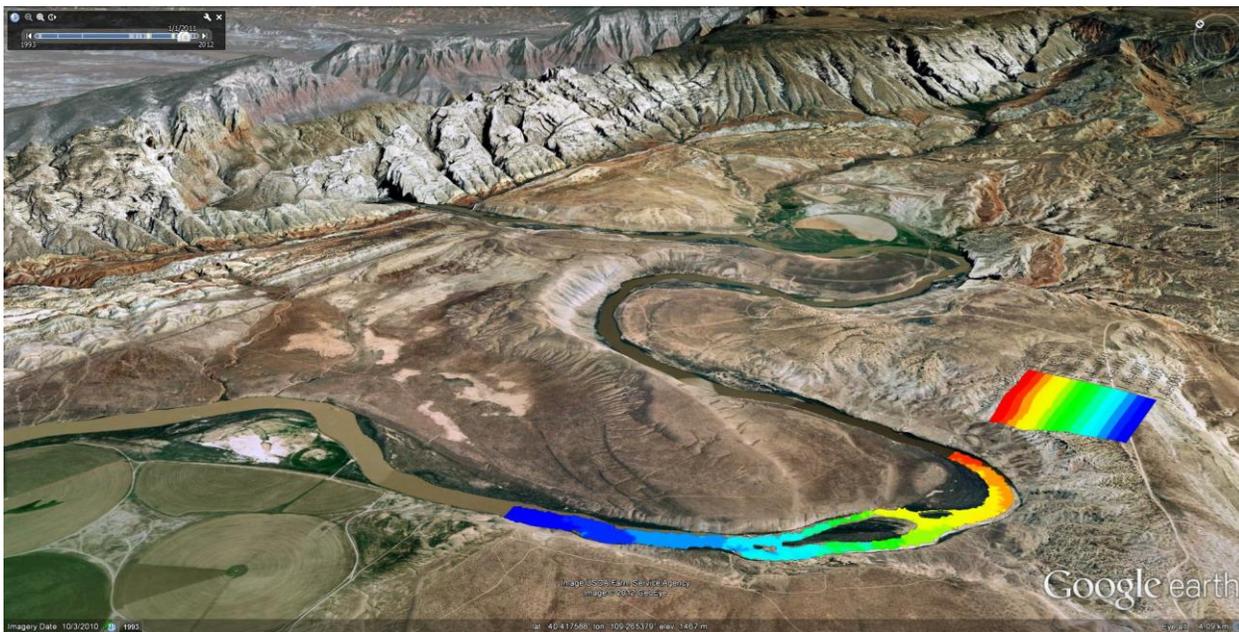




**iRIC Software**

*Changing River Science*

# ***FaSTMECH Tutorial 2***



Green River, Dinosaur National Park, Colorado, USA

# Contents

Introduction.....	2
Getting Started .....	2
Pre-Processing .....	3
Importing Topography.....	3
Importing Measured Water Surface Elevations.....	5
Creating a Numerical Grid.....	5
Create grid:.....	6
Mapping geographic data to the grid attributes. ....	9
Defining the Calculation Conditions and Running a Simulation .....	11
Post-processing.....	13
Map Visualizations .....	13
Displaying Scalar Results.....	13
Displaying Vector Results.....	16
Model Calibration and Refinement.....	17
Developing a Calibration Curve .....	17
Spatially Varying Drag Coefficient.....	19

## Introduction

In this tutorial you will use the FaSTMECH solver in the iRIC application along with measured topography and water-surface elevations to model flow through a reach of the Green River in Utah. This tutorial will demonstrate the following tasks necessary to complete a simulation:

- Getting Started
  - Introduction to navigating the iRIC interface
- Pre-processing
  - Importing geographic information
  - Creating a numerical grid
- Defining the Calculation Conditions and Running a Simulation
- Post-processing
  - Visualizing results
  - Graphing results
- Model Calibration and Model Refinement
  - Developing a calibration curve
  - Spatially variable roughness

## Getting Started

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:

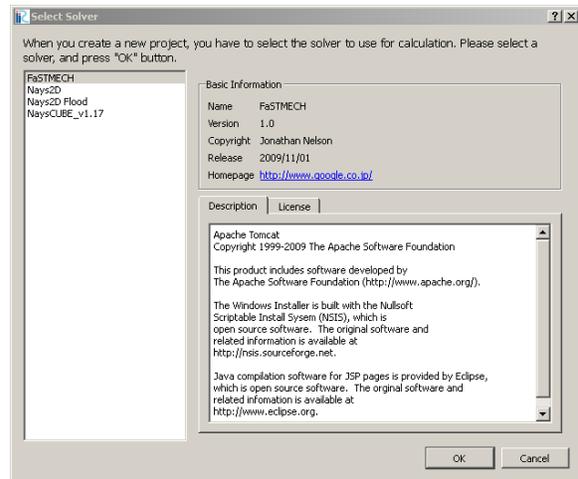
**Create New Project**—allows you to select the solver you wish to use from a list of solvers currently available in the application or to select the solver from a list of recently used solvers.

**Open Project File**—allows you to open an existing project using a browser window or to select from a list of recent projects.

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).

## Pre-Processing

### Importing Topography

The measured topography is the most important piece of information required to create a numerical model of the river reach of interest. Ideally the elevation data will extend well beyond the specific area of interest and be of sufficient spatial resolution for the application.

- Topography can be imported through the **Menu Bar** by selecting File ► Import ► Geographic Data ► Elevation. In the Select File to Import dialog, navigate to the following folder: FaSTMECH \ Tutorials\Tutorial 2. iRIC can import several different file formats; for this tutorial please select “.tpo” in the Files of Type drop down menu and select the following file: “GR\_Topo\_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 in the canvas and new data appears in the **Object Browser** under Geographic Data | Elevations | Points1 (Figure 2). In the Object Browser the topography can be made visible or not visible by checking or unchecking the box next to Elevation. Add a scalebar from the **Menu Bar** by selecting Geographic Data ► Set up Scalarbar. Make sure Elevation is displayed in the drop down menu and check the Visible box (Figure 2).

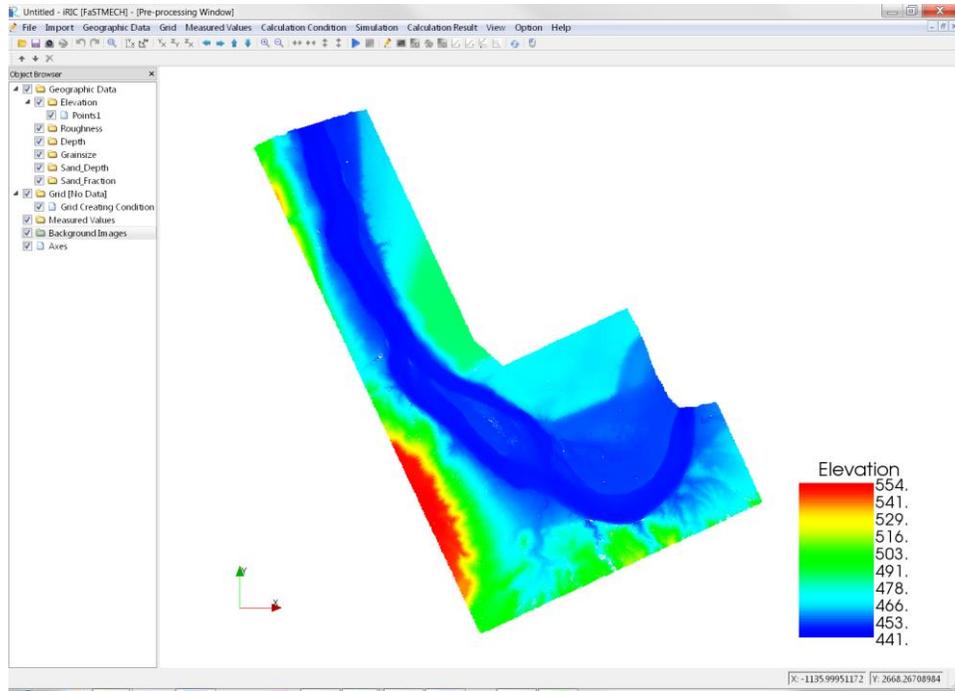


Figure 2. Pre-processing Window display of the Elevation data.

- To adjust how the elevation points are displayed, highlight Elevation | Points1 in the **Object Browser** and right click. This will bring up a dialog that allows you to rename the data in the Object Browser, Export the data, Delete the data, and adjust properties. Select properties and change the point size to 2 (Figure 3).

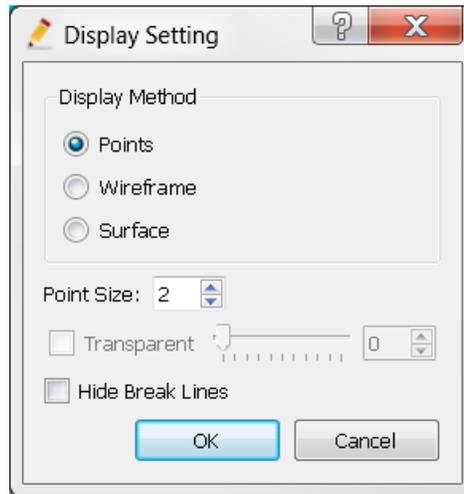


Figure 3. Display Setting dialog to select graphic attributes for the Elevation data

- Save (for example, Tutorial 2) by selecting File ► Save as File (\*.ipro) from the Menu Bar.

- Import an image to place in the background of the data. Background images can be imported through the **Menu Bar** by selecting **Import ► Geographic Data ► Background Image**. In the Open Image dialog select the **Green River.jpg** file.. The image will be displayed in the canvas (Figure 4). To turn the image off, uncheck **Background Images** in the **Object Browser**.

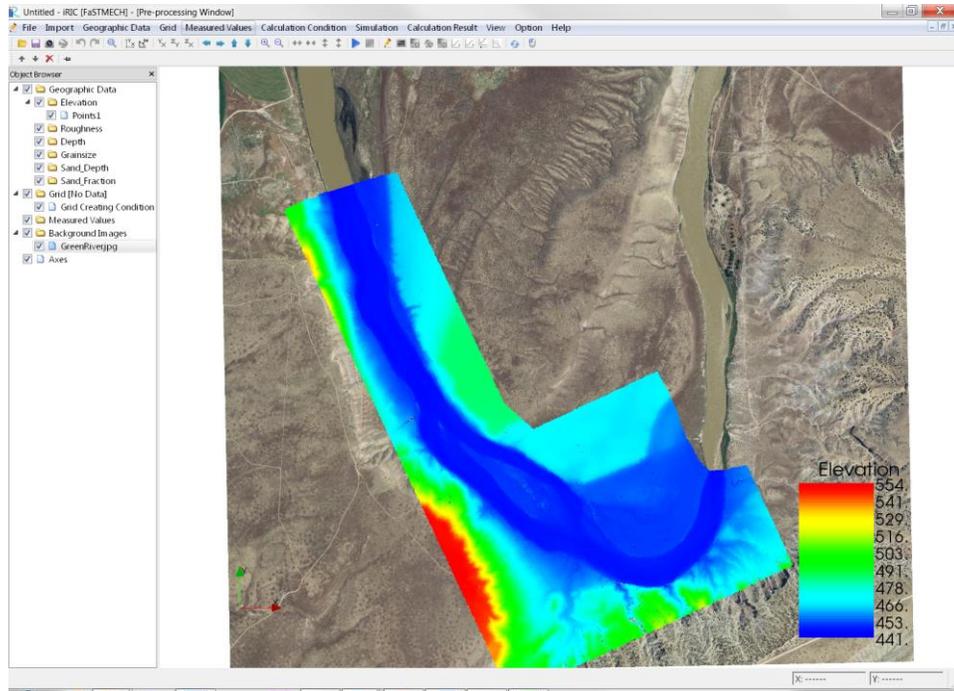


Figure 4. Addition of the background imagery.

## Importing Measured Water Surface Elevations

- To verify the modeled water-surface elevation, you need to first import measured water-surface elevation data. Select **Import ► Measured Values** from the Menu Bar. Select the file “GR\_wse.csv” in the File Open dialog. The data is added to the Pre-processing Window **Object Browser** under **Measured Data**.
- You will see that importing the measured data adds a legend by default. To turn the legend off in the **Object browser** select and right click on **Measured Values | C:\(path)\wse.csv | Scalar** and in the resulting pop-up dialog select **Property**. In the **Scalar Setting** dialog select the **Color Bar Setting** button, and then deselect the **Visible** attribute.

## Creating a Numerical Grid

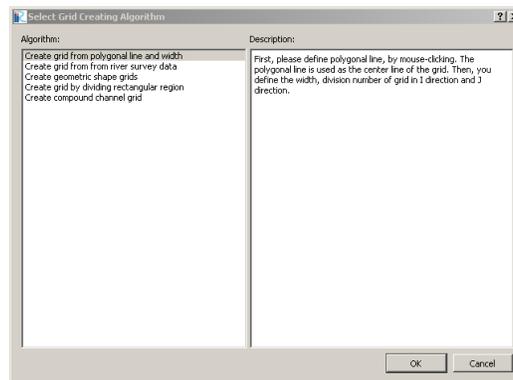
All solvers require a computational or numerical grid to perform the calculations. iRIC contains a number of different methods or algorithms that can be used to generate the grid you need for your model. The FaSTMECH solver uses a structured curvilinear-orthogonal grid. A curvilinear-orthogonal grid defaults to a rectilinear grid when only two points define the centerline. 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.

When creating a grid there are a number of things to keep in mind; 1) Keep the grid within the bounds of the data; 2) Define the upstream and downstream boundary of the grid beyond the area of computational interest to account for the effects of upstream and downstream boundary conditions; 3) Locate the upstream and downstream boundaries

(the edge of the grid) in areas where flow is as uniform as possible. Avoid areas of recirculating flow or abrupt transitions in the bed or width; 4) Orient the upstream and downstream boundaries perpendicular to the main flow direction; 5) Select a grid width that minimizes the amount of the grid outside the active channel, therefore maximizing the number of nodes in the grid contributing to the solution; 6) Insure that the grid does not overlap itself in the active channel. The application will give an error if the grid overlaps.

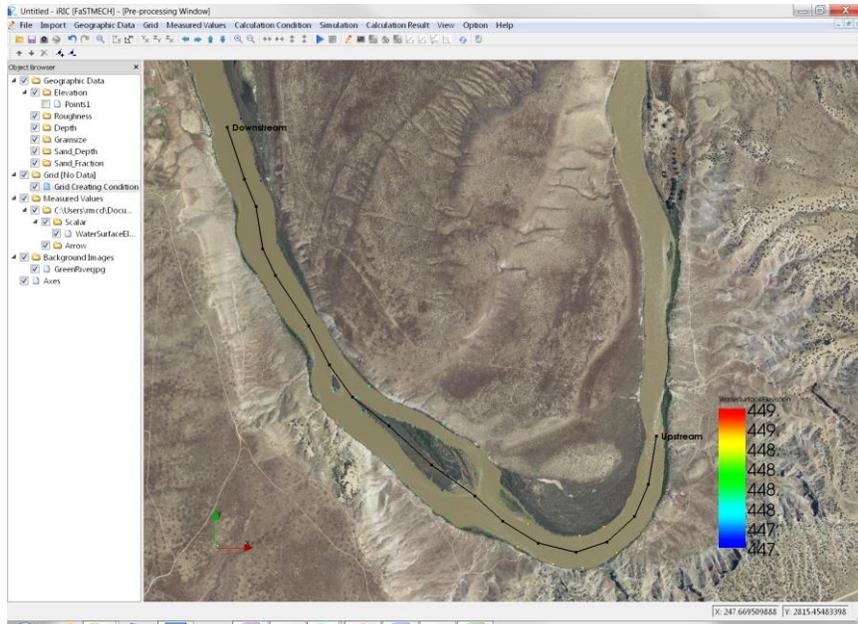
### Create grid:

- To better view the data, click on the bottom right corner of the Pre-processing Window and expand the window or select the maximize button along the top of the Pre-processing Window to expand the window.
- In the **Menu Bar** select Grid ► Select grid creating algorithm. This opens a dialog (Figure 5), select “Create grid from polygonal line and width.” Click OK. Another dialog will open providing instructions on using this feature. Click OK.



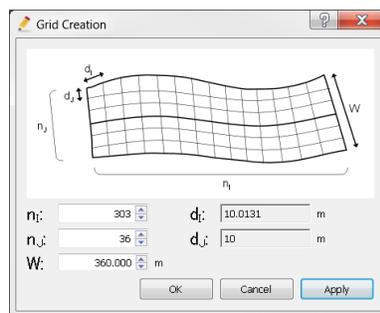
**Figure 5.** Example of the Select Grid Creating Algorithm dialog.

- 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 6). The upstream end of the channel in Figure 6 is at the bottom right and the channel centerline should be drawn in the direction from lower-right to upper-left. When finished press “Enter” on the keyboard. Use Figure 6 and the blue water-surface points on the background image as a guide in selecting the boundary locations.



**Figure 6.** Location to draw centerline. Flow is from bottom to top so start at the lower end of the channel and select points ending near the top of the channel.

- A new dialog opens that 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 7). Set the grid width equal to 360 meters and define the number of points in the streamwise and cross-stream dimension to give corresponding increments of about 10 meters (displayed by  $d_i$  and  $d_j$ ). The distance between nodes in the stream-wise direction along the center line and the cross-stream direction is constant everywhere on the grid. Use the Apply button on the dialog to dynamically change the view of the grid to find the desired spacing of nodes in the stream-wise and stream-normal directions. Select OK when you are done. If you need to return to this menu, select *Grid* ► *Create Grid* from the **Menu Bar**.



**Figure 7.** Grid Creation dialog that allows you to specify grid characteristics.

- 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. Your grid should look similar to Figure 8

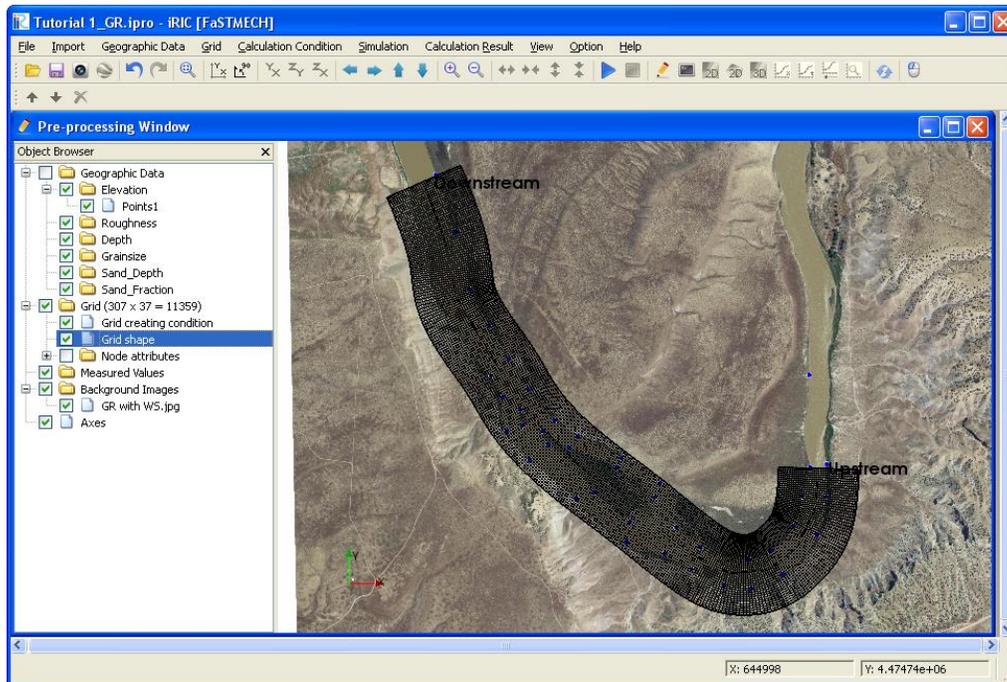


Figure 8. Example grid for the Green River.

- 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
- Adjust the grid so that the upstream and downstream boundaries are roughly perpendicular to the channel and the upstream and downstream extent of the grid encompasses the measured water-surface elevations. Be careful at the upstream end because the measured water-surface elevation is right at the limit of the measured elevations. Extending the grid too far upstream will result in poor elevation values at the upstream end of the grid.
- Add or remove centerline points by selecting *Grid ► Grid Creating Condition* on the Menu Bar and select the option you need. Be aware that new centerline points must be added along the current line; if you click too far away from the centerline no points will be added.
- Your result should look similar to Figure 9.

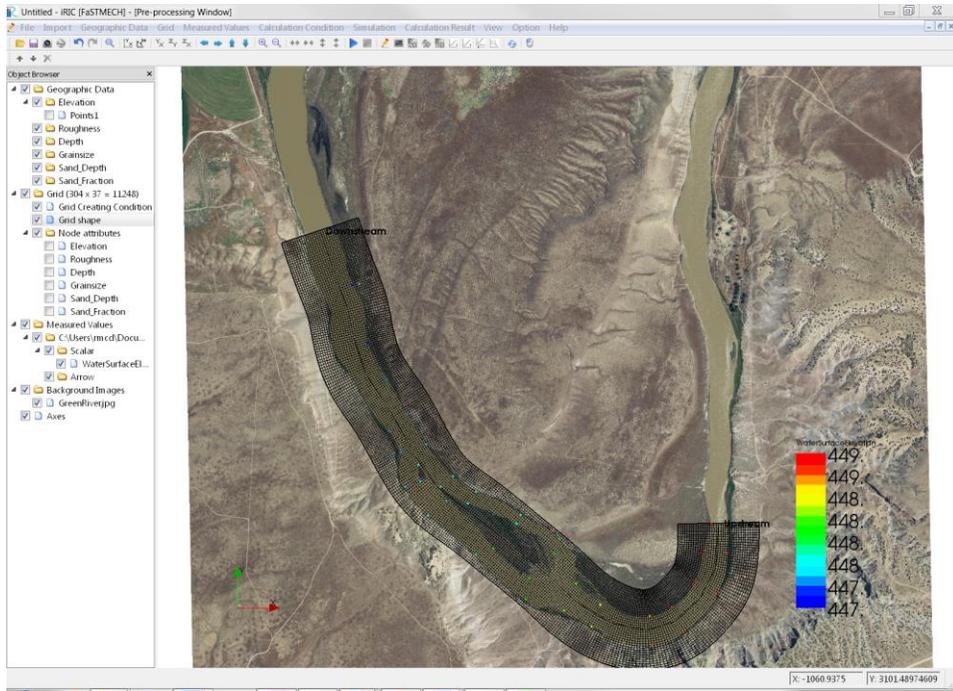


Figure 9. Edited grid location.

### Mapping geographic data to the grid attributes.

Once you are satisfied with your computational grid, you need to make sure that elevation is specified for each cell on the grid. There are two ways to do this. The first method uses a triangular irregular network (TIN) to interpolate the elevation data at each grid cell. The second method is based on a template that uses a nearest neighbor approach to interpolate. In a previous step we set the mapping algorithm to the template method.

- We need to set the width and length of the template. From the Menu Bar select *Grid ► Attributes Mapping ► Setting*. This opens the Grid Attribute Mapping Setting dialog. We previously set the Execute attribute to Manual and the Mapping attribute to Template. Select the Detail button near the bottom of the dialog (Figure 10).

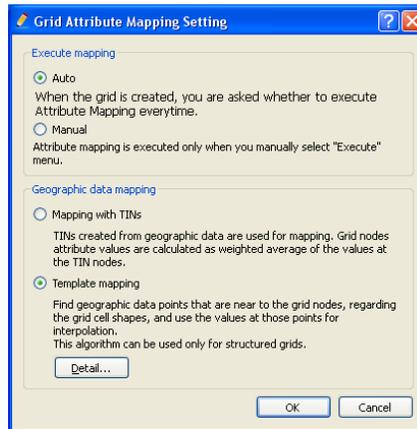


Figure 10. Grid Attribute Mapping dialog

- This opens the Template Mapping dialog. Set the Search region to Manual and set the other attributes to the same values as Figure 11 and select OK. Since the data has a high density and the spacing of the points are very uniform we'll set the template algorithm to act like a nearest neighbor search by selecting the length and width of the template to be the same value.

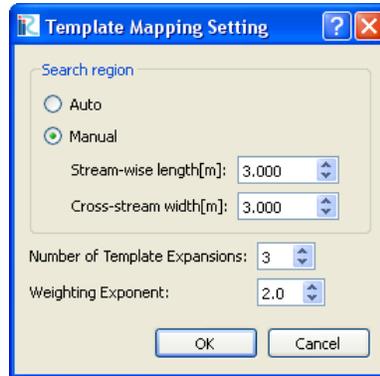


Figure 11. Template Mapping dialog

- To execute the mapping method, select *Grid* ► *Attributes Mapping* ► *Execute* from the Menu Bar. Select OK when notified that mapping is complete. Node attributes will be added to the Object Browser under the other Grid features. Expand the Node attributes and make sure that Elevation is selected.
- You will likely need to adjust the Color bar so you can view the results of mapping elevation to the grid. From the **Object Browser** select and right-click on Grid() | Node Attributes | Elevation and in the resulting pop-up menu select Property. In the Grid Node Attributes Display Setting dialog uncheck the box next to automatic. Set the max value to 449 and the min value to 444. Set the Contour setting attribute to Contour Figure. Select OK to obtain results that look like Figure 12.

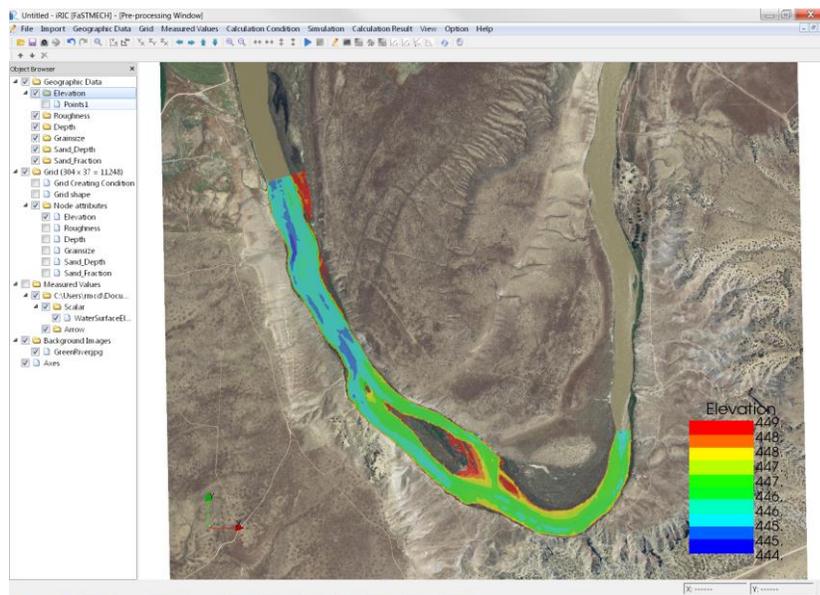


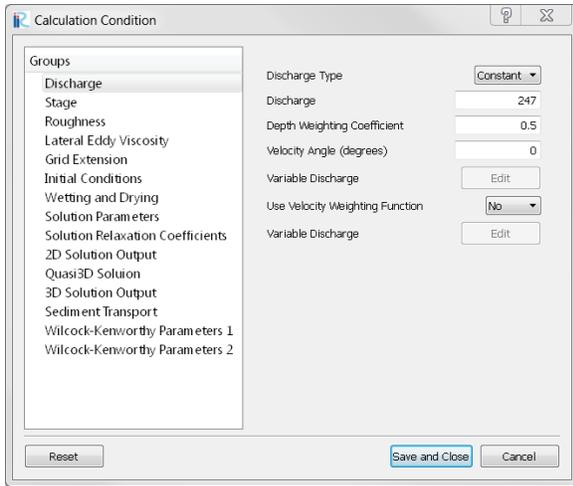
Figure 12. The resulting computational grid with the measured elevations mapped to the grid.

- Save the Project.

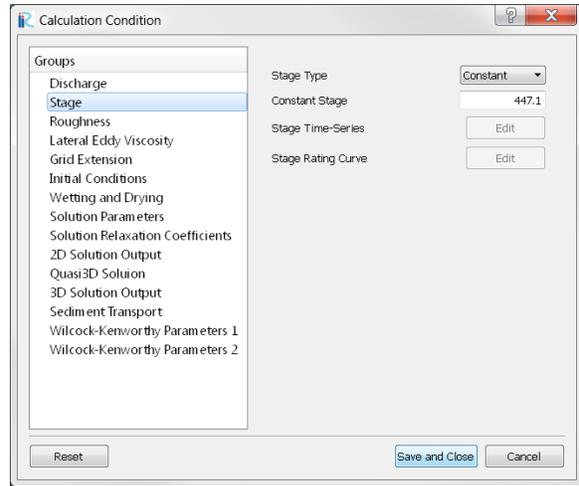
# Defining the Calculation Conditions and Running a Simulation

Once the grid is complete, the next step is to set up the required information, or calculation conditions, for the FaSTMECH solver.

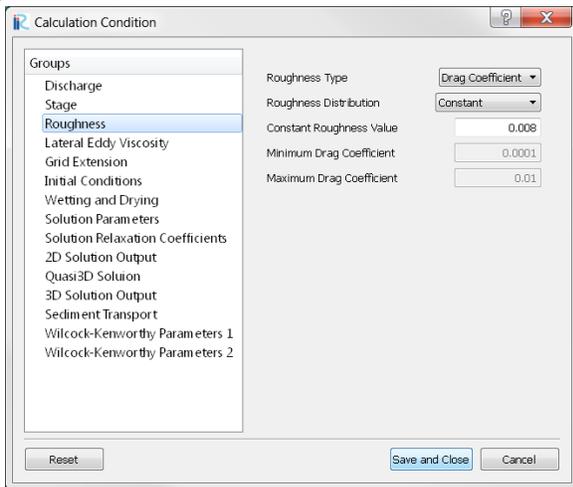
- In the Menu Bar select *Calculation Conditions* ► *Setting*. This opens the Calculation Conditions dialog. The left window of the dialog displays the primary parameter groups that can be set using the FaSTMECH solver. Values for various parameters are entered in the fields on the right side of the window.
- Enter the required parameters based on the values in Figure 13 and select Save and Close when complete.



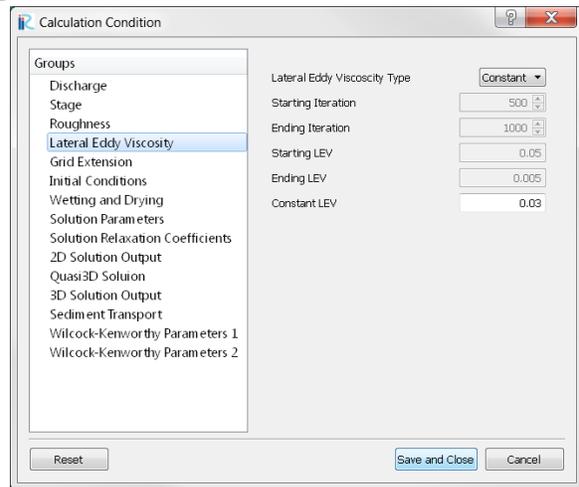
A



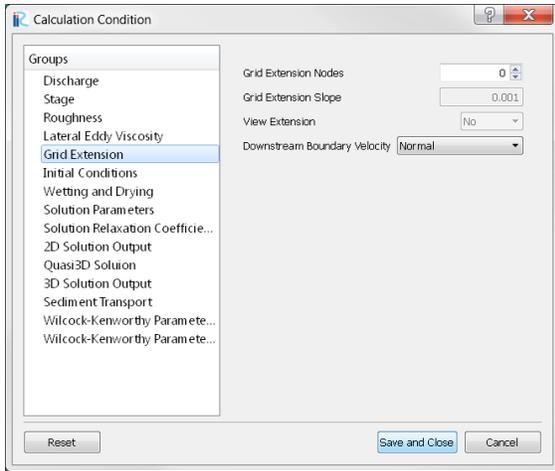
B



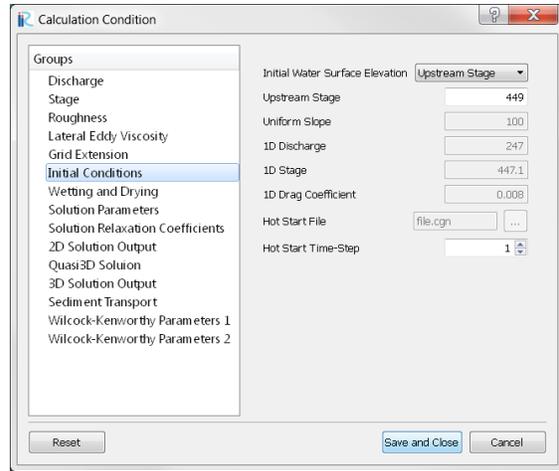
C



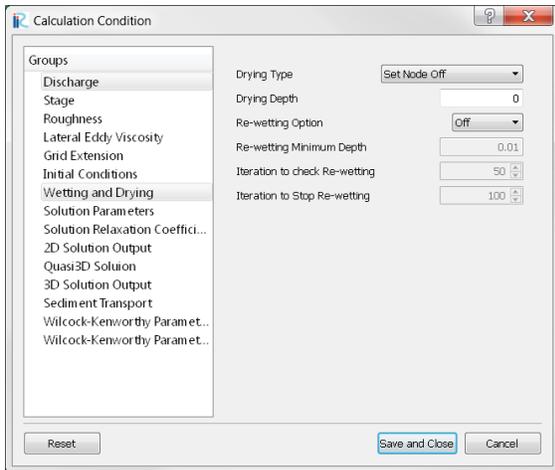
D



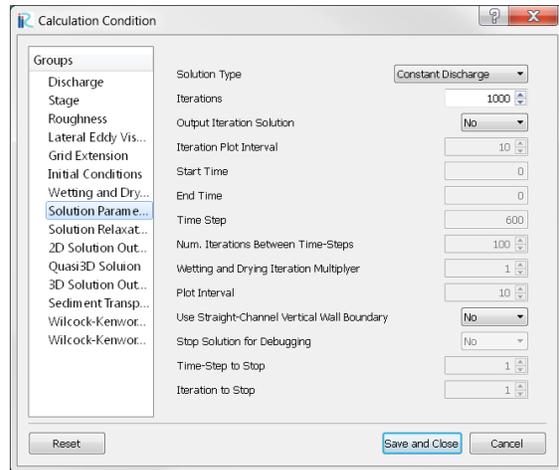
E



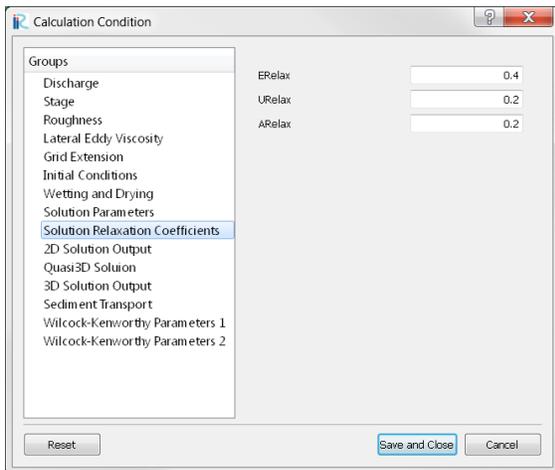
F



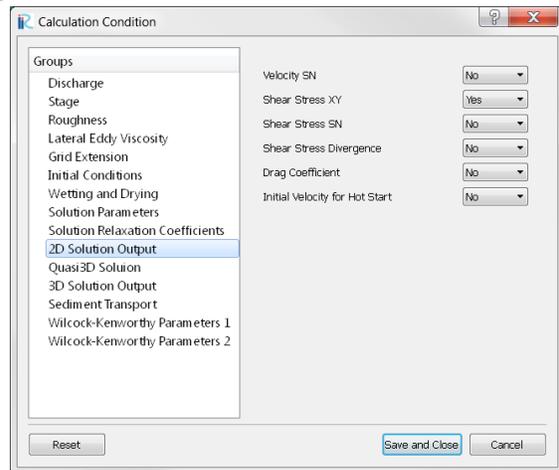
G



H



I



J

**Figure 13.** FaSTMECH Calculation Conditions dialog. (A) Discharge, (B) Stage, (C) Roughness, (D) Lateral Eddy Viscosity, (E) Grid Extension, (F) Initial Conditions, (G) Wetting and Drying, (H) Solution Parameters, (I) Solution Relaxation, (J) 2D Solution Output.

- From the **Menu Bar** select Simulation ► Run. 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 (Figure 14). Note that the Solve Console displays the “mean error on discharge.” Typically this value should decrease as the simulation converges until it reaches a low value (typically values less than ~3 percent) and stabilizes. A dialog opens to notify you when the calculation is complete. Click OK to close the dialog.
- If the solution fails an error message will be displayed in the Solver Console and you must select  from the Main Toolbar and then in the Menu Bar select *Calculation Conditions* ► *Setting* and make sure your setting are correct (See Figure 13).

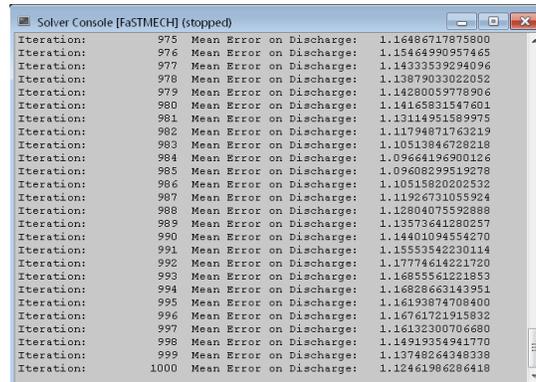


Figure 14. Example of the Solver Console.

## Post-processing

iRIC provides a complete suite of tools for visualizing 2D model results. Map visualizations of model calculated flow characteristics are viewed in a 2D Post-processing Window. 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.

### Map Visualizations

Calculation results can be viewed by opening a new 2D Post-processing Window by selecting  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** select the check box next to iRICZone | Scalar | WaterSurfaceElevations (Figure 15).

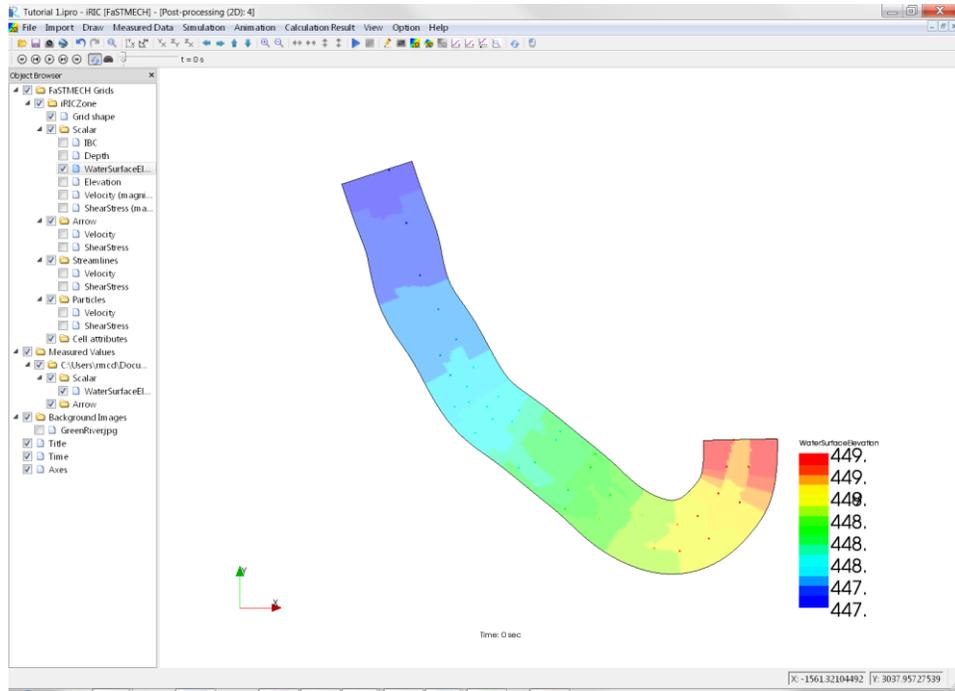


Figure 15. Initial display of the simulated water-surface elevation.

- Change how the contours are displayed by selecting from the **Menu Bar** Draw ► Contours (Figure 16). Experiment with attributes in the main Scalar setting dialog.

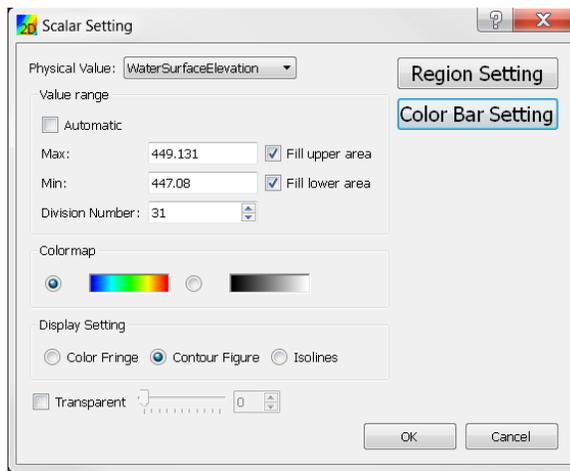
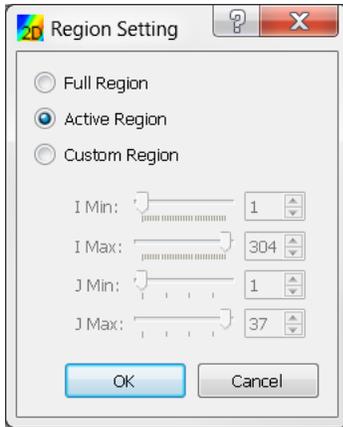
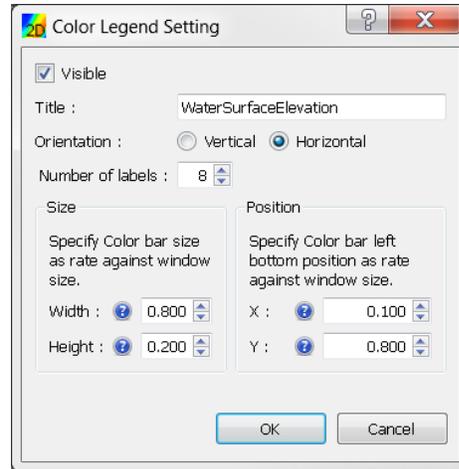


Figure 16. Scalar Setting dialog

- Change the spatial extent displayed by selecting Region Setting in the Scalar setting dialog. Select Active Region to display only wet nodes (Figure 17A). Or select a specific area of interest to display using the Custom Region options.
- Change the display of the legend or scalebar by selecting the Color Bar Setting button in the Scalar Setting dialog (Figure 17B).



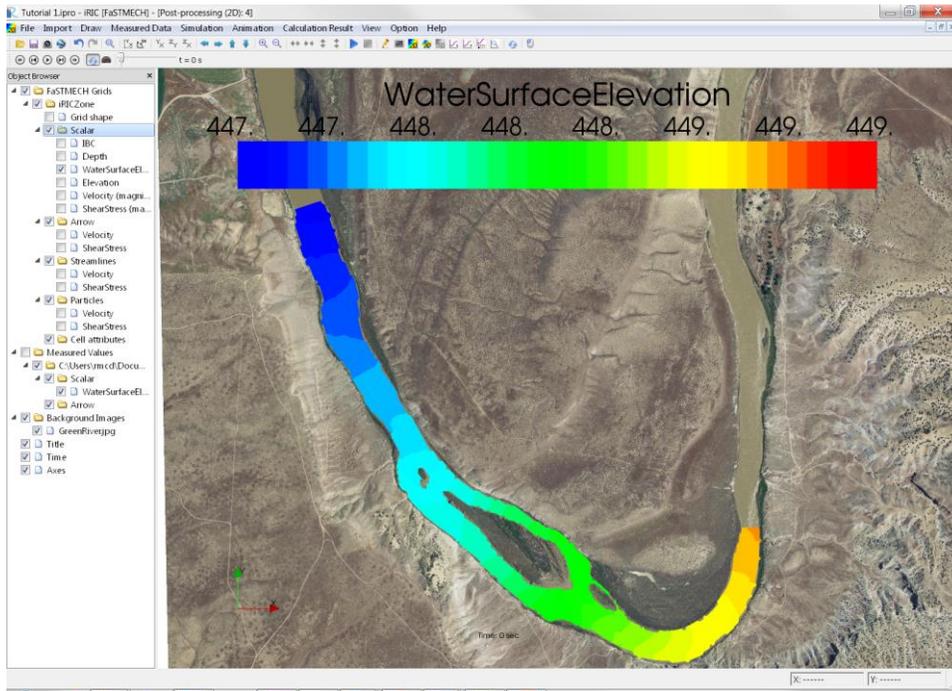
A



B

**Figure 17.** Additional display options available in the Scalar Setting dialog include the (A) Region Setting dialog and the (B) Color Legend Setting dialog (B).

- Experiment with these features and to achieve a satisfactory display of the model solution (Figure 18).



**Figure 18.** Example display of the simulated water-surface elevation.

## Displaying Vector Results

Vector results such as the orientation and magnitude of velocity and shear stress can also be displayed using the 2D Post-Processing Window.

- For now, in the **Object Browser** uncheck the box next to iRICZone | Scalar to turn off the display of this type of data. Check the box next to Arrow and select Velocity.
- The default values for the vectors often need adjustment. From the **Menu Bar** select Draw ► Arrows to bring up the Vector Attributes dialog (Figure 19). The result is shown in Figure 20.

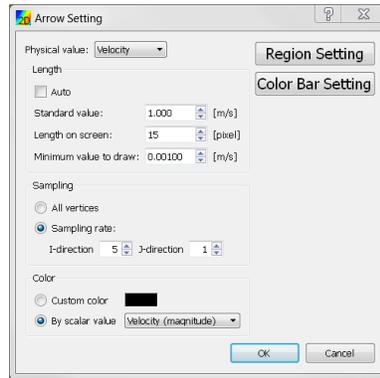


Figure 19. The Vector Attributes dialog.

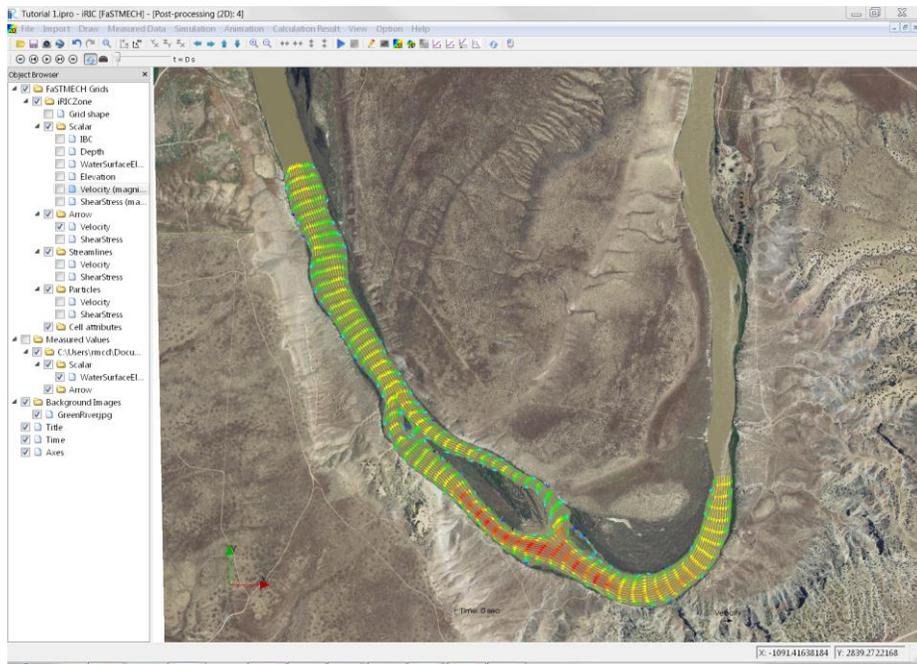


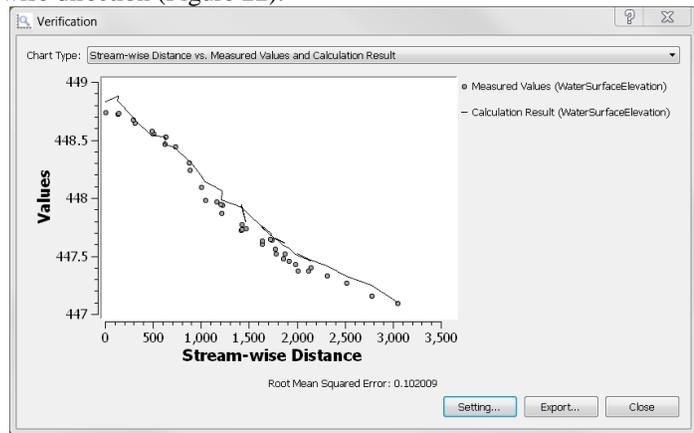
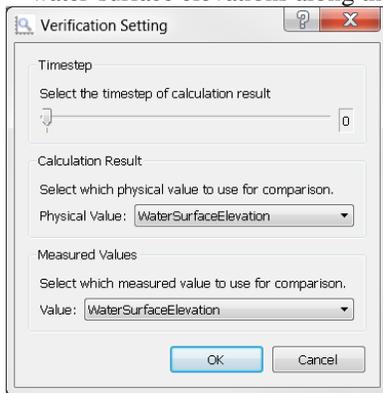
Figure 20. A plot of velocity in the modeled reach represented as vectors. Vectors are color coded and scaled according to their magnitude.

# Model Calibration and Refinement

## Developing a Calibration Curve

Model calibration is an important step in developing a flow model. Typically simulated water-surface elevations are compared to measured water-surface elevation data collected at a known discharge. Comparing measured and simulated results helps to determine whether or not calculation conditions like roughness have been set appropriately. Other types of measured data can also be compared to the simulated values, such as velocity.

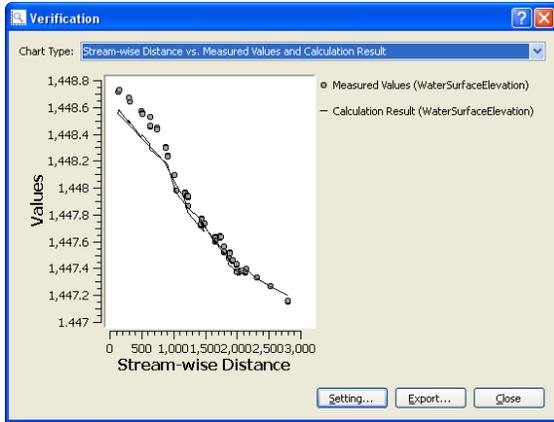
- In the **Menu Bar** select Calculation Results ► Compare with measured values or select the  tool In the Main Toolbar. This opens a dialog that allows you to select which variables to compare. Set the fields as shown in Figure 21, and select OK. A Verification Window opens that plots the measured and simulated water-surface elevations along the stream-wise direction (Figure 22).



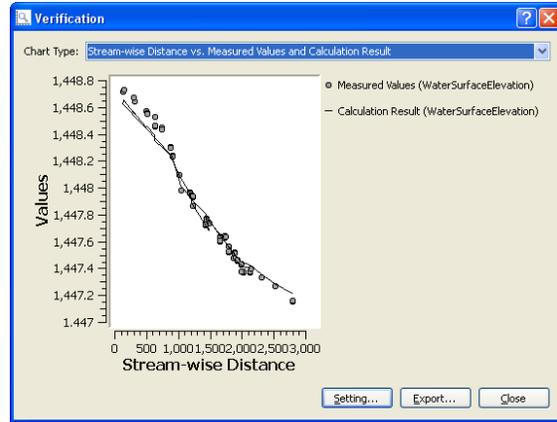
**Figure 21.** Use the Verification dialog to select data for comparison. If more than one measured data set has been imported there is an additional field to select which file to use.

**Figure 22.** Example verification plot showing the measured and simulated water-surface elevation. Note that the simulated water-surface elevation is too high compared to the measured data at the downstream end of the modeled reach. the Root Mean Squared Error is 0.1.

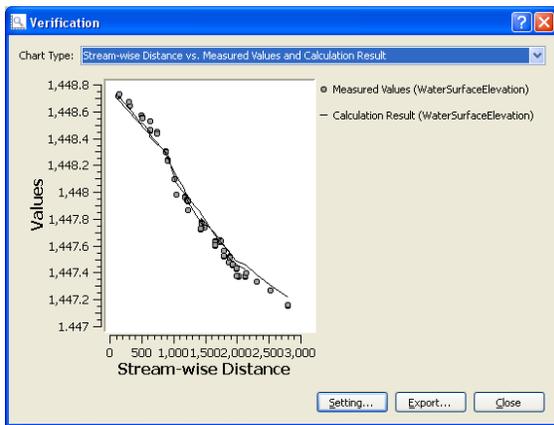
- To calibrate the model you could try adjusting the roughness value, Cd, to see if you can obtain a better match between the simulated and the measured values. Figure 23 shows the verification plot and the gives the RMS error for a range of simulated roughness values. Looking at these results, it appears that a Cd of about 0.007 yields the lowest RMS error. However, none of the simulations are a good match at all locations.



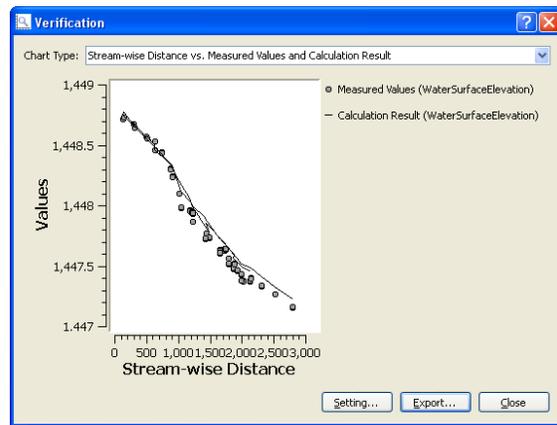
A.  $C_d = 0.005$  RMS = 0.098



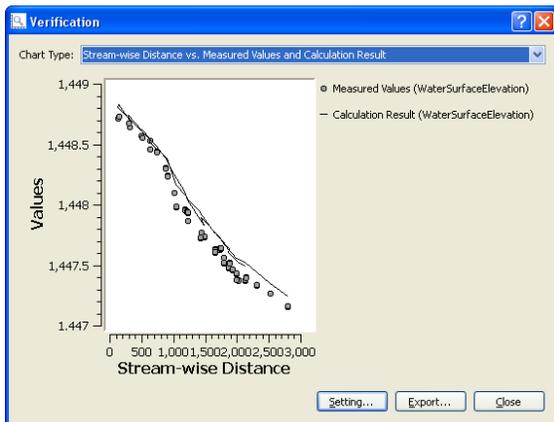
B.  $C_d = 0.006$  RMS = 0.064



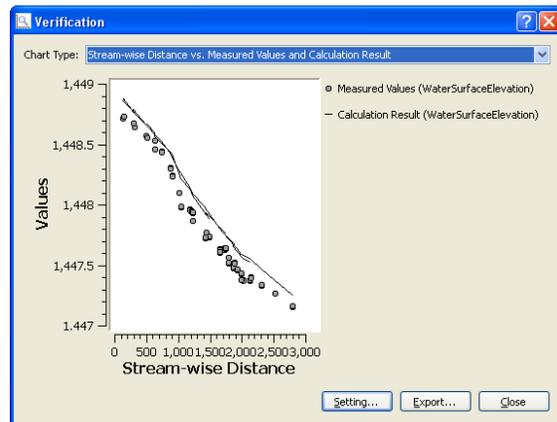
C.  $C_d = 0.007$  RMS = 0.059



D.  $C_d = 0.008$  RMS = 0.085



E.  $C_d = 0.009$  RMS = 0.121



F.  $C_d = 0.01$  RMS = 0.160

**Figure 23.** Plots of the predicted and observed water-surface elevations for a range of drag coefficient values between 0.005 – 0.01 (A-F). The RMS error indicates that the best calibrated value for the drag coefficient is between 0.005 and 0.007.

- Another way to look at these results is to plot a calibration curve which shows the roughness value on the x-axis and the RMS error on the y-axis (Figure 24). This curve indicates how sensitive the simulated water-surface elevation is to the roughness value. Additional simulations could further determine the drag coefficient that gives the lowest error between measured and predicted water-surface elevations.

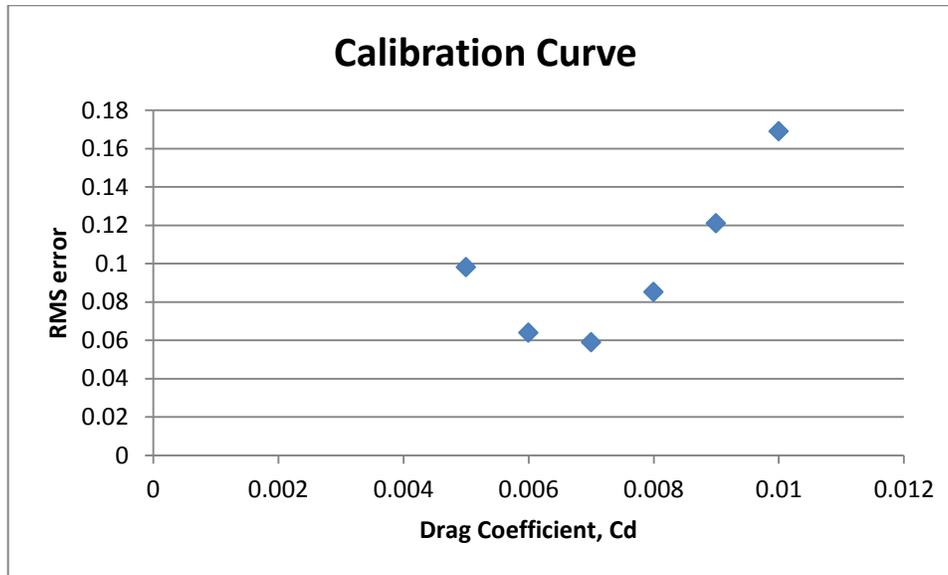


Figure 24. Calibration curve showing the relationship between roughness and the RMS error for the simulated water-surface elevations.

## Spatially Varying Drag Coefficient

Examination of the simulated and measured water-surface elevations in Figure 23 indicates that using a constant drag coefficient for the entire reach yields unsatisfactory results. Generally the predictions are good for either the upper or lower half of the reach but not the reach as a whole. This indicates that there is a change in roughness through the reach. This can be the result of many factors including a change in grain-size, different bedform geometries, and the presence of aquatic or submerged vegetation.

Maps of grain size distribution or vegetation can be used to adjust the roughness in specific regions. In this tutorial we simply vary the drag spatially by defining an area with a higher drag coefficient in the vicinity of the large island mid-way through the modeled reach.

To spatially vary roughness we'll return to the Pre-processing module. Note that Roughness is one of the Geographic data sets.

- The first step is to delineate polygons that spatially define regions of roughness.
  - In the Pre-processing Window turn on the background image and in the **Object Browser** right-click on Geographic Data | Roughness and in the resulting pop-up menu select Add ► Polygon. Click OK when done reading the instructions and define a polygon that encloses the entire model Grid. When done, hit the Entry Key and enter a roughness value of 0.005 in the dialog.
  - Create another polygon and this time define a polygon similar to the one in Figure 25. At the prompt enter a roughness value of 0.012.
  - Make sure that both polygons are displayed in the **Object Browser** under Geographic Data | Roughness.

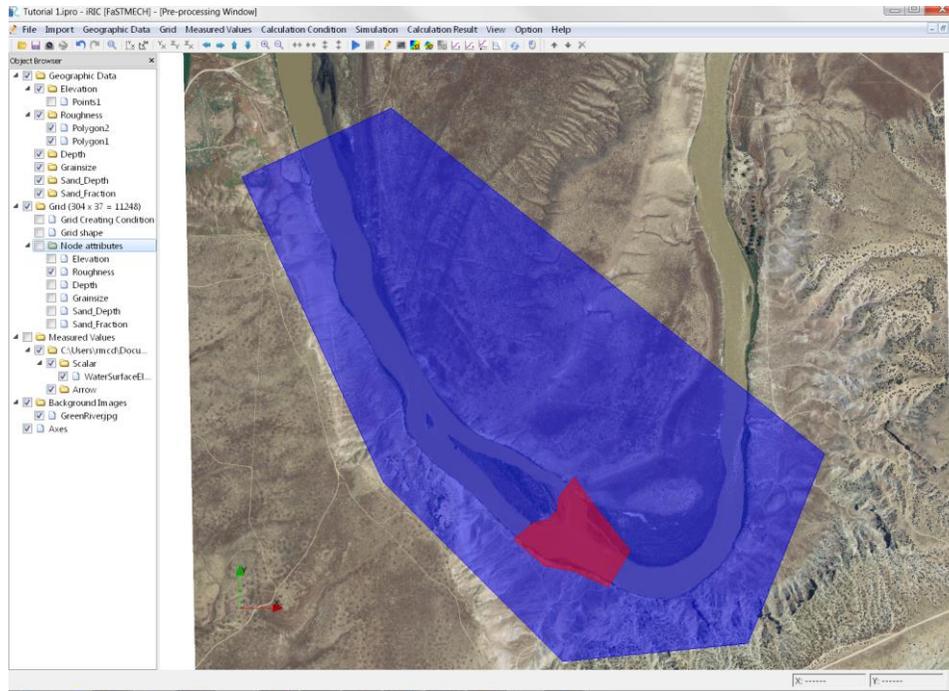


Figure 25. Example of the polygon mapping

- The next step is to map or specify the roughness for each cell in the grid. The iRIC application knows when you have changed any values in the Geographic Data tree and will make sure the new values are mapped before you run the simulation. However in order to see how the polygons are mapped to the grid we'll do it manually.
  - First highlight Geographic Data | Roughness in the **Object Browser**. Then on the **Menu Bar** select **Grid ► Attribute Mapping ► Execute**.
  - Make sure that the roughness values have been mapped to the grid by checking the box next to roughness under *Grid | Node Attributes* in the **Object Browser**. You may need to expand those sections to see the full list. Make sure you get a result that looks like Figure 26.
  - You can edit the location of polygon nodes by selecting the polygon you want to edit in the Geographic Data tree, and then much like the grid centerline you can move the mouse until it's over a node to move the node or over a line between two nodes to move the whole polygon. You will also notice that there are tools in the Operation Toolbar to add or delete nodes in a polygon.

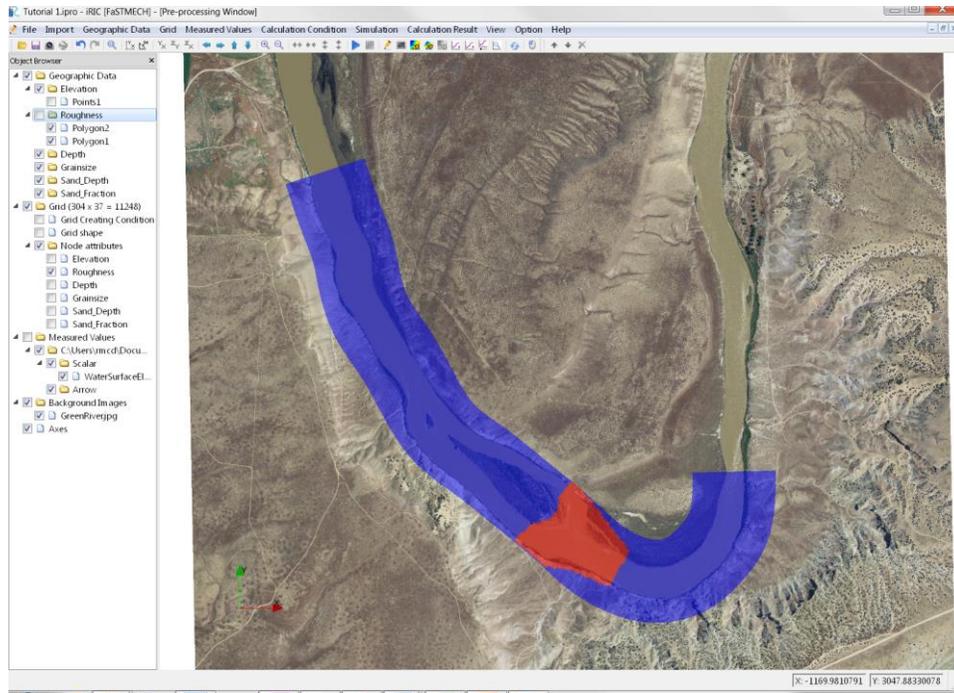


Figure 26. Final roughness attribute mapping

- Now update the calculation condition so that it uses a variable instead of a constant roughness value.
  - From the **Menu Bar** select Calculation Condition ► Setting. In the Calculation Condition dialog select roughness and change the settings to look like Figure 27. Select the Save and Close button.

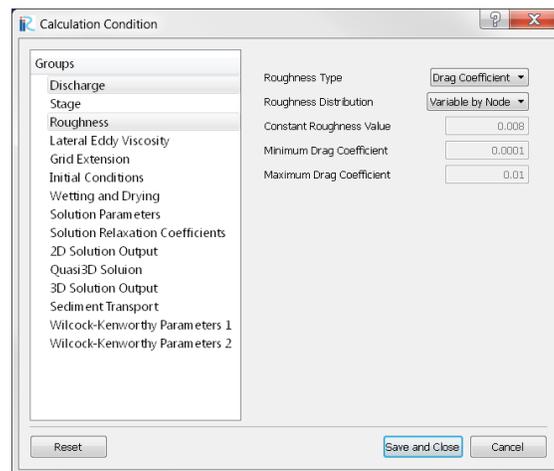


Figure 27. Calculation condition, roughness dialog

- Now rerun the simulation by clicking the run button or by selecting Simulation ► Run from the Menu Bar.
- When the simulation is complete, look at the Verification (Figure 28).

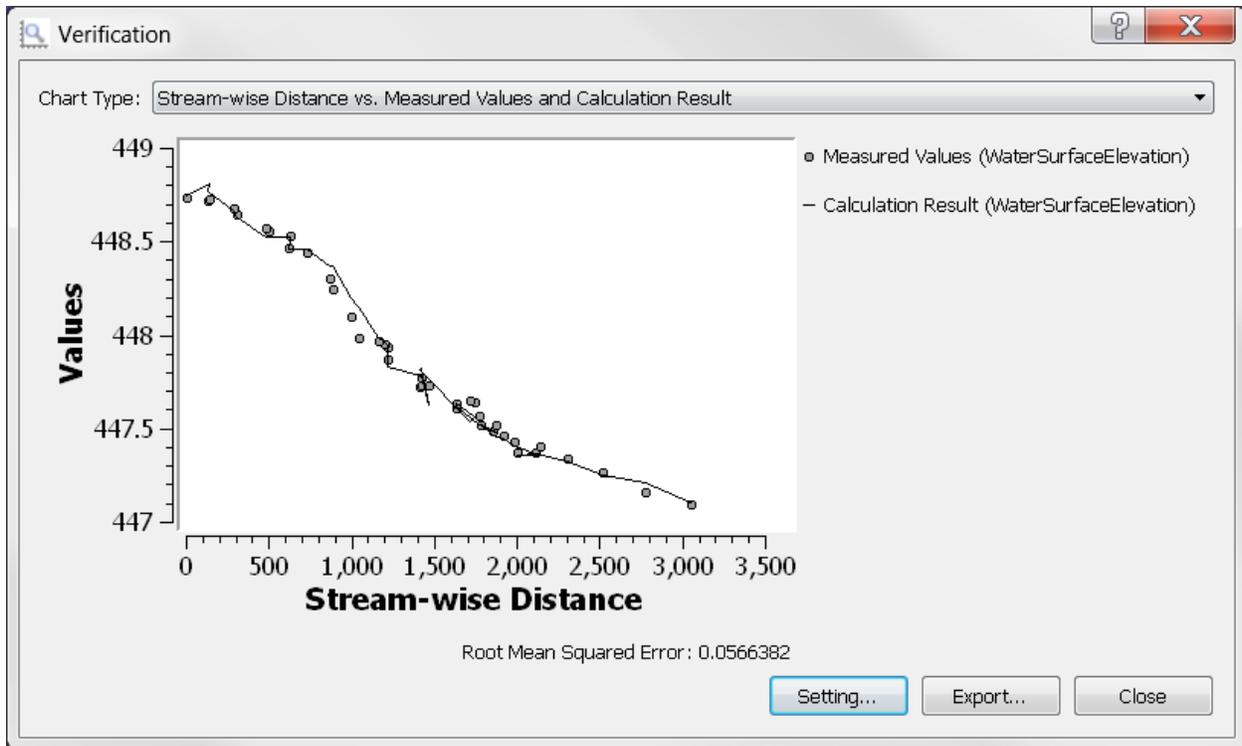


Figure 28. Comparison of the measured and simulated water-surface elevation using a spatially variable drag coefficient. RMS error is 0.056.