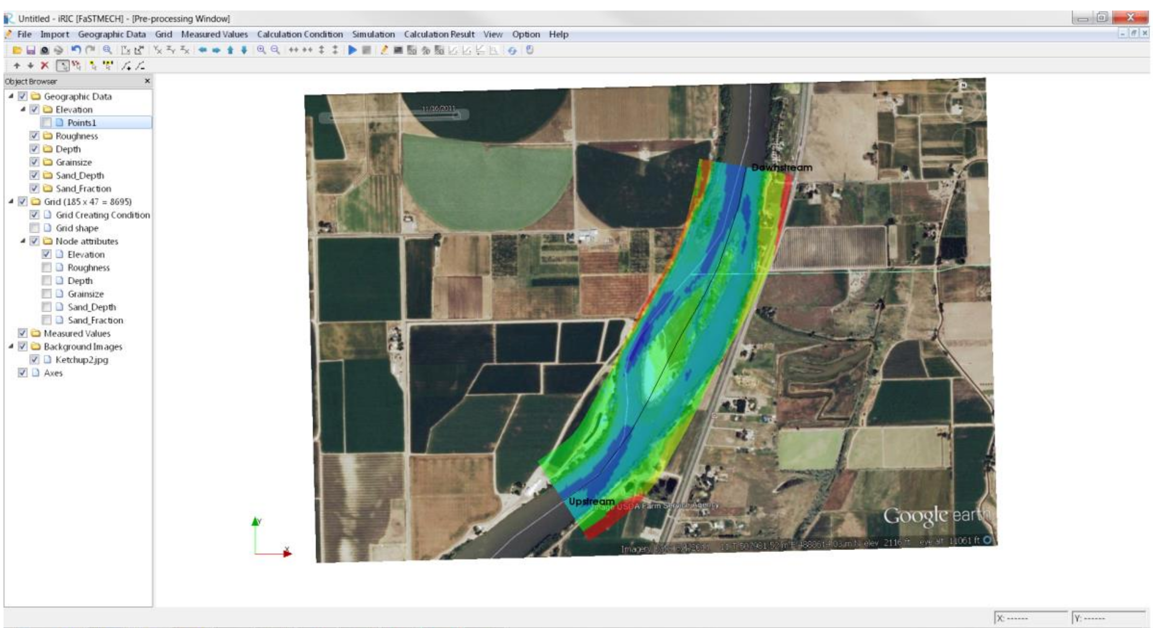


Figure 13. Elevation mapped to the grid with the template algorithm.

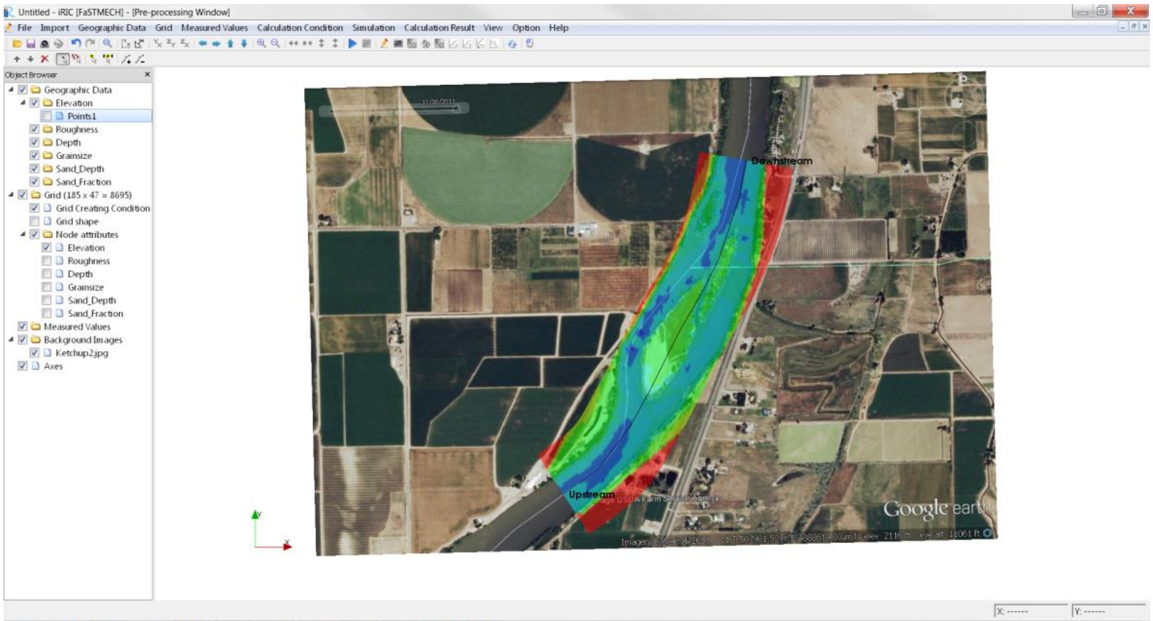


Figure 14. Elevation mapped to the grid with the TIN algorithm

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, the Operation Toolbar (Figure 5)` shows a suite of tool that can be used to add points using a simple point adding tool , interpolating between known points , or setting breaklines to the TIN. 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.
  - Import the geo-referenced 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 TIN with a result that should look similar to that shown in Figure 15.
- Figure 16 shows a zoomed-in image of the downstream, river-right section of the grid, shown as a red box in Figure 15. Because of the way the TIN algorithm creates triangles, cusps can be formed along the banks of the channel as shown by the zig-zag line in Figure 16.
- 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 | Points1 and in the resulting pop-up menu select Property and then in the Display Setting dialog select Wireframe as the Display Method (Figure 17). 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.
  - **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. Also the value along the breakline is a linear interpolation of the bounding points. We will be editing Geographic Data | Elevation | Points1 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).
    - Select the breakline tool in the Operation toolbar () and draw a break line between the four points shown in Figure 18. 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. 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 19.
  - **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 | Points1. This enables the data editing toolbar (See Exercise 1 – Figure 5).

- Select the interpolation tool . 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".
- Right-click anywhere in the Graphics View and in the resulting pop-up menu select "Remesh TINs". Right-click on Geographic Data | Points 1 in the **Object Browser** and select Property. Change the Display method to Wireframe and view the results (Figure 20).
- From the Menu Bar select Grid->Attribute Mapping -> Execute. View the result in Figure 21 and compare with Figure 19.

- Save the project.

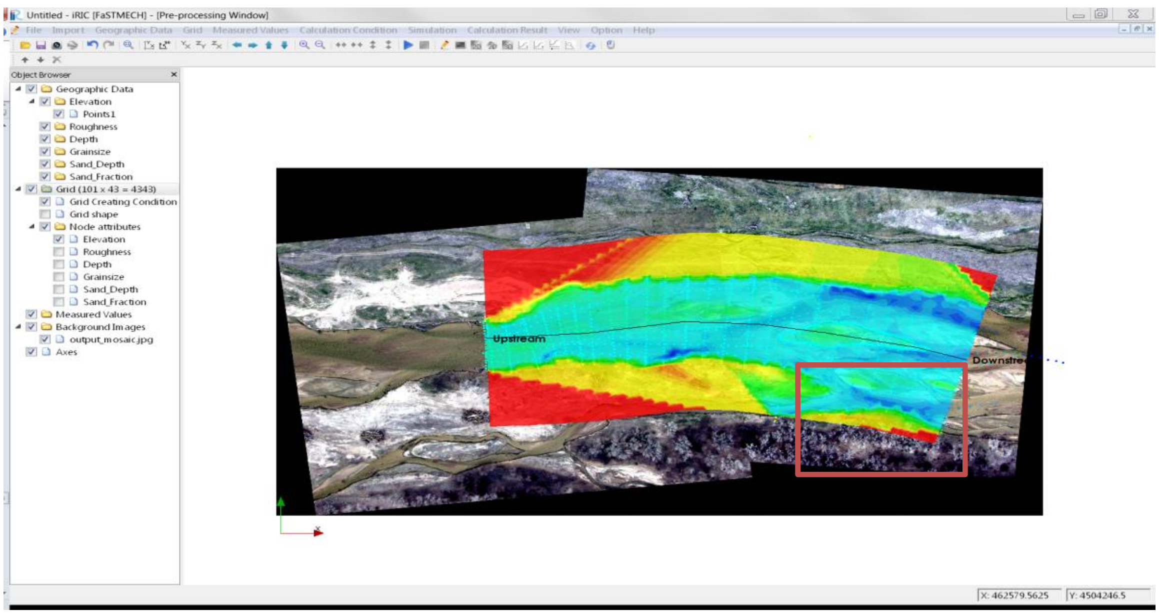


Figure 15. Location (red box) of reach that displays cusping along the bank.

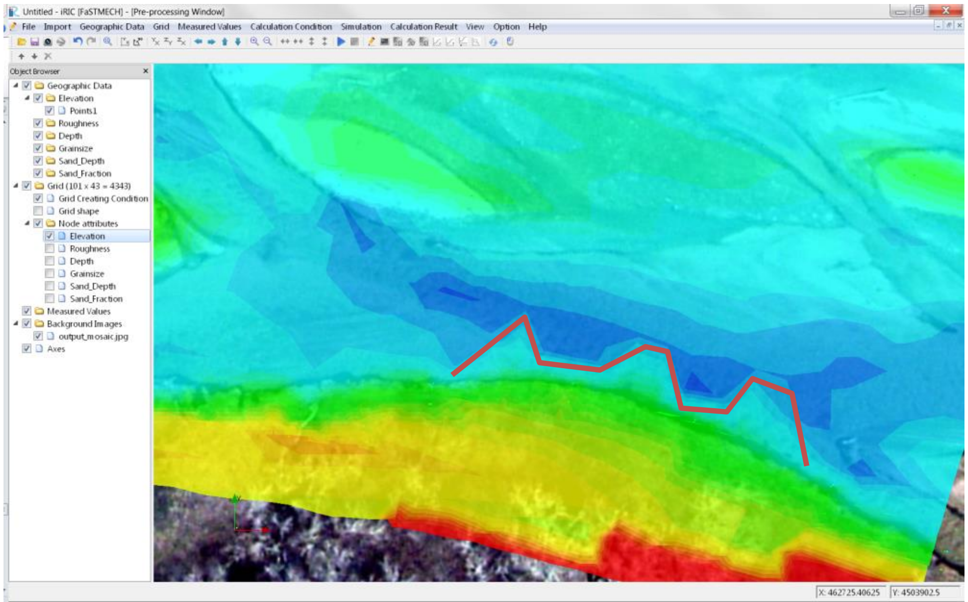


Figure 16. Cusps along the bank from the TIN algorithm

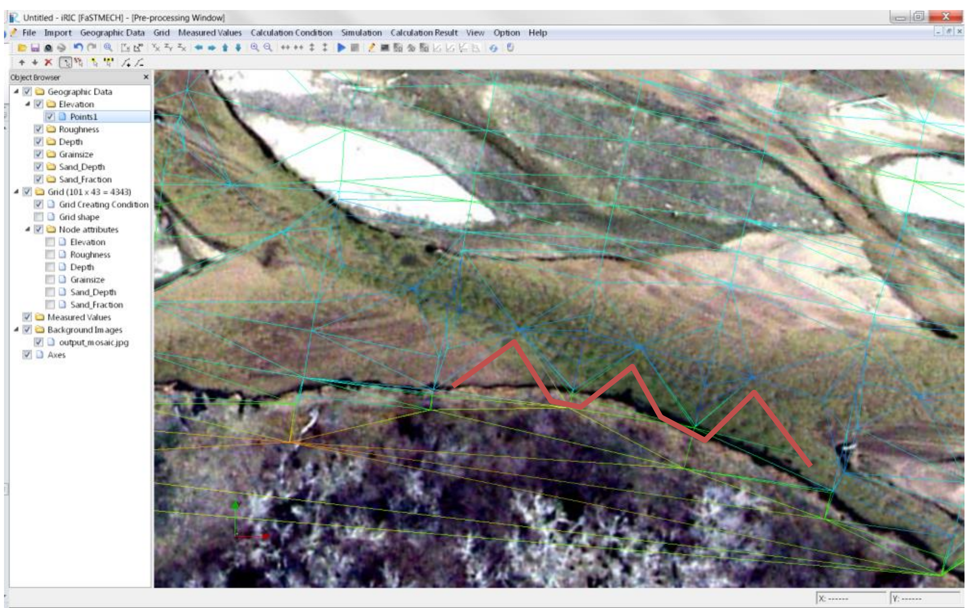


Figure 17. The TIN and location of cusps.

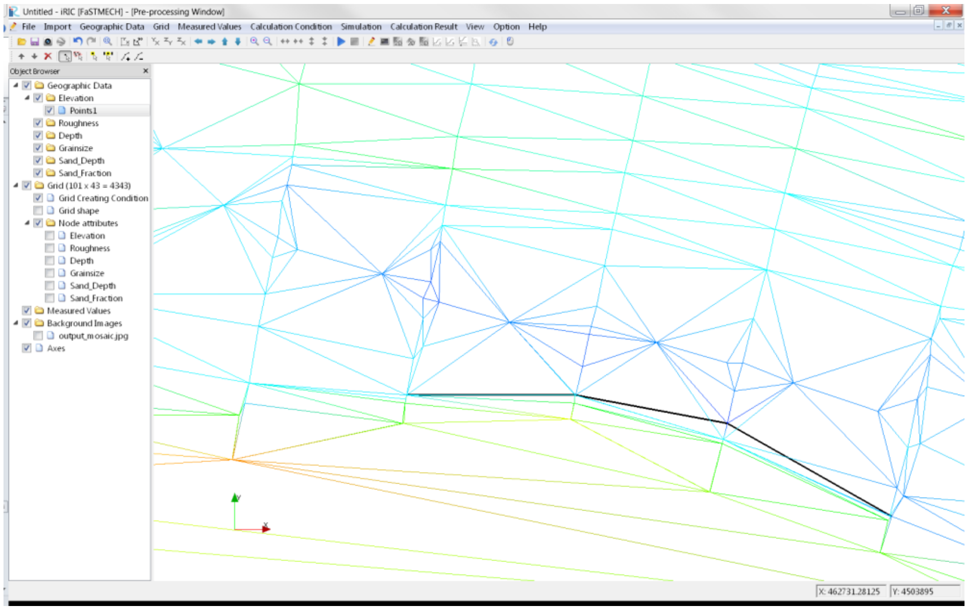


Figure 18. Location of breakline (black line).

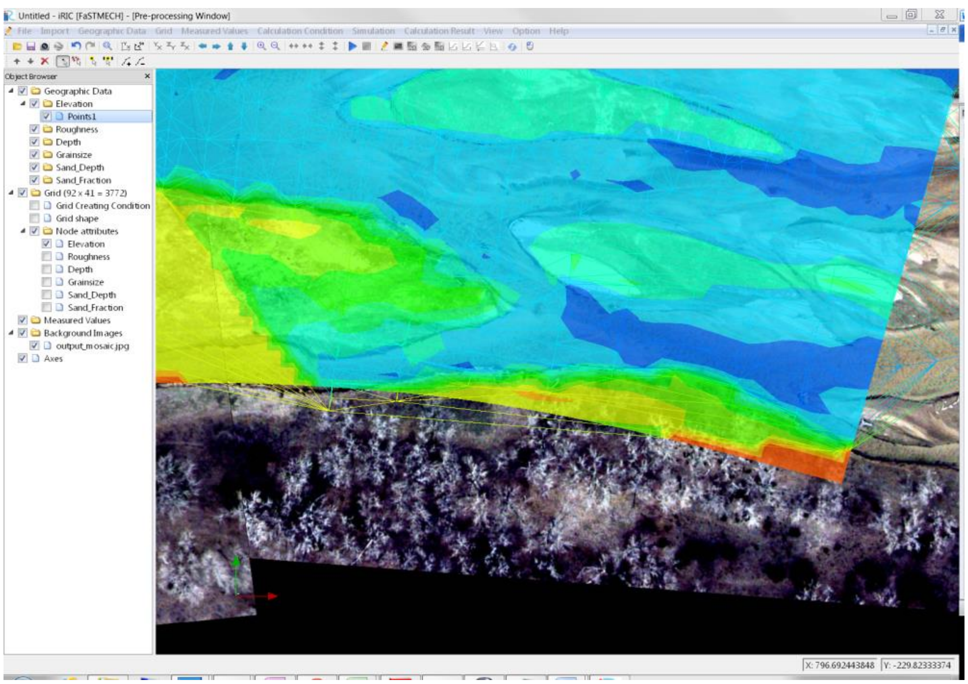


Figure 19. Result of mapping elevation with the TIN and breakline. Note cusping is gone. Compare to Figure 16.

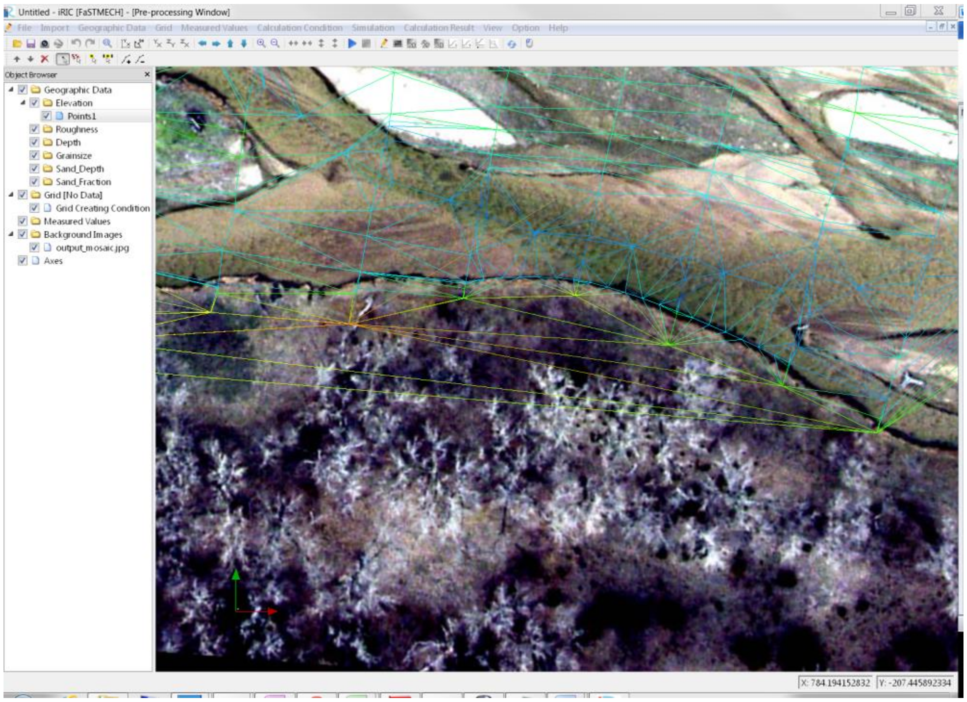


Figure 20. TIN after creating an interpolation line along the bottom of the bank and Re-meshing the TIN

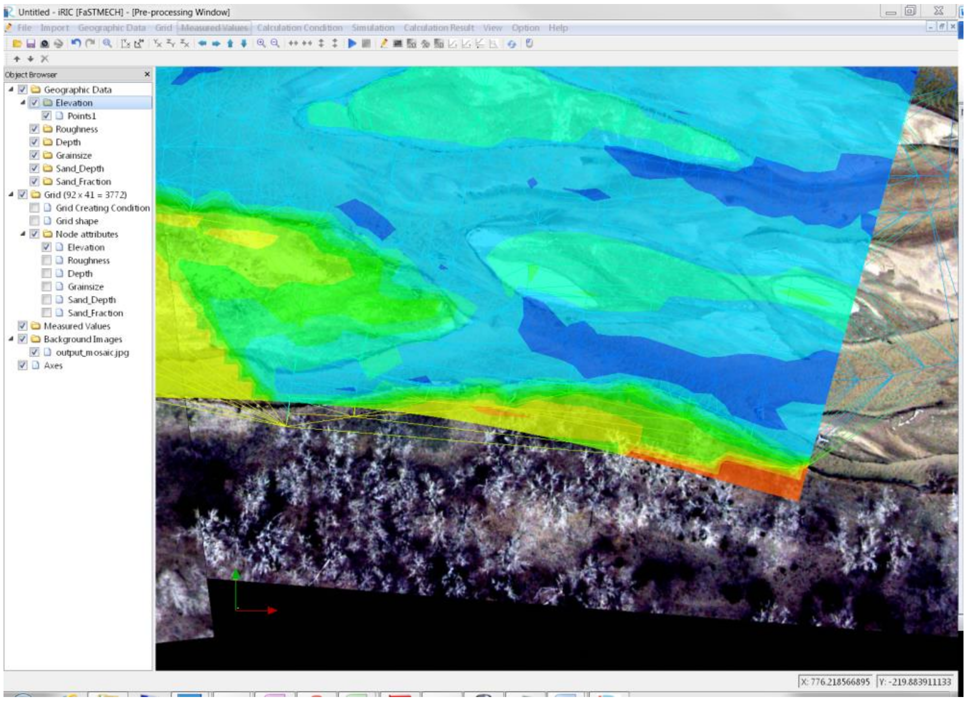


Figure 21. The topography mapped to the grid following the addition of an interpolation breakline.

Compare with Figure 19.

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.

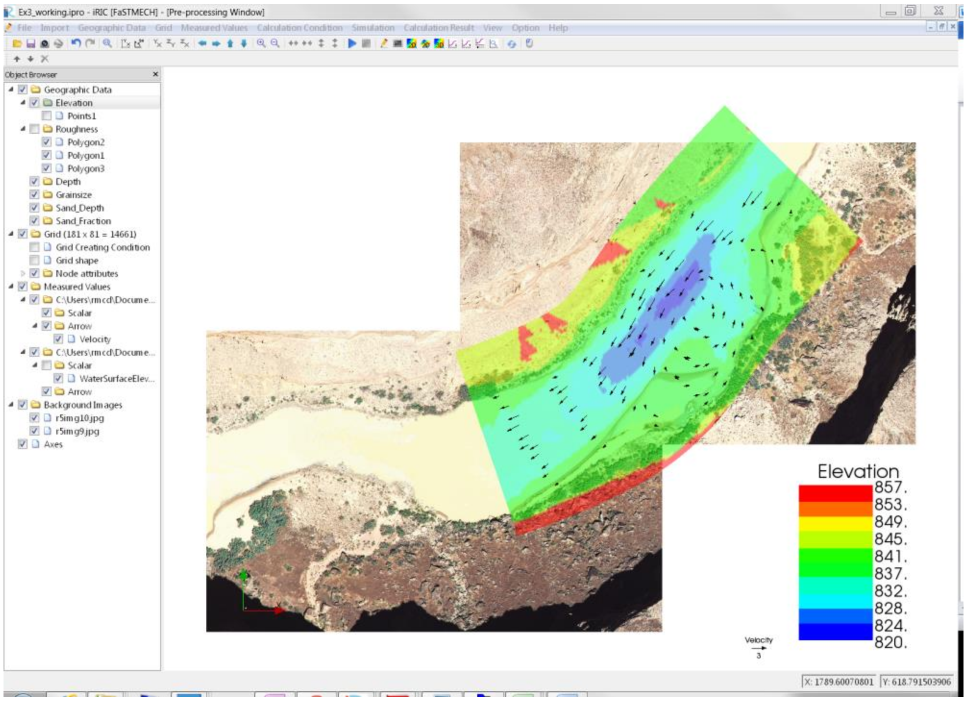
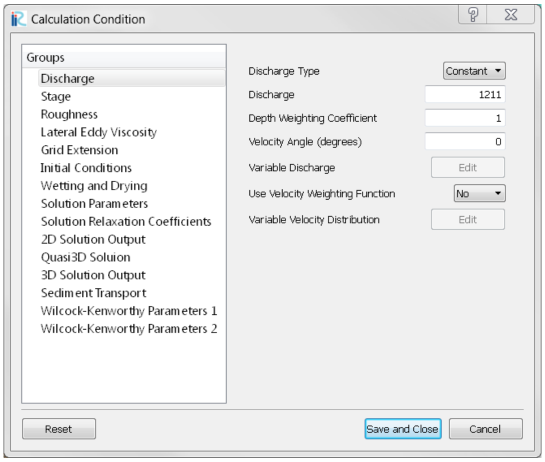
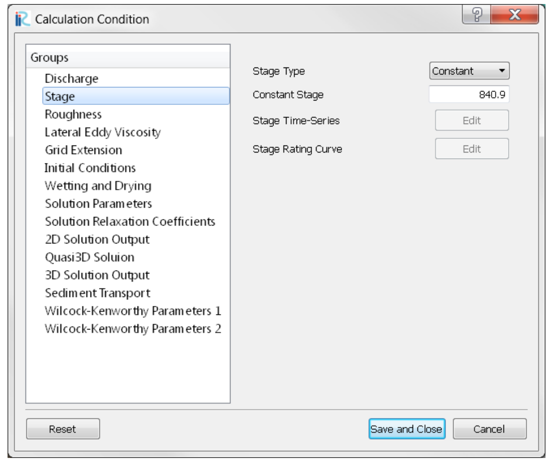


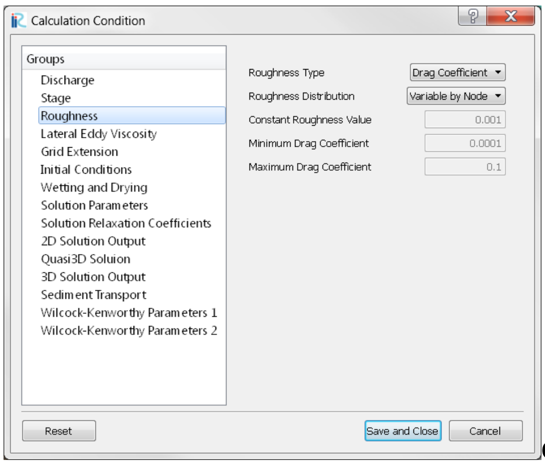
Figure 1. Domain for Exercise 3 including the measured velocity and gridded elevation.

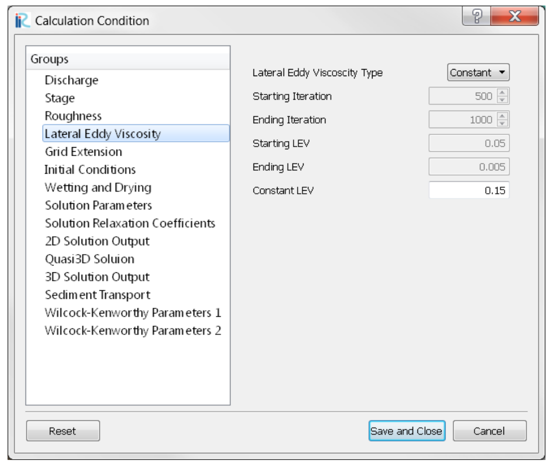
Defining Calculation Conditions

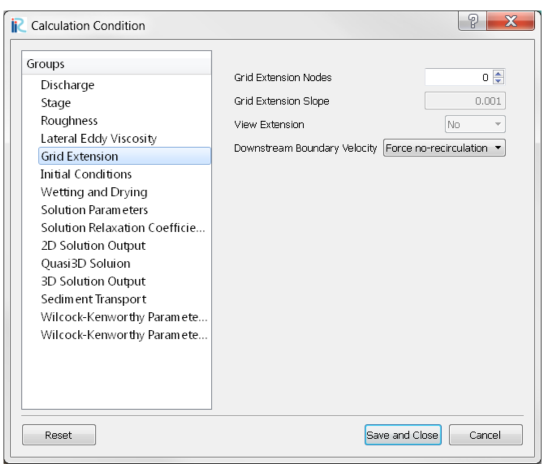
- Open an existing project from Menu Bar by selecting *File* **->***Open* and select the Ex3.ipro project file in the Tutorial 1\Exercise 3 folder (Figure 1).
- To define the calculation conditions select Menu Bar select *Calculation Conditions* **->***Setting.* Enter the parameters as shown in Figure 2. Select Save and Close when you are done.

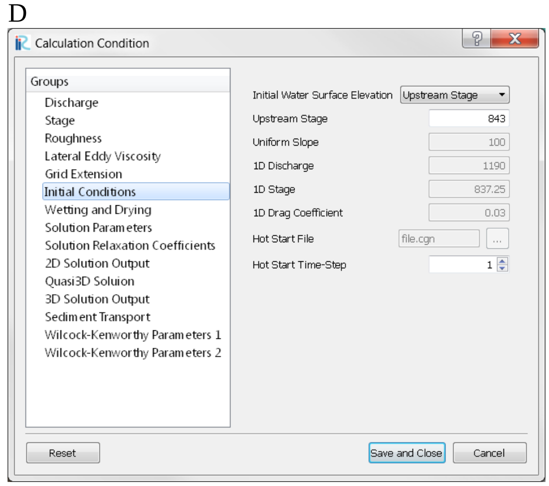

Calculation Condition
Groups
Discharge
Stage
Roughness
Lateral Eddy Viscosity
Grid Extension
Initial Conditions
Wetting and Drying
Solution Parameters
Solution Relaxation Coefficients
2D Solution Output
Quasi3D Soluion
3D Solution Output
Sediment Transport
Wilcock-Kenworthy Parameters 1
Wilcock-Kenworthy 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 Soluion
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 Soluion
3D Solution Output
Sediment Transport
Wilcock-Kenworthy Parameters 1
Wilcock-Kenworthy Parameters 2
Roughness Type Drag Coefficient
Roughness Distribution Variable by Node
Constant Roughness Value 0.001
Minimum Drag Coefficient 0.0001
Maximum Drag Coefficient 0.1
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 Soluion
3D Solution Output
Sediment Transport
Wilcock-Kenworthy Parameters 1
Wilcock-Kenworthy Parameters 2
Lateral Eddy Viscocsity 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 Coefficie...
2D Solution Output
Quasi3D Soluion
3D Solution Output
Sediment Transport
Wilcock-Kenworthy Paramete...
Wilcock-Kenworthy Paramete...
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 Soluion
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

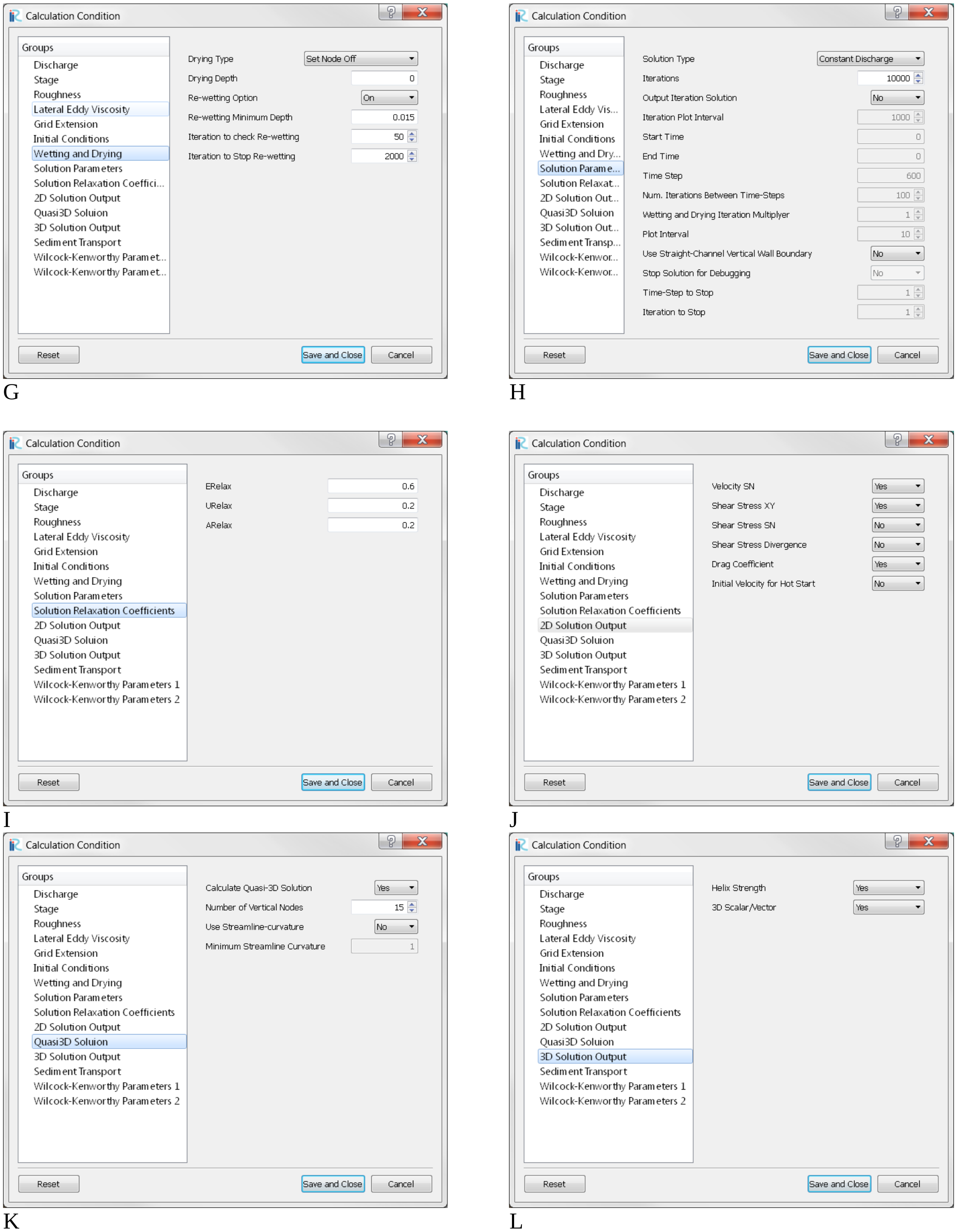


Figure 2. A. Discharge, B) Stage, C) Roughness, D) Lateral Eddy Viscosity, E) Grid Extension, F) Initial Condition, G) Wetting and Drying, H) Solution Parameters, I) Solution Relaxation Coefficients, J) 2D Solution Output, K) Quasi 3D Solution, L) 3D Solution Output..

- 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 OK. 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.

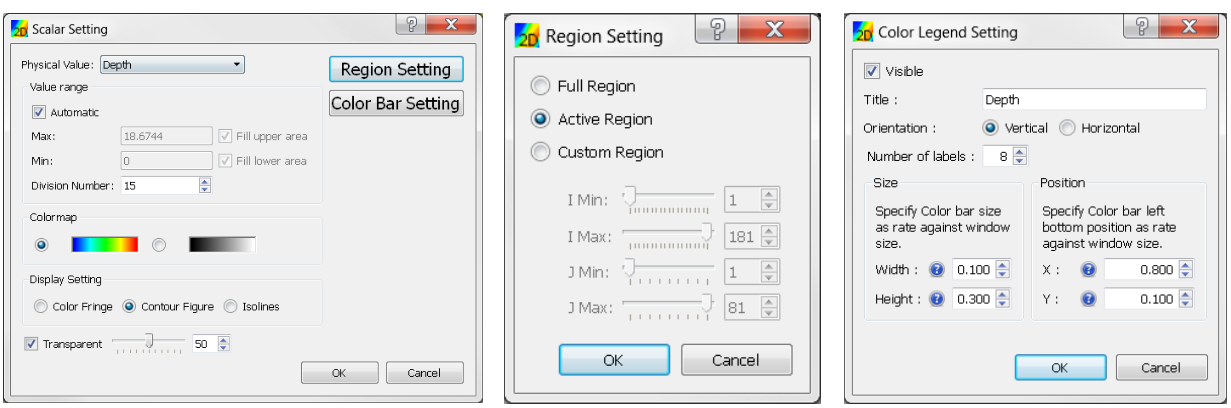
2D Post-processing

Calculation results can be viewed by opening a new 2D Post-processing Window by selecting *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.

- To begin with in the Object Browser turn off Measured Values.
- In the Object Browser select the check box next to 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 and in the pop-up menu Select Property, and in the resulting Scalar Setting dialog (Figure 3A) select the Region Setting button.
    - In the Region Setting dialog (Figure 3B) select the Active Region radio button and then select OK.
    - In the Scalar Setting Dialog set the Display Setting Attribute to Contour Figure and take note of other attributes that can be set from this dialog including the min, max, and number of intervals for the contour. Select OK when finished.
- In the Object Browser turn the Background Images on. The result should look like Figure 4.
- Change the display of the legend or scalebar by selecting the Color Bar Setting button in the Scalar setting dialog (Figure 3C).



A B C

Figure 3. A) Scalar Setting dialog and associated B) Region Setting, and C) Color Legend Setting dialogs.

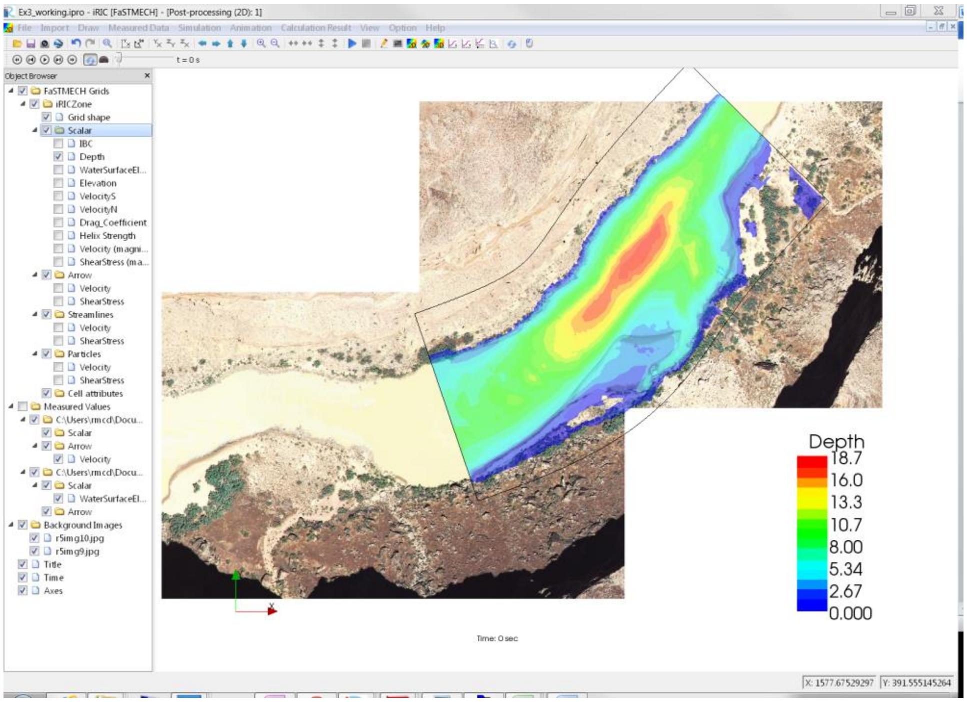


Figure 4. Scalar solution of Depth.

Displaying 2D Vector Results

- In the Object Browser turn on 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 5A) 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- direction. The result is shown in Figure 5B.
    - Alternatively set the Arrow Setting attributes as in Figure 5C with a result shown in Figure 5D.

Compare Measure Velocity Vectors to Solution Vectors

- In the previous step you set the Arrow Setting attributes as in Figure 5D. 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\:(your 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 5C such that both the measured and solution vectors have the same scale. Change the Custom Color attribute to red.
- The result is shown in Figure 6.

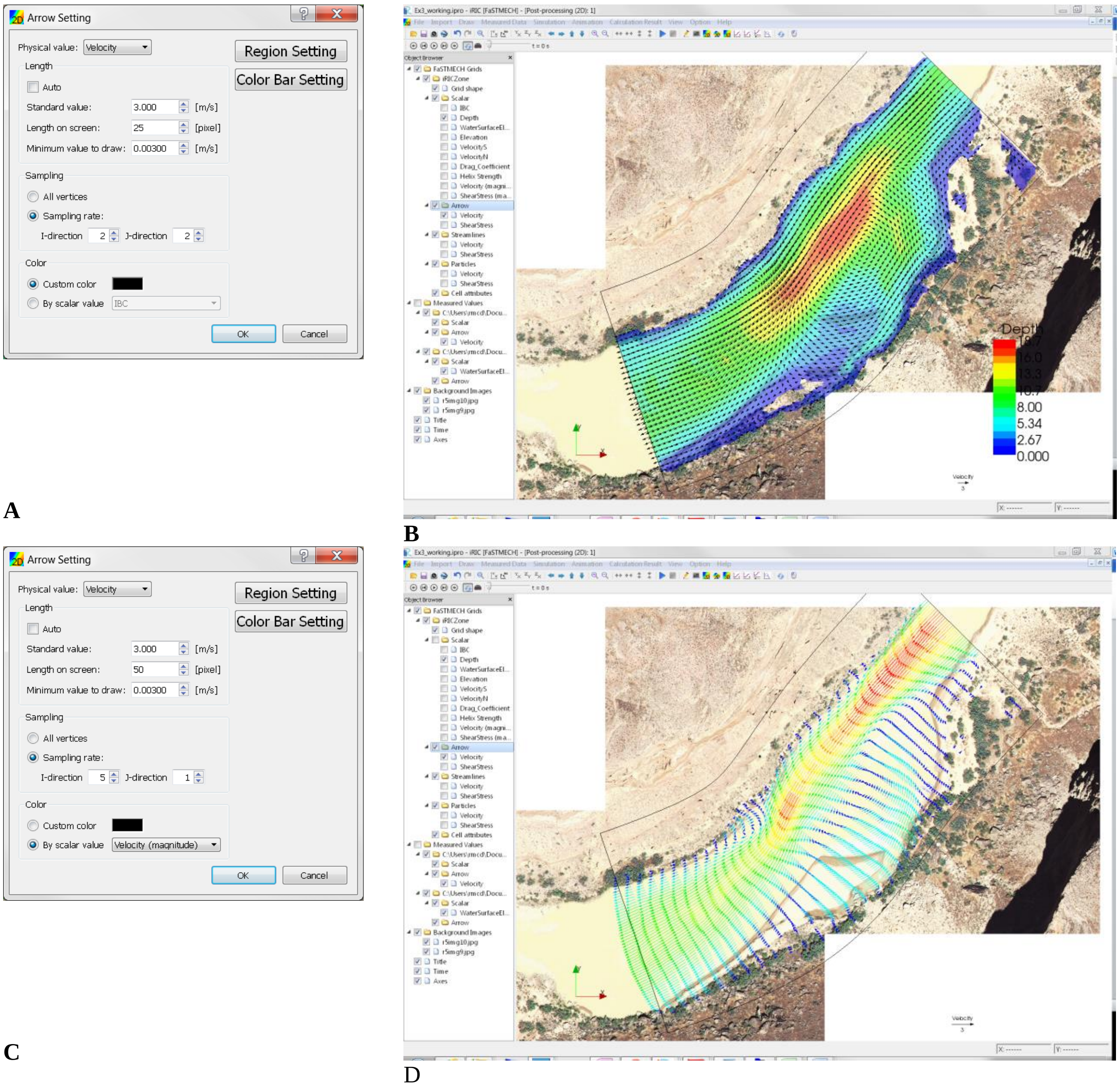


Figure 5. Arrow setting attributes and resulting plot of vectors.

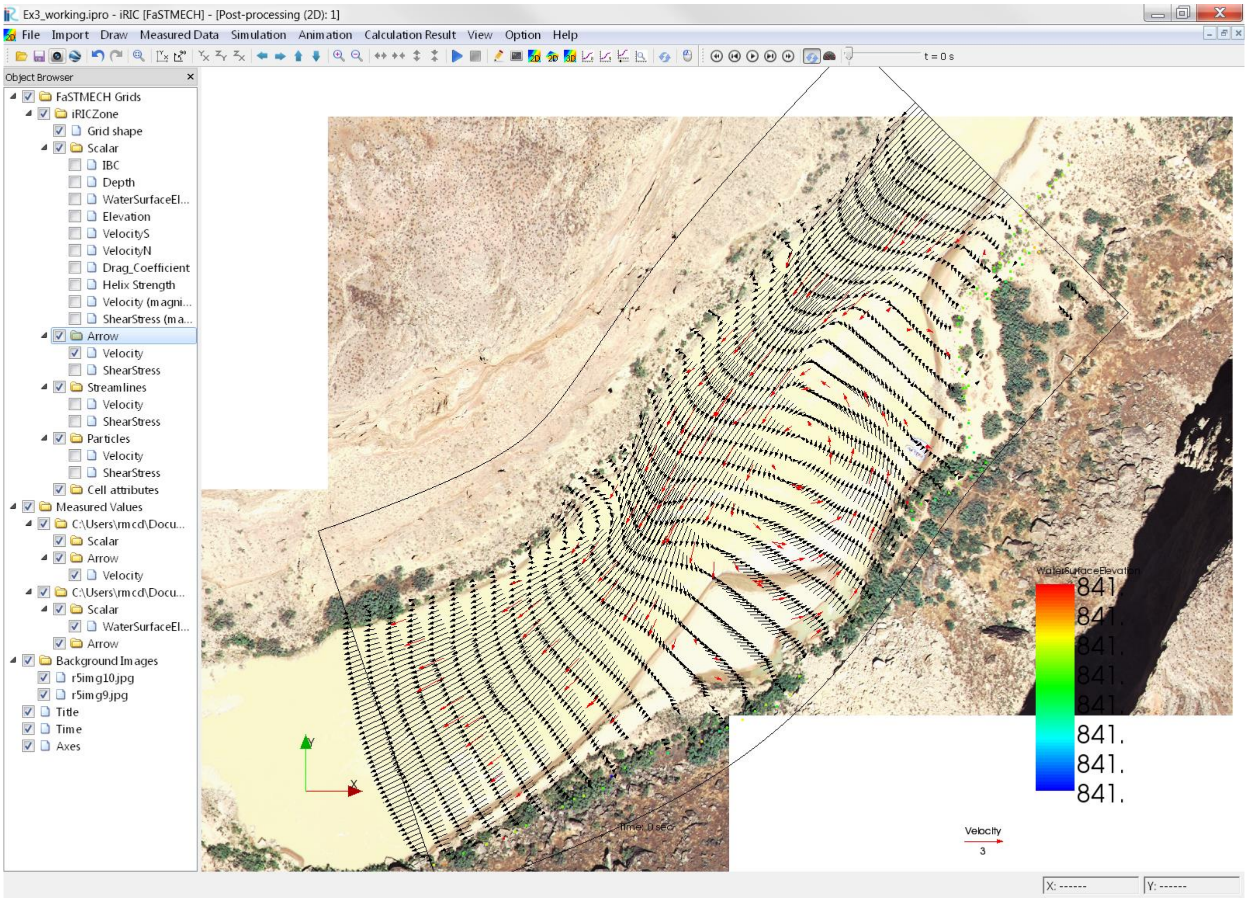


Figure 6. Solution vectors (Black) and measured vectors (Red).