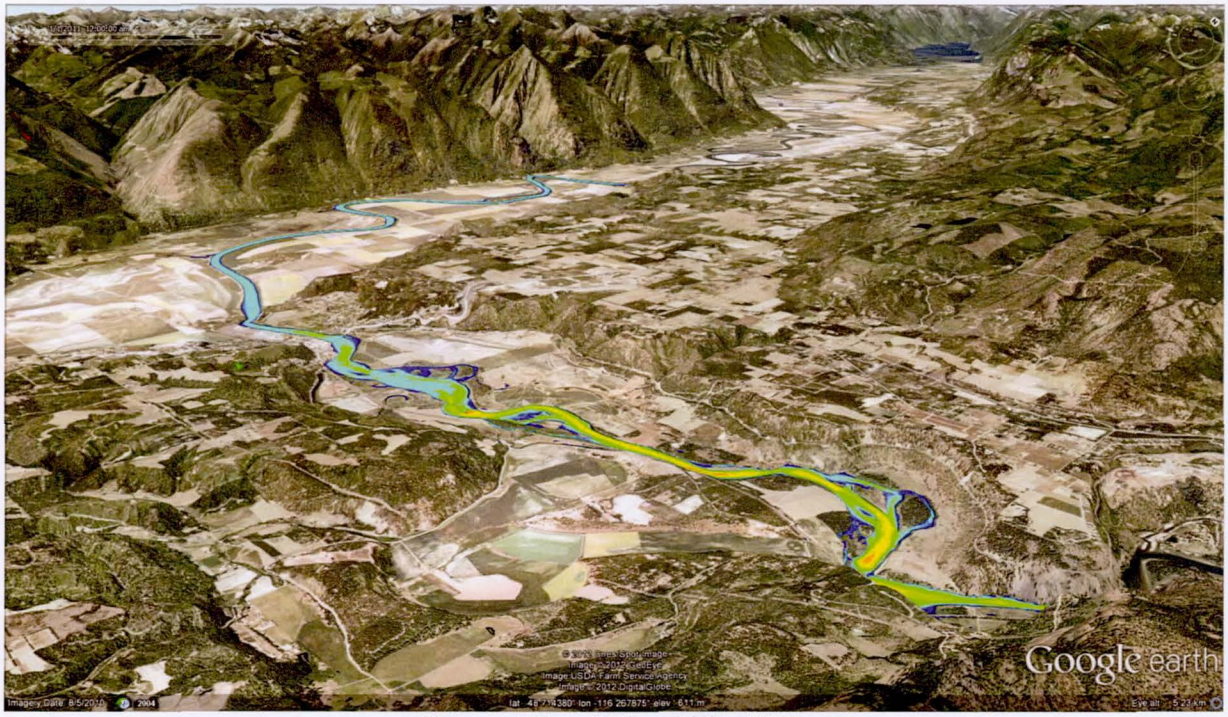
iRIC Software

Changing River Science

FaSTMECH Tutorial 1

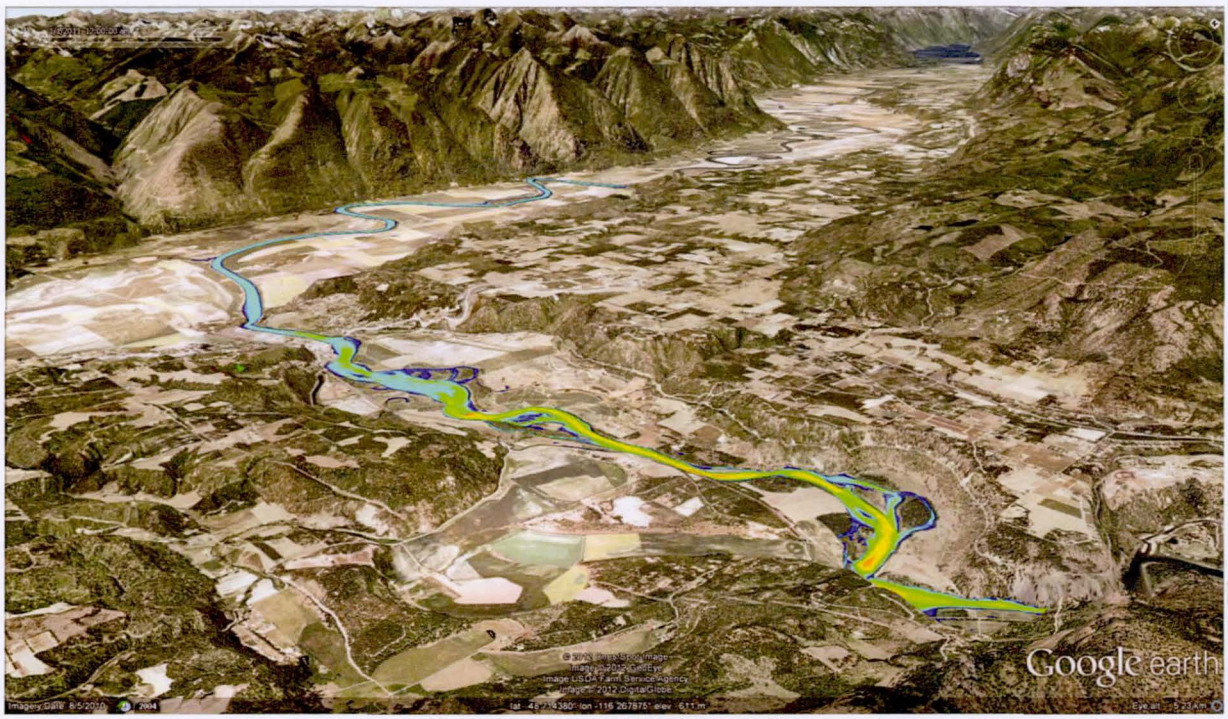


Kootenai River Valley, Idaho, USA

iRIC Software

Changing River Science

FaSTMECH Tutorial 1



Kootenai River Valley, Idaho, USA

Contents

Introduction to the iRIC Graphical User Interface 2
Exercise 1: Importing Data 4
Import Topography 5
Import Images 7
Simple Data Editing 7
Import Data for Model Calibration and Verification 9
Import Measured Velocity Data 9
Import Measured Water-Surface Elevations 9
Exercise 2: Grids 10
Creating a Numerical Grid 11
Create the Curvilinear Grid: 11
Mapping Geographic Data To The Grid Attributes. 15
Mapping With TINs 16
Mapping With the Template Method 16
Edit Data to Improve Mapping 21
Exercise 3: Defining Calculation Conditions and Post-processing 26
Defining Calculation Conditions 26
2D Post-processing 29
Displaying Scalar Results 29
Displaying 2D Vector Results 30
Compare Measure Velocity Vectors to Solution Vectors 30

Introduction to the iRIC Graphical User Interface

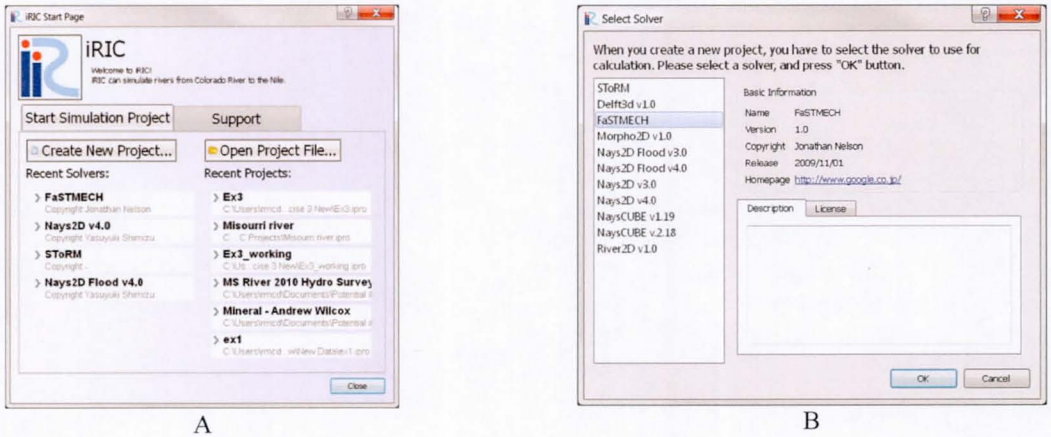
Launch iRIC 2.3 by selecting iRIC 2.3 from the Program Menu list or click on the iRIC icon ( ) on the desktop. The iRIC Start Page (Figure 1A) opens and displays several options to start a project under the Start Simulation Project tab:

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)

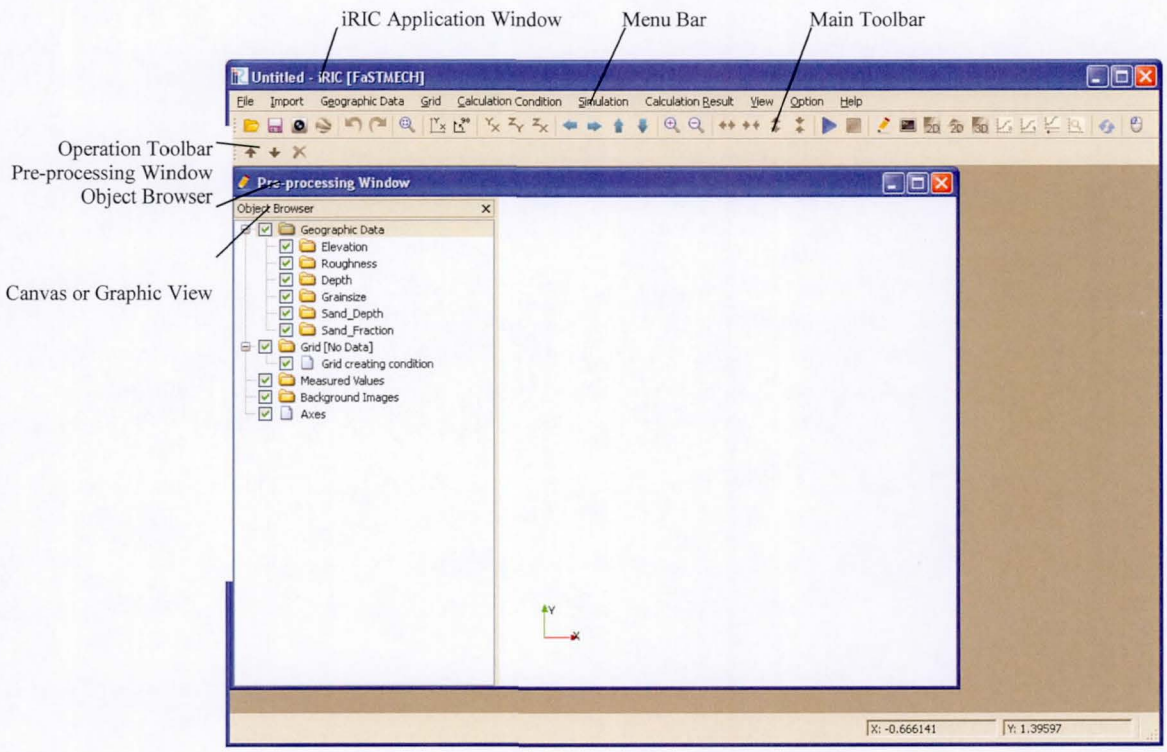


Figure 2. Example of the main iRIC interface

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 ( ). 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 1). Experiment with viewing the data as wireframe and as a surface.
- Save by selecting File **->**Save as File (*.ipro) from the Menu Bar.

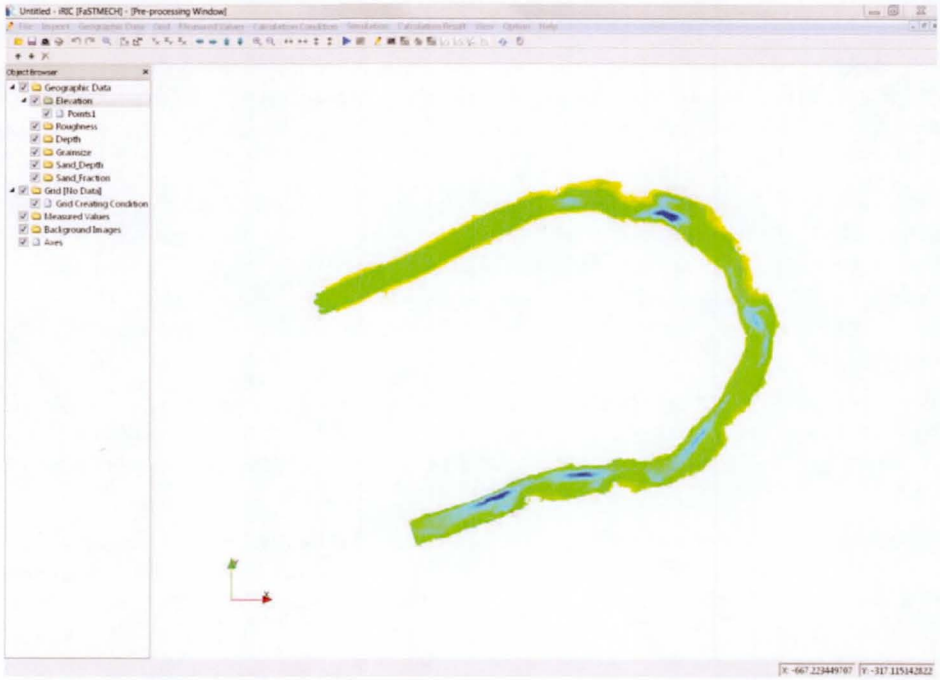


Figure 1. Pre-processing Window display of the elevation data.

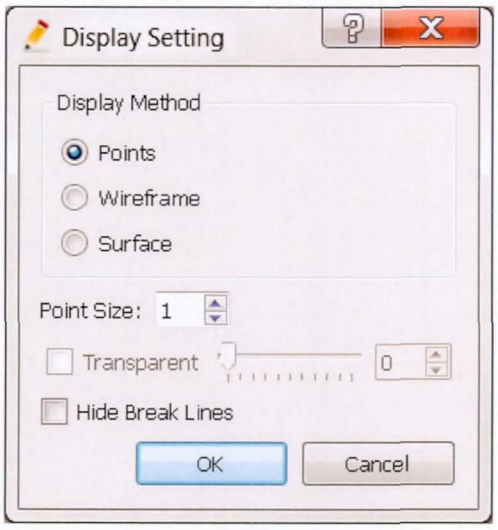


Figure 2. Display properties options for Elevation Data

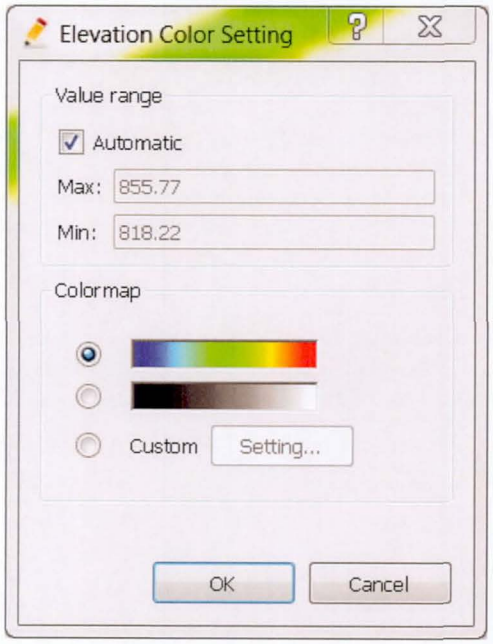


Figure 3. Display properties options for Elevation Color Setting

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

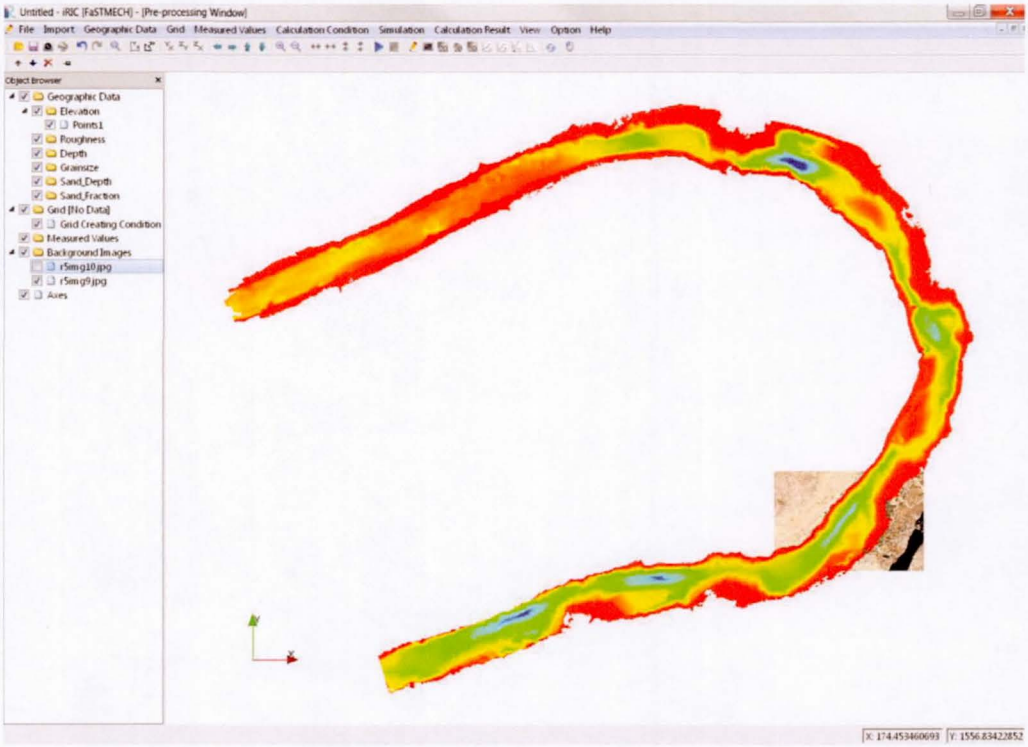


Figure 4. Surveyed topography and background image.

Simple Data Editing

- We would like to focus on the region of the data set encompassed by the two imported images. iRIC 2.3 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.

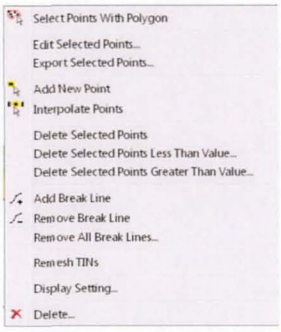


Figure 6. Pop-up menu for actions applied to selected points

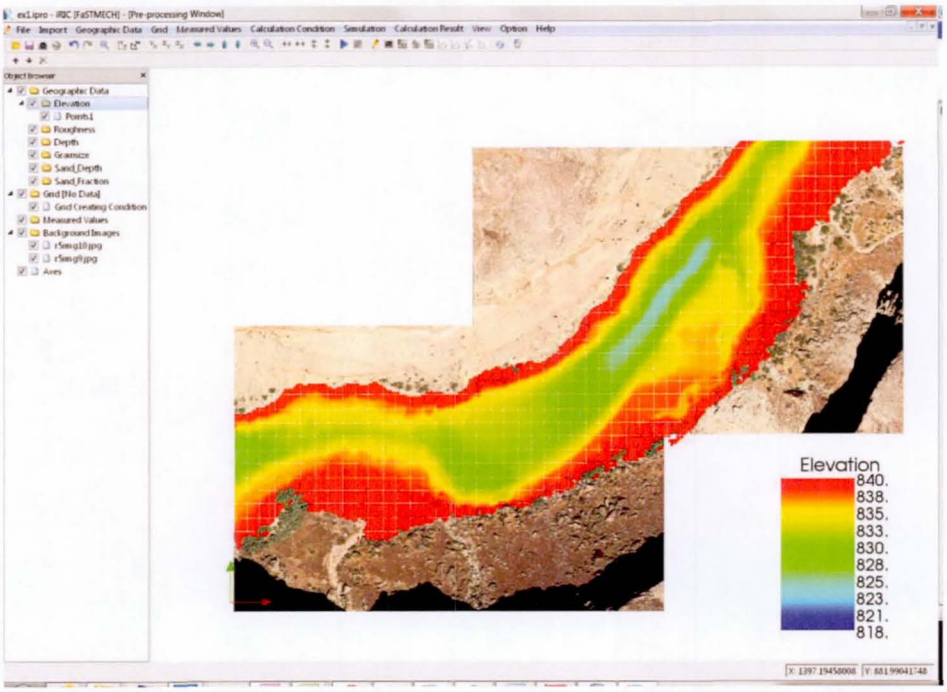


Figure 7. Image and elevation points following data point editing.

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.

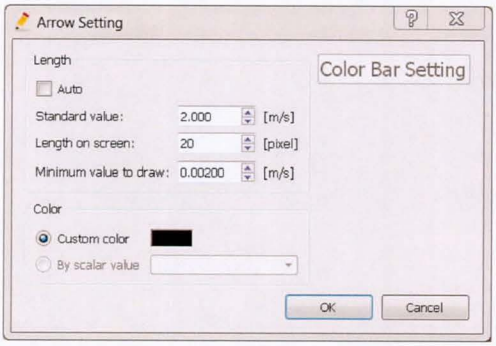


Figure 8. Arrow setting to set measured vector properties

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.
- 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

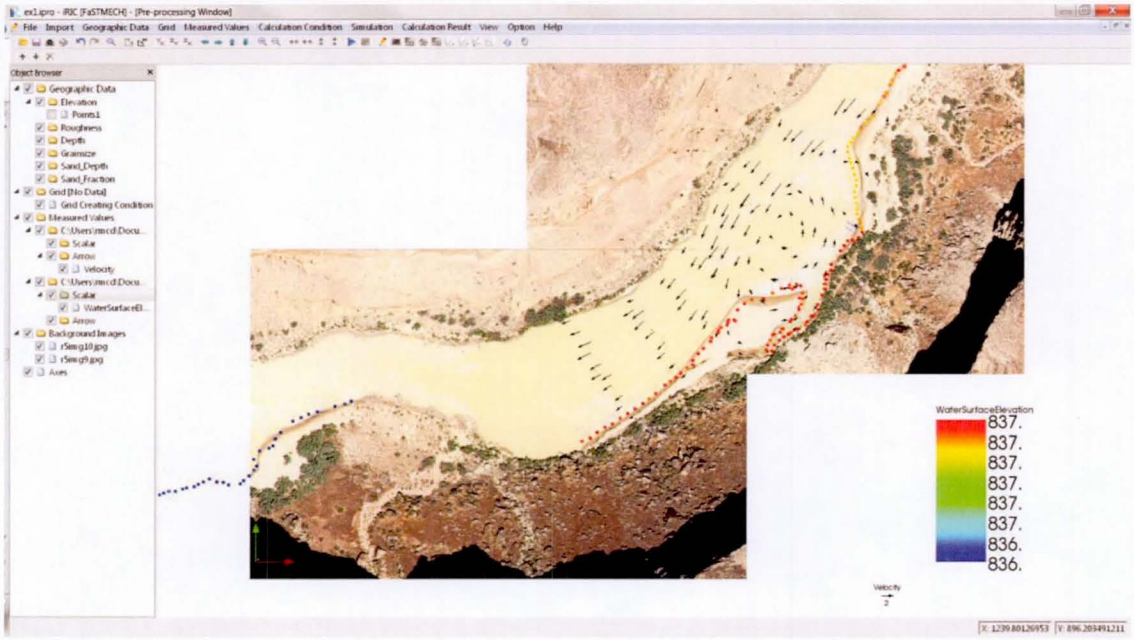


Figure 9. Measured velocity vectors and water-surface elevations scalar values shown on top of geo-referenced images.

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.

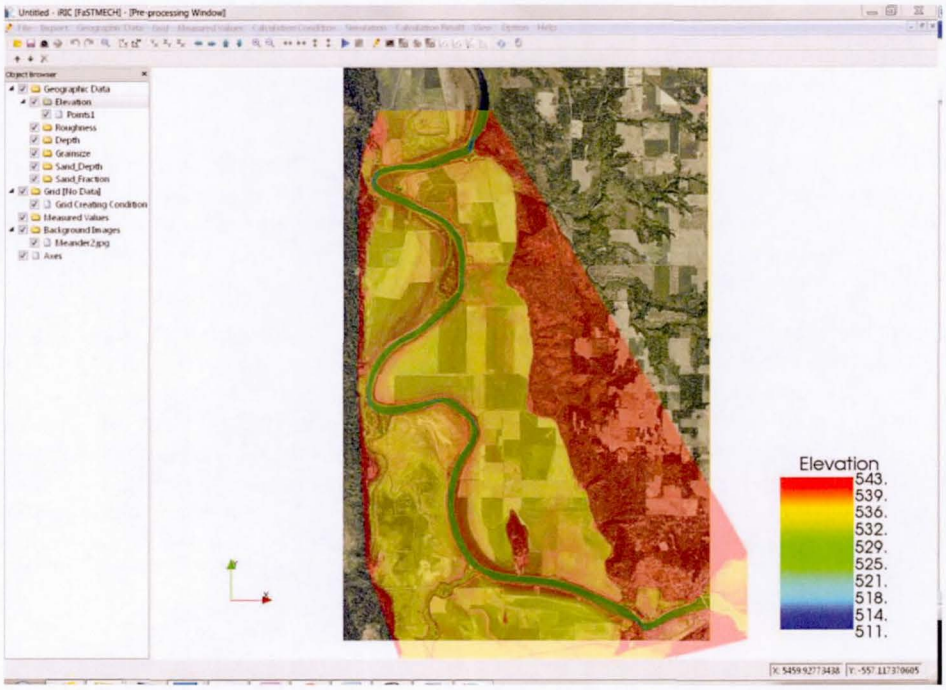


Figure 1. Kootenai River topography and image.

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.

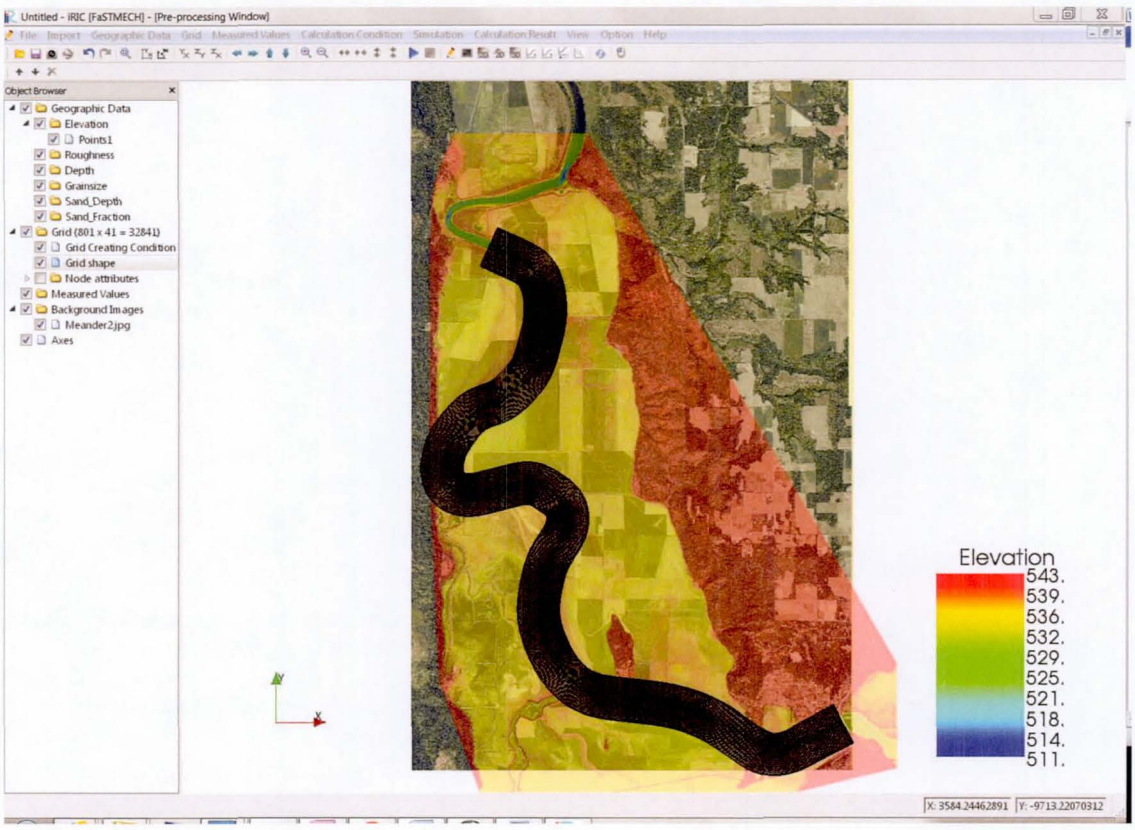


Figure 2. A user-defined grid following the channel of the Kootenai River.

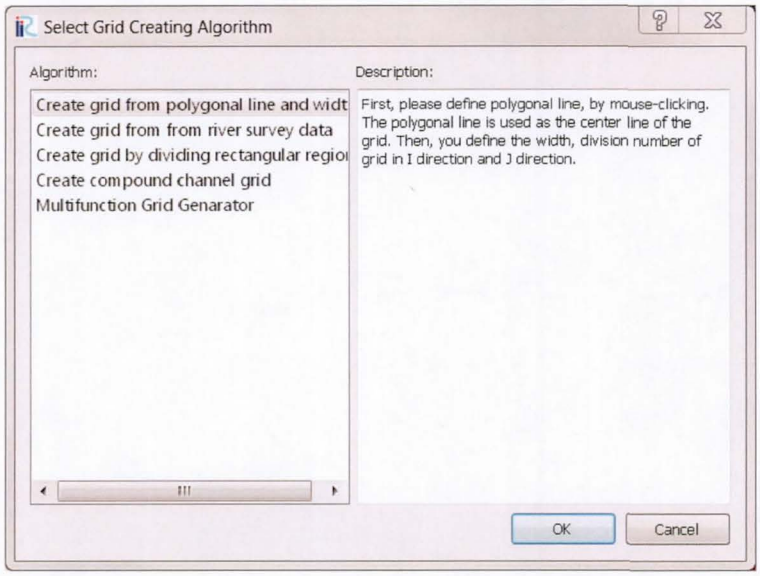


Figure 3. Select Grid Creating Algorithm dialog.

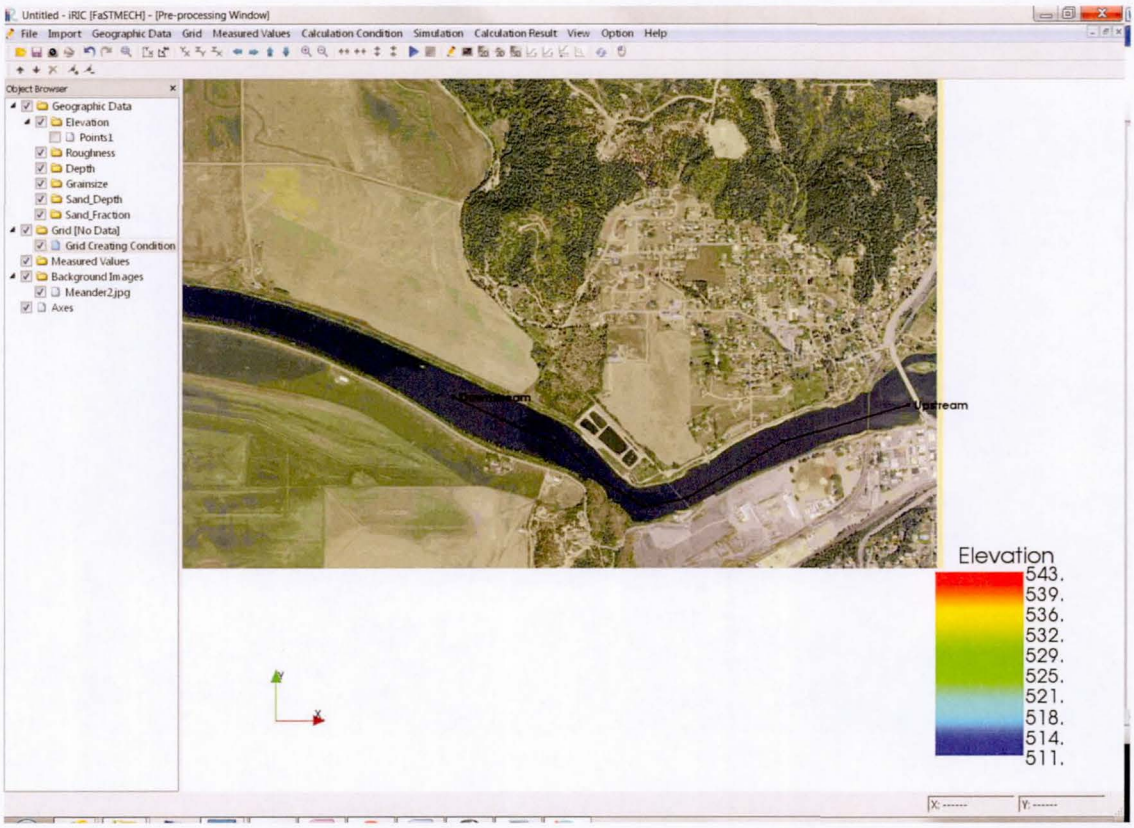


Figure 4. Process to draw the grid centerline. Note the Upstream and Downstream labels are defined.

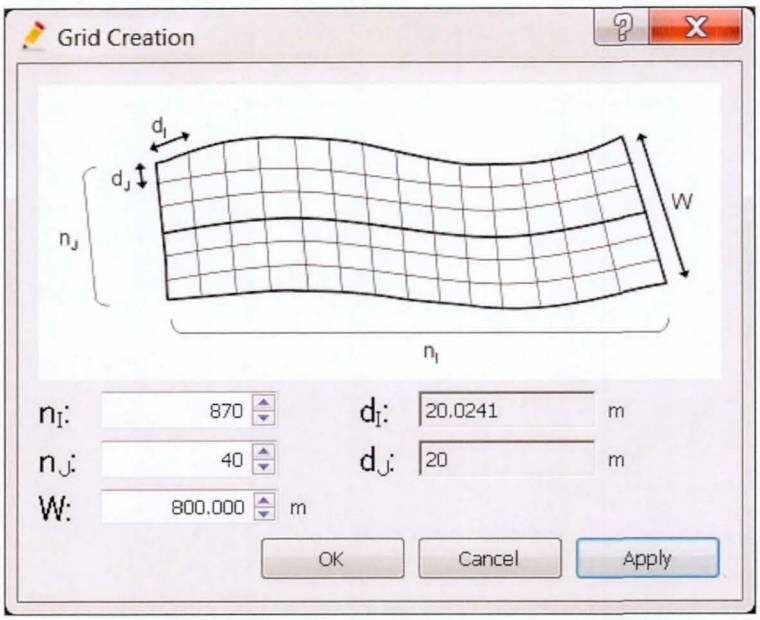


Figure 5. The Grid Creation dialog

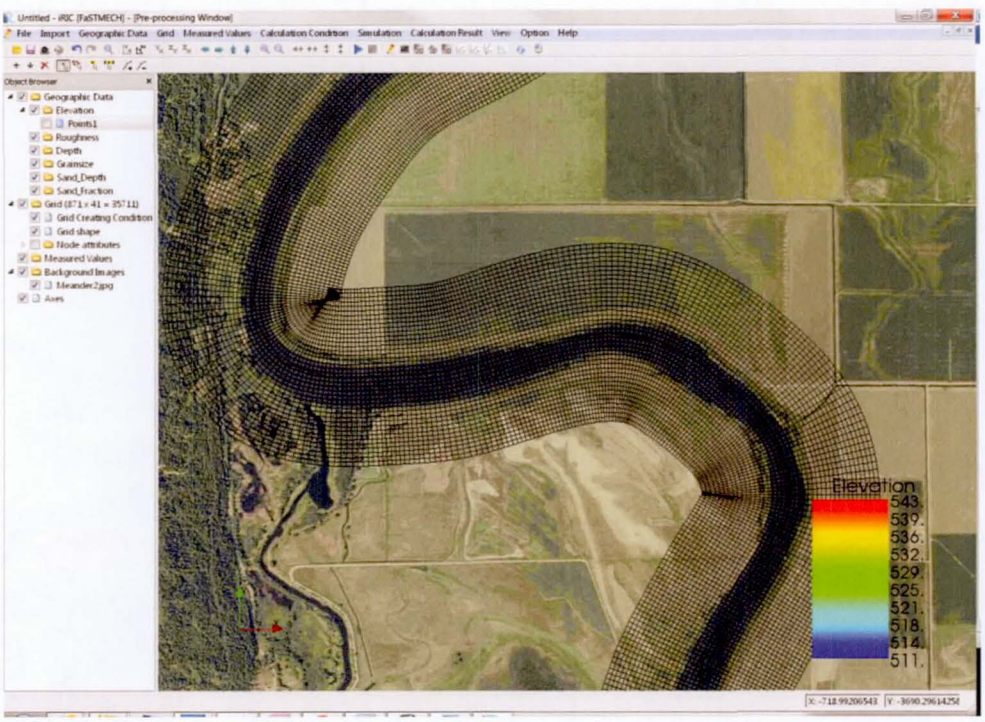


Figure 6. Curvilinear grid with grid overlap.

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.

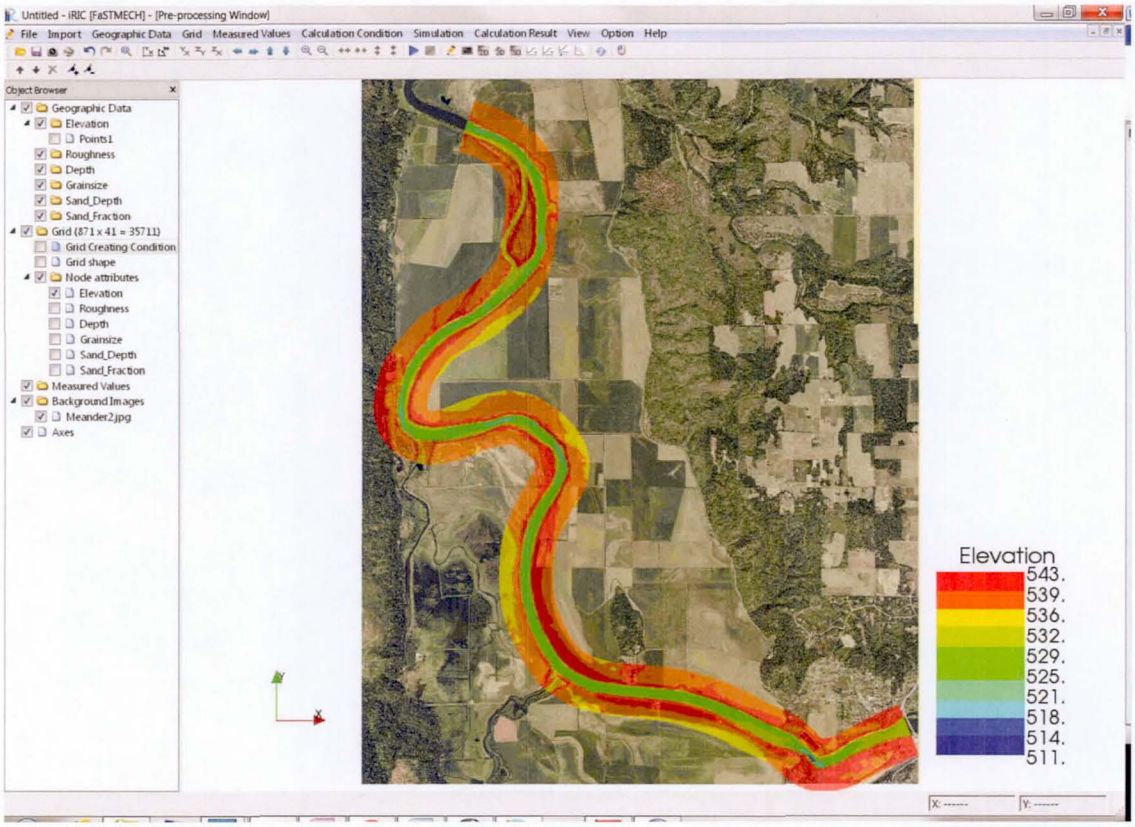


Figure 8. Elevation mapped to the grid using the TIN Algorithm.

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

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

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 and open the ketchupisl.tpo file.
3. From the Object Browser right-click Background Images and select Add Image in the resulting pop-up menu and open ketchup2.jpg (Figure 9).
4. 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.
5. 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).
6. Continue with Option 2 – step 2.

Option 2

1. From the Menu Bar select File -> Open and find the following project file iRIC Tutorials\FaSTMECH\Tutorial 1\Exercise 2\ Ketchup.ipro.
2. 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).
3. 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.
4. 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.
5. 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).
6. Save the file.

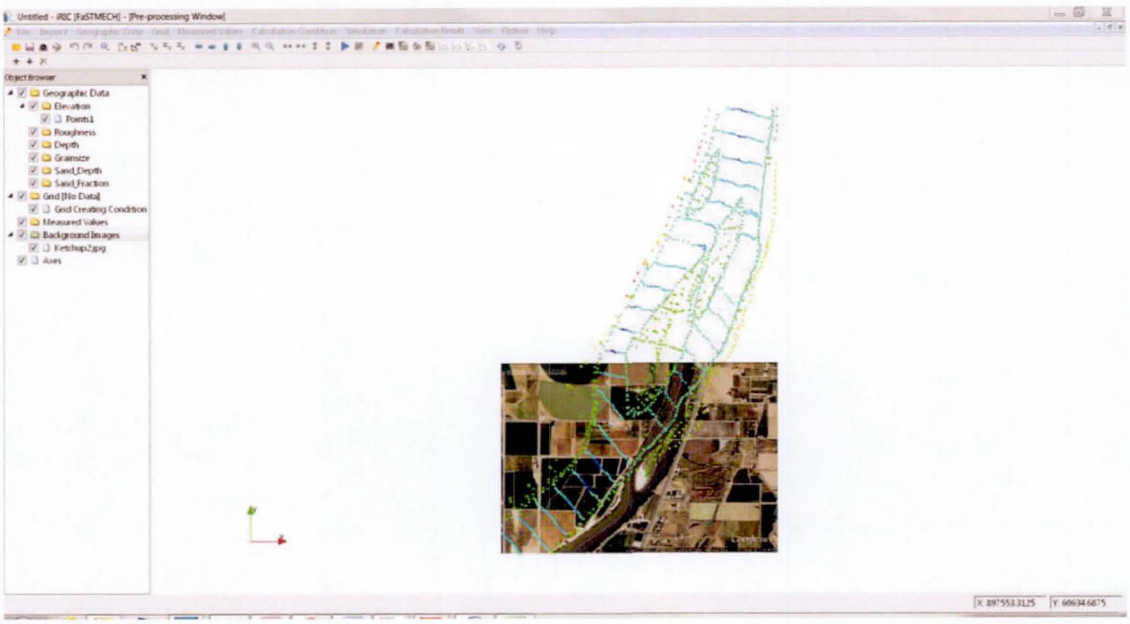


Figure 9. A non-geo-referenced image imported into iRIC.

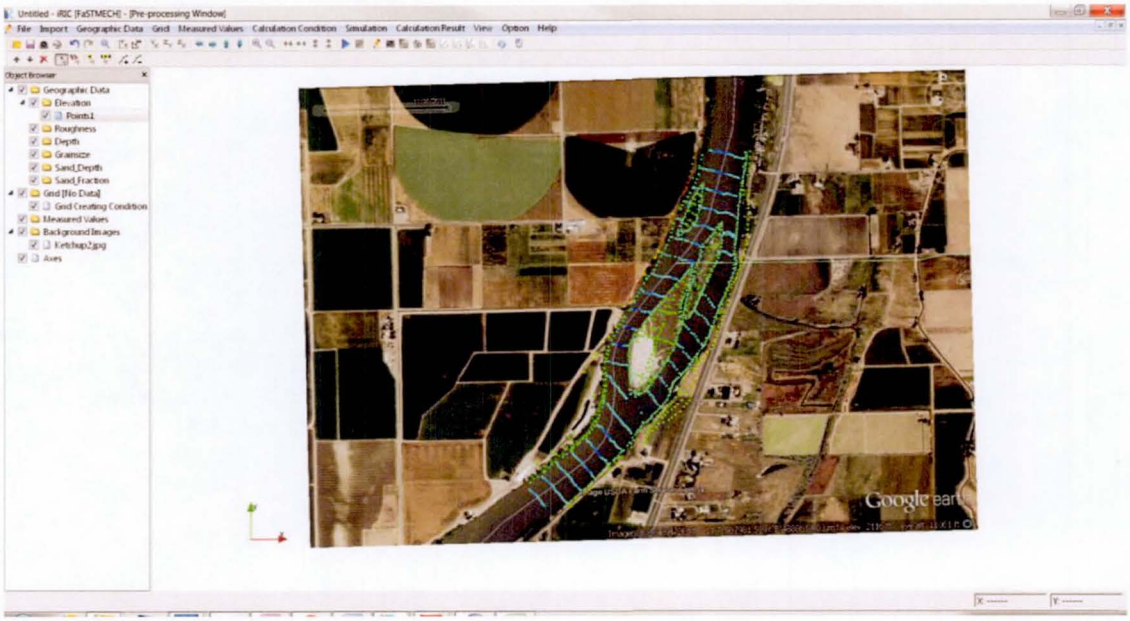


Figure 10. The background image fit to the data.

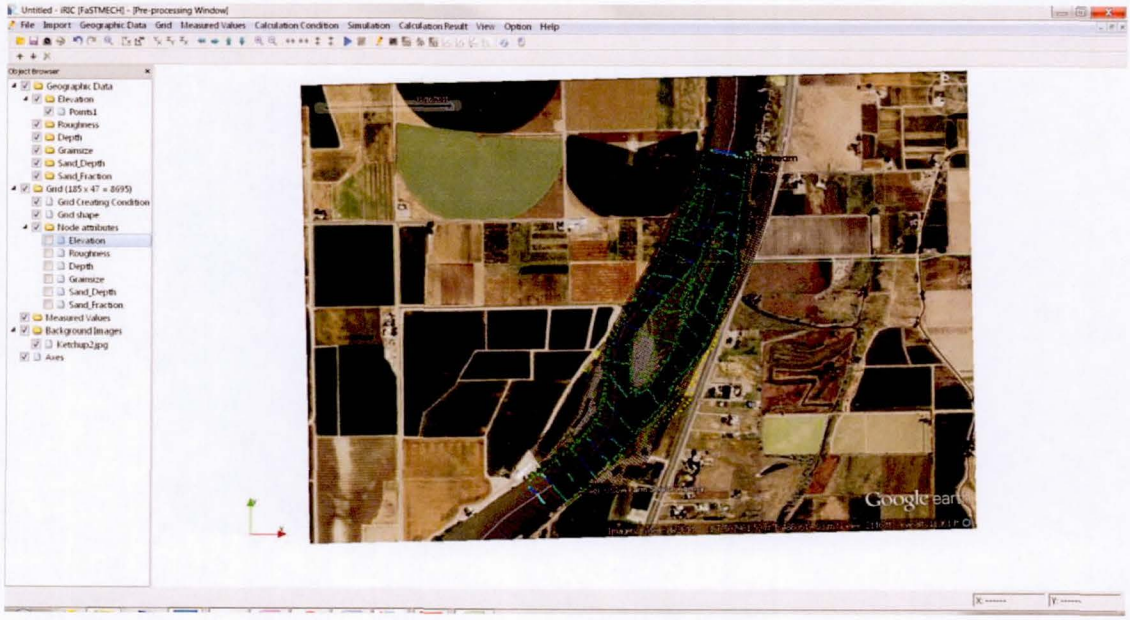


Figure 11. Location of the grid.

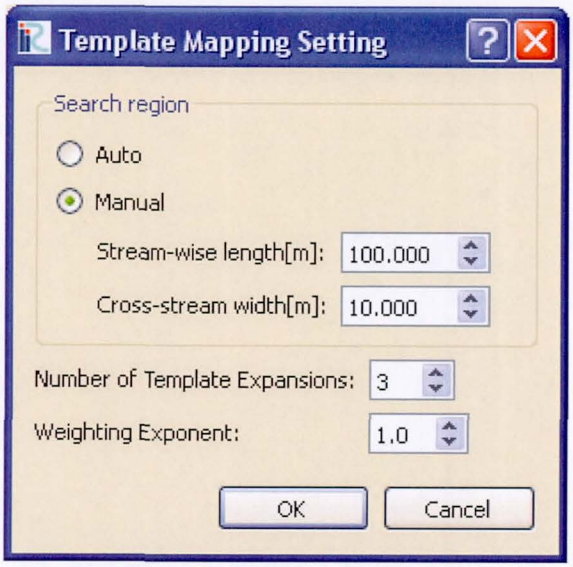


Figure 12. Template mapping dialog

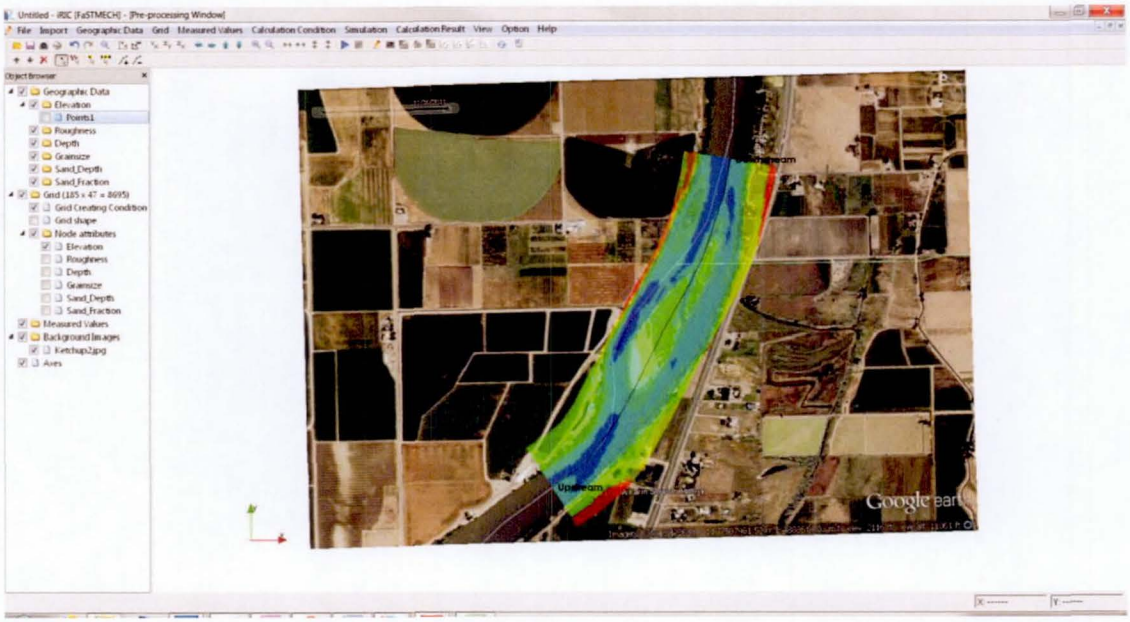


Figure 13. Elevation mapped to the grid with the template algorithm.

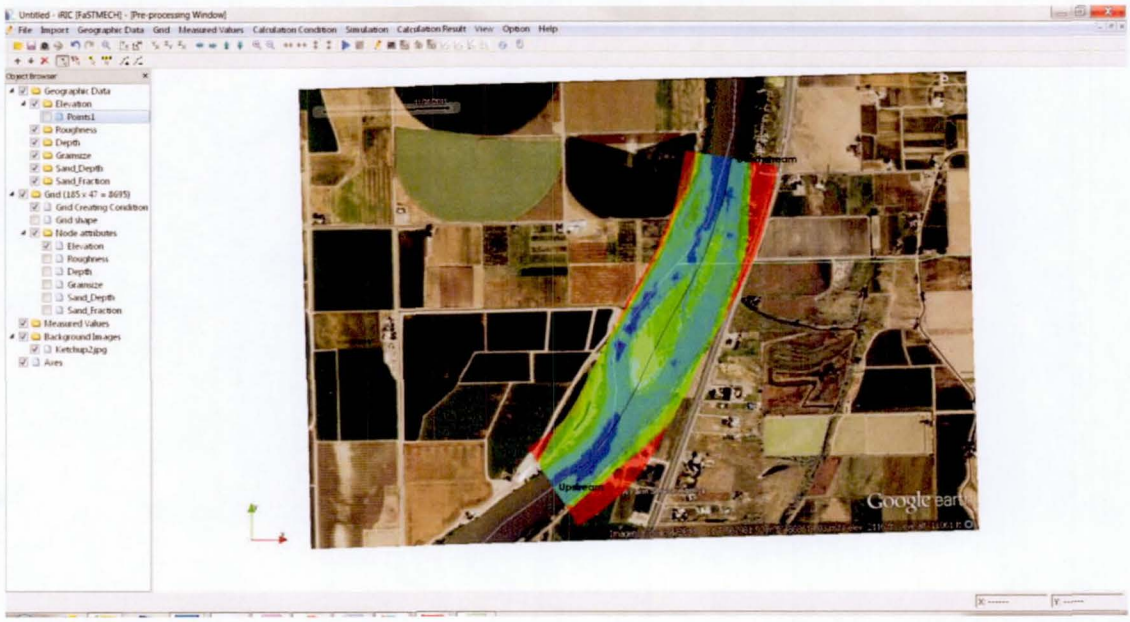


Figure 14. Elevation mapped to the grid with the TIN algorithm

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.

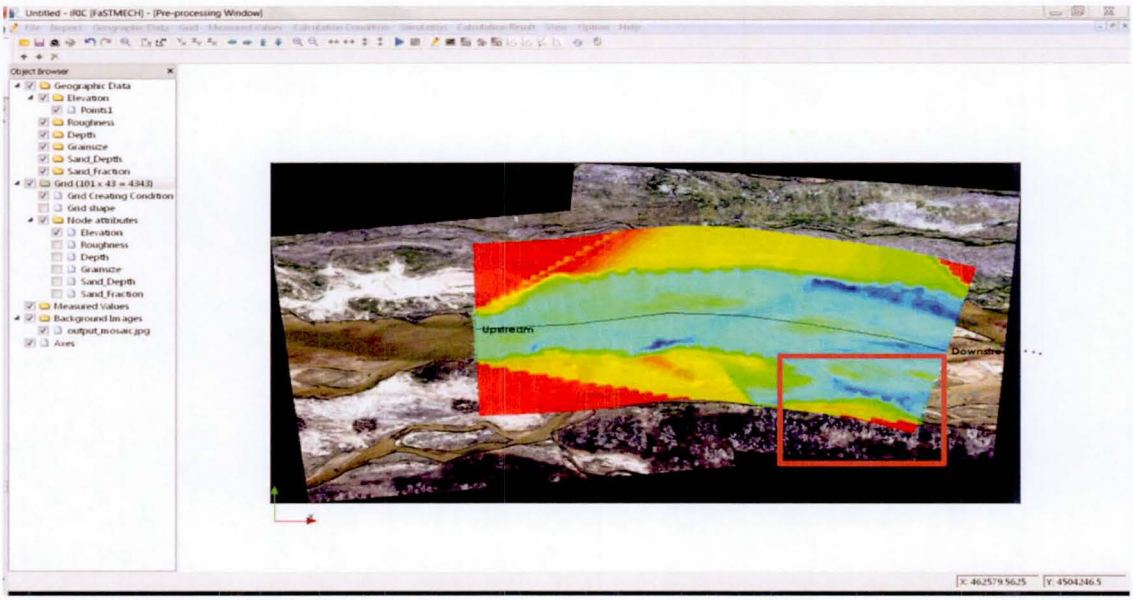


Figure 15. Location (red box) of reach that displays cusping along the bank.

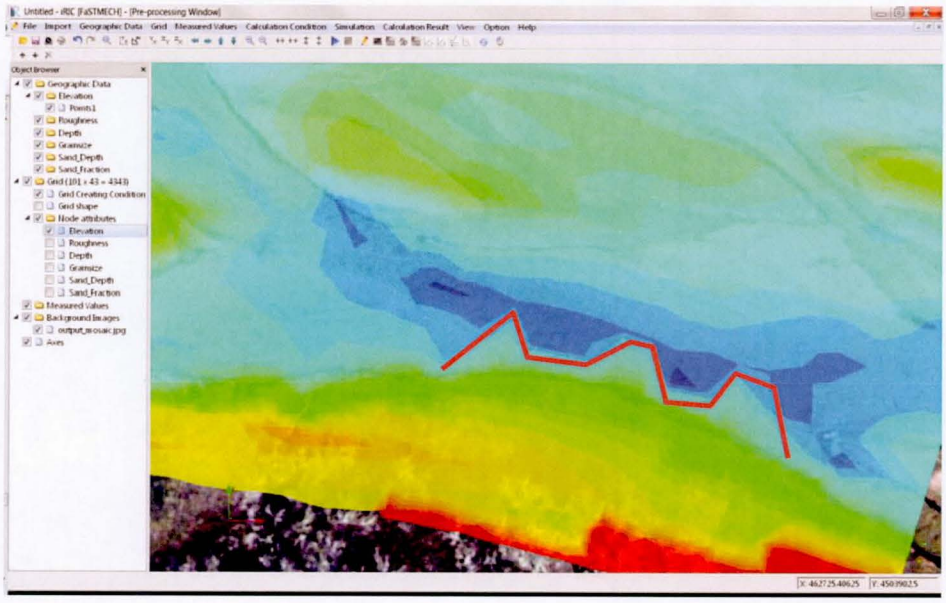


Figure 16. Cusps along the bank from the TIN algorithm

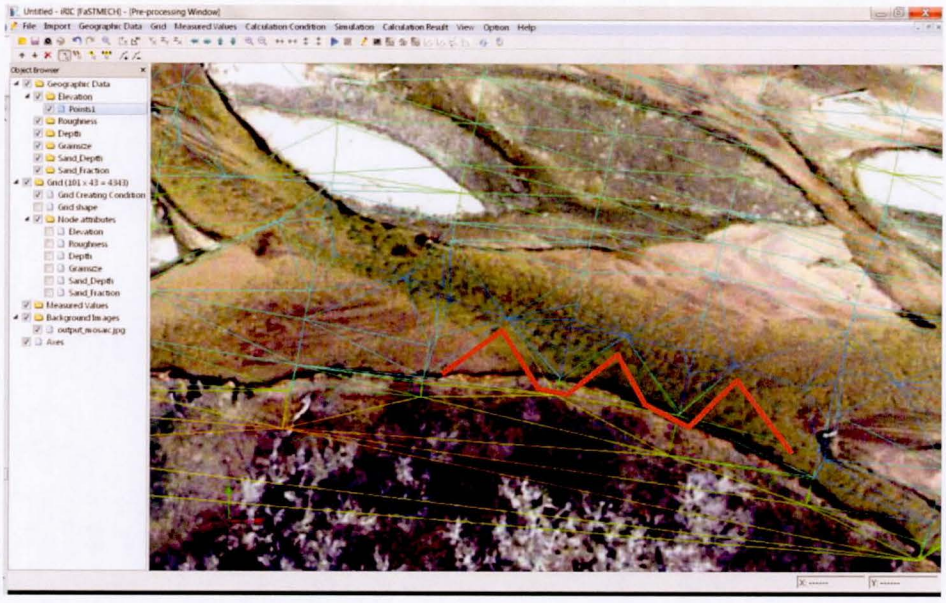


Figure 17. The TIN and location of cusps.

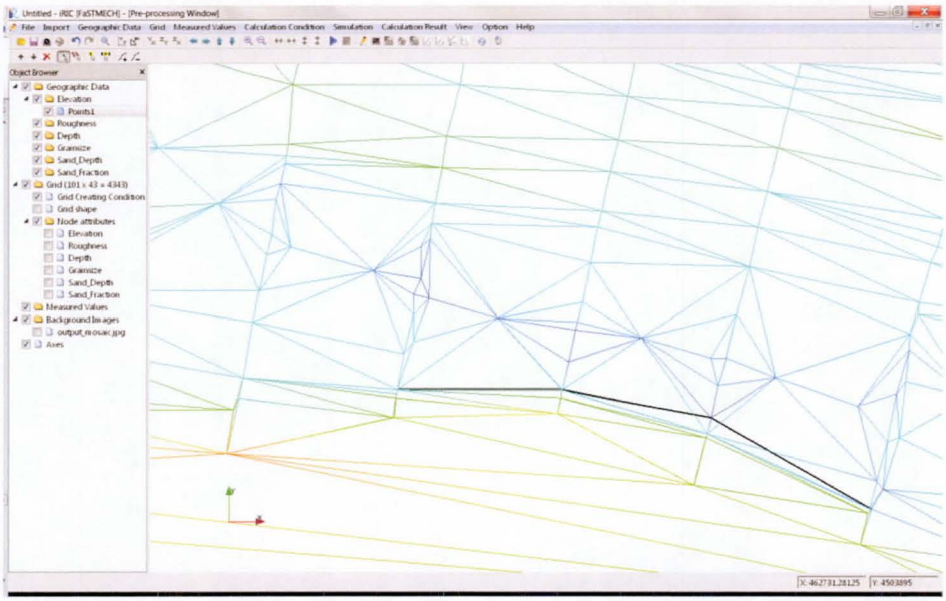


Figure 18. Location of breakline (black line).

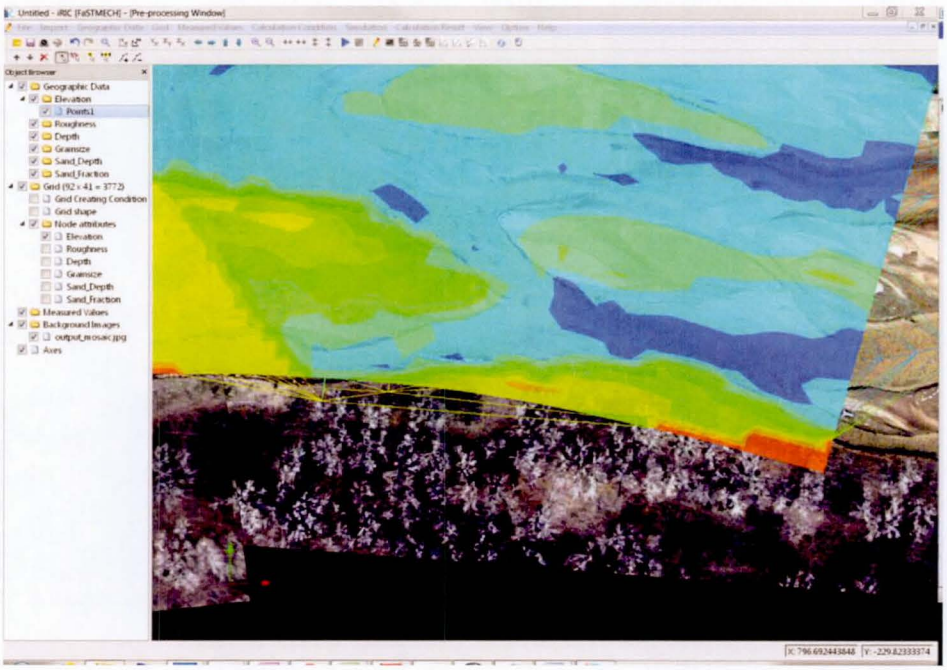


Figure 19. Result of mapping elevation with the TIN and breakline. Note cusping is gone. Compare to Figure 16.

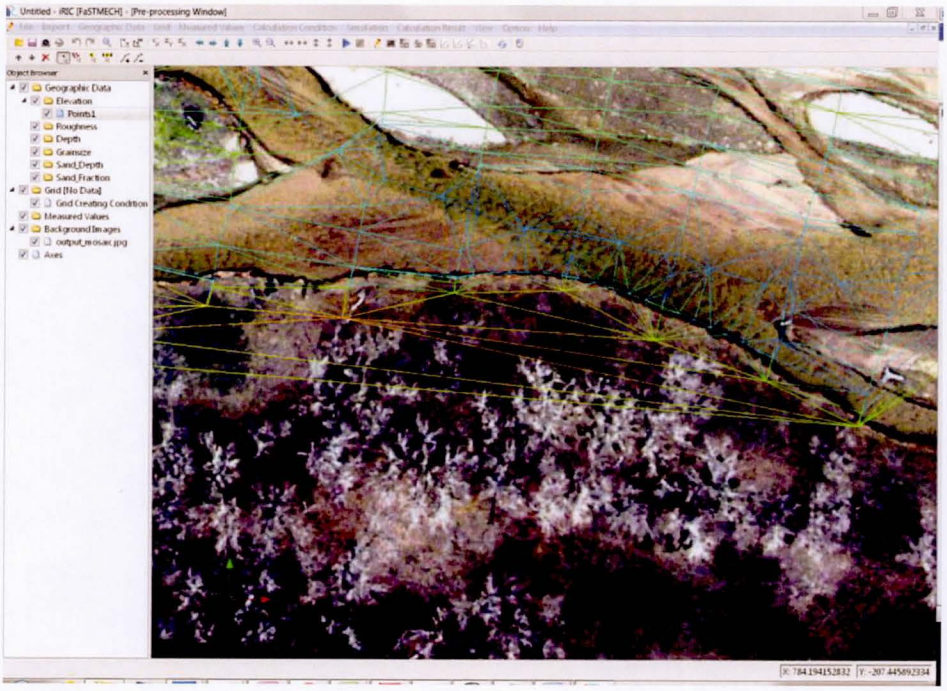


Figure 20. TIN after creating an interpolation line along the bottom of the bank and Re-meshing the TIN

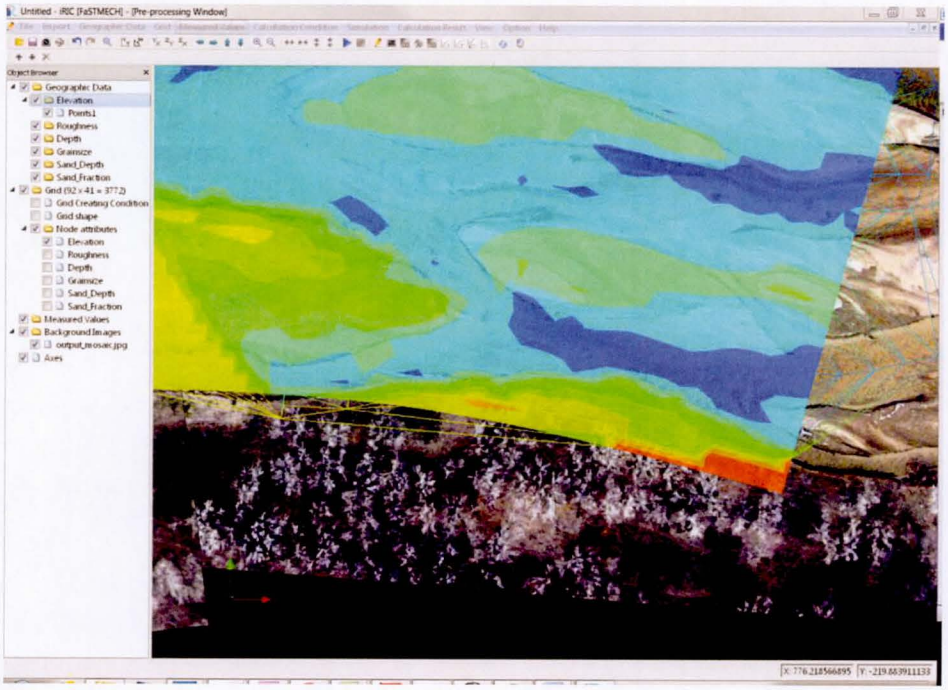


Figure 21. The topography mapped to the grid following the addition of an interpolation breakline. Compare with Figure 19.

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.

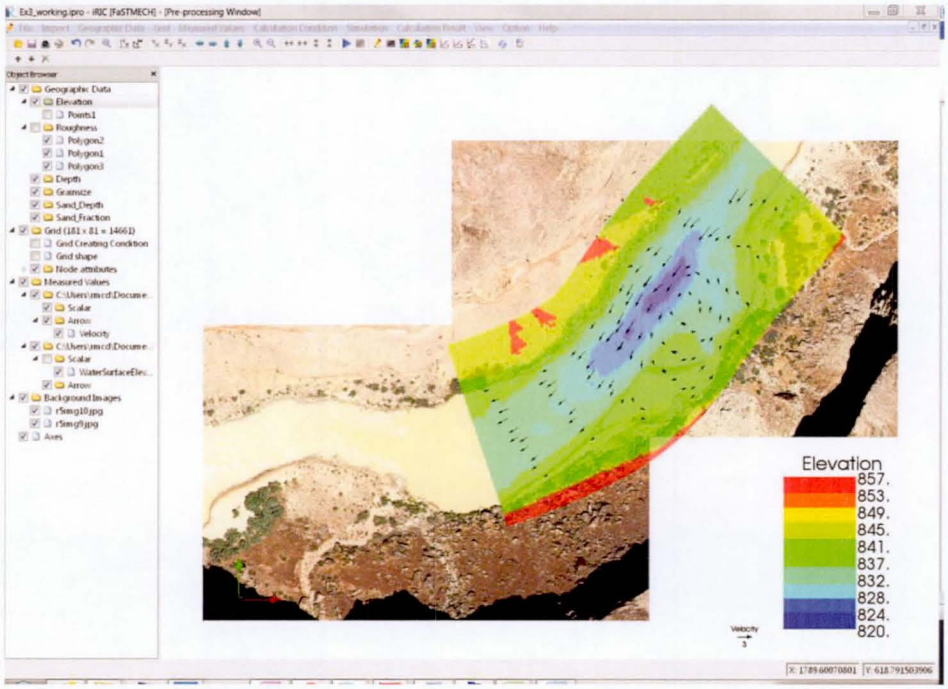
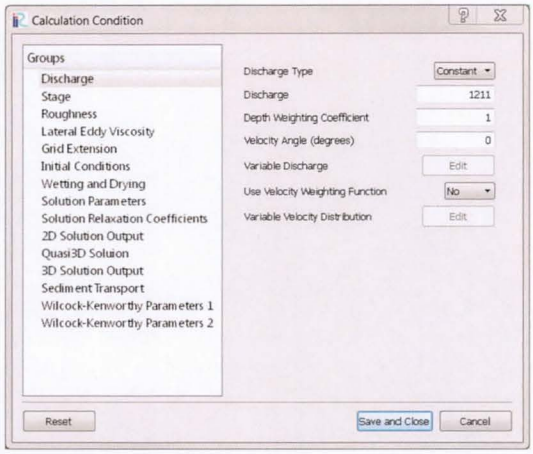


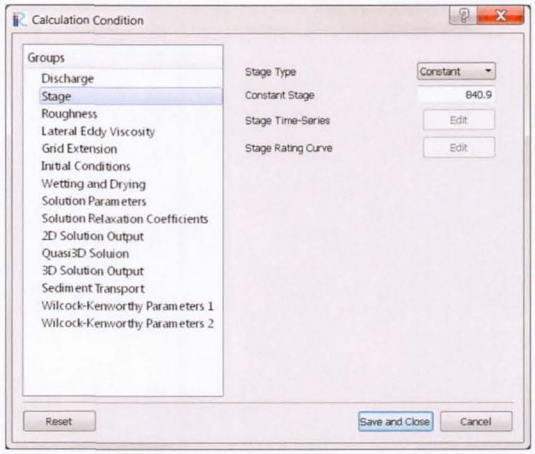
Figure 1. Domain for Exercise 3 including the measured velocity and gridded elevation.

Defining Calculation Conditions

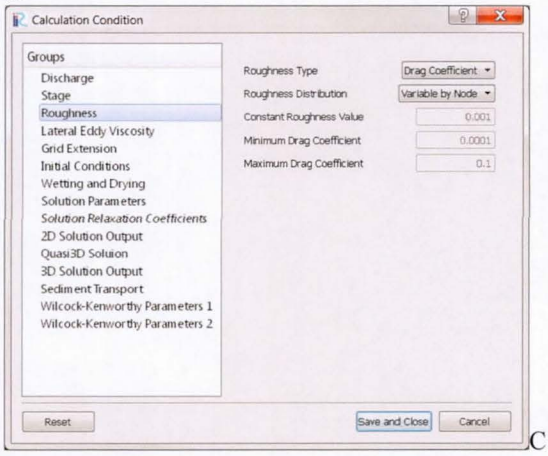
- Open an existing project from Menu Bar by selecting *File* **->***Open* and select the Ex3.ipro project file in the Tutorial 1\Exercise 3 folder (Figure 1).
- To define the calculation conditions select Menu Bar select *Calculation Conditions* **->***Setting.* Enter the parameters as shown in Figure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

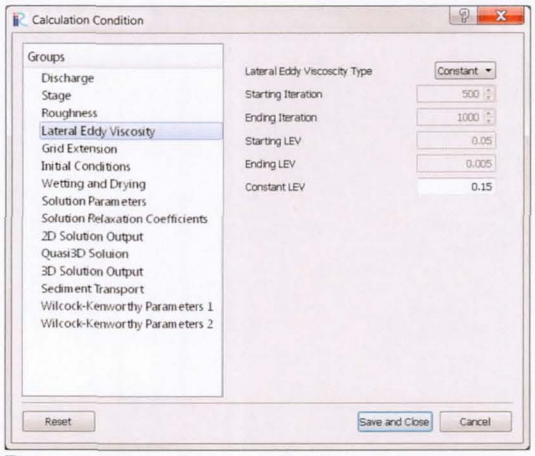
A


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

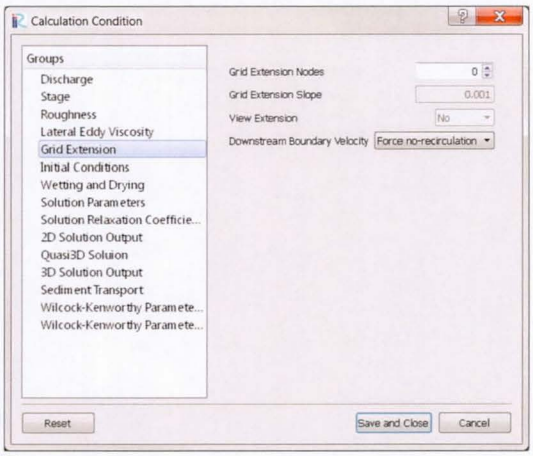
B


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

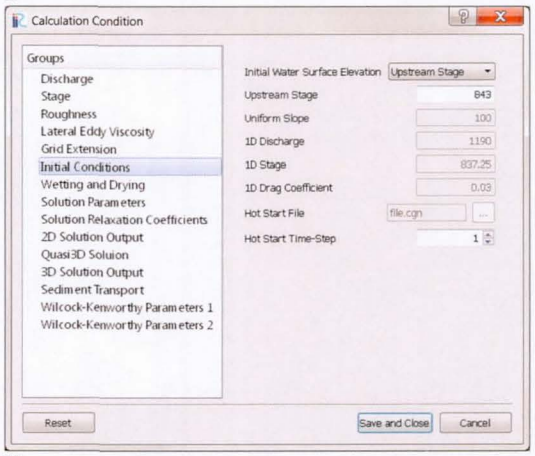
C


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 Viscosity 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

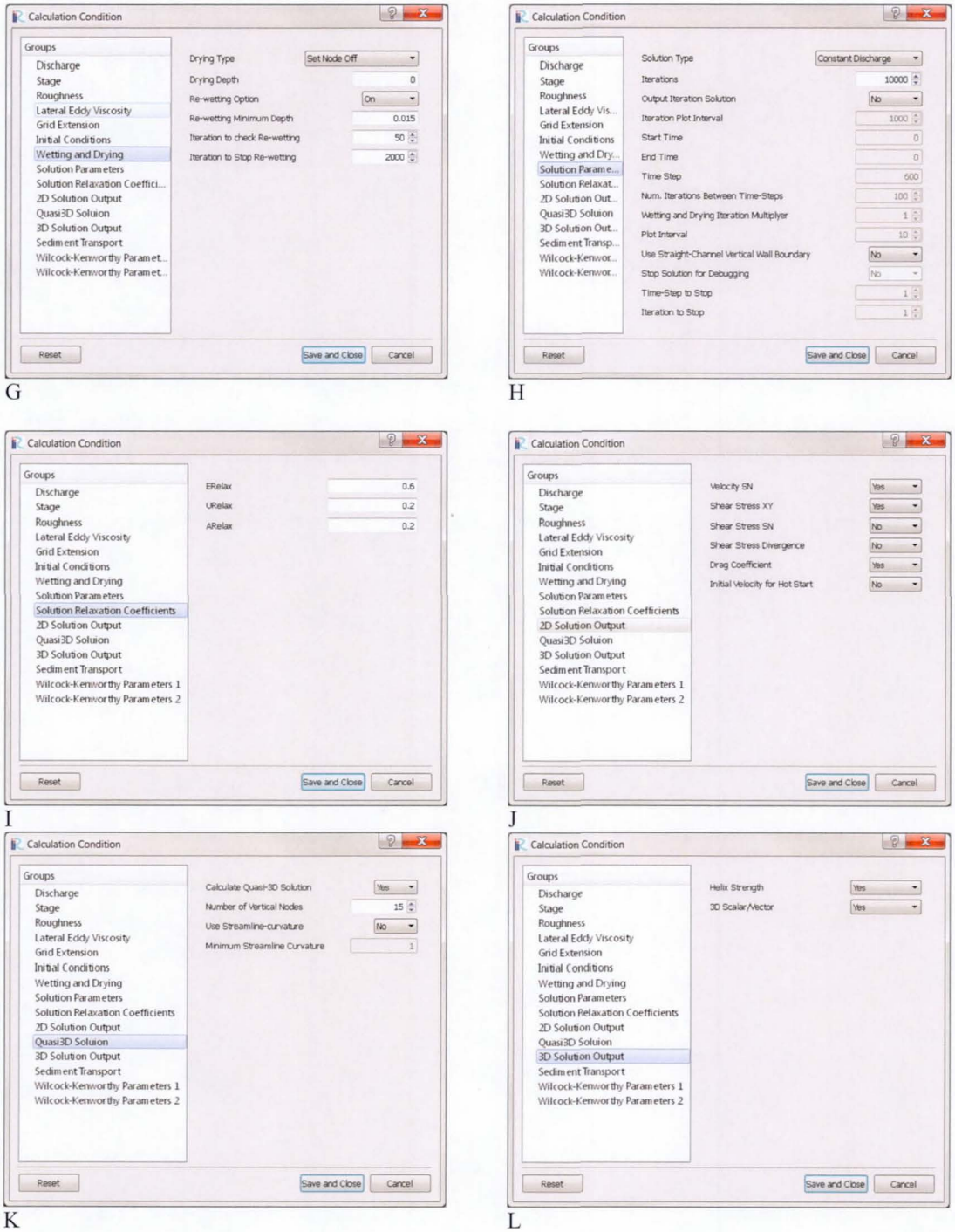


Figure 2. A. Discharge, B) Stage, C) Roughness, D) Lateral Eddy Viscosity, E) Grid Extension, F) Initial Condition, G) Wetting and Drying, H) Solution Parameters, I) Solution Relaxation Coefficients, J) 2D Solution Output, K) Quasi 3D Solution, L) 3D Solution Output..

- 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.

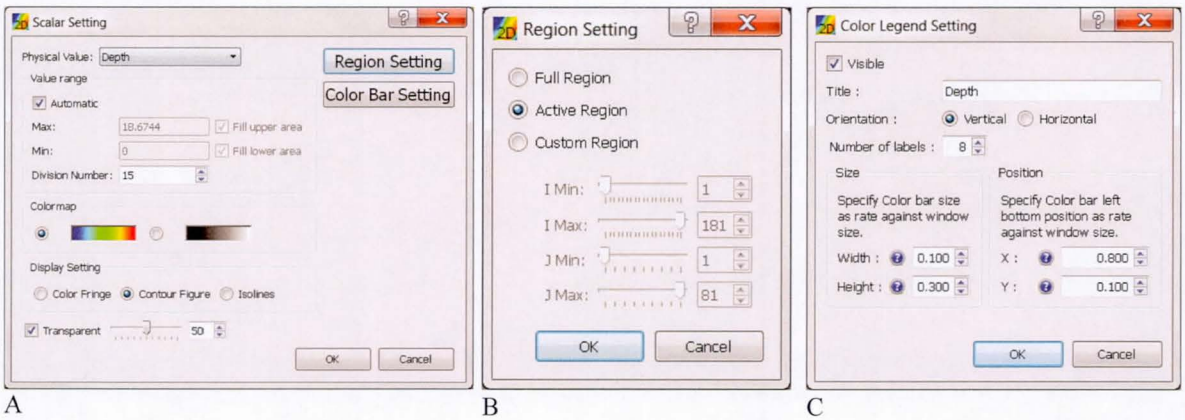
2D Post-processing

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.

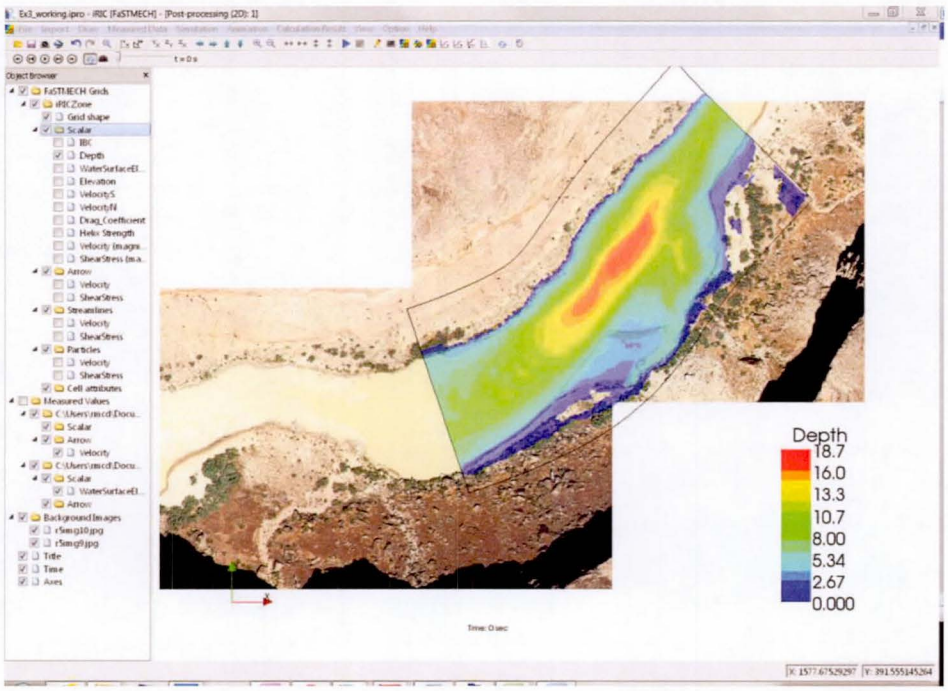


Figure 4. Scalar solution of Depth.

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.

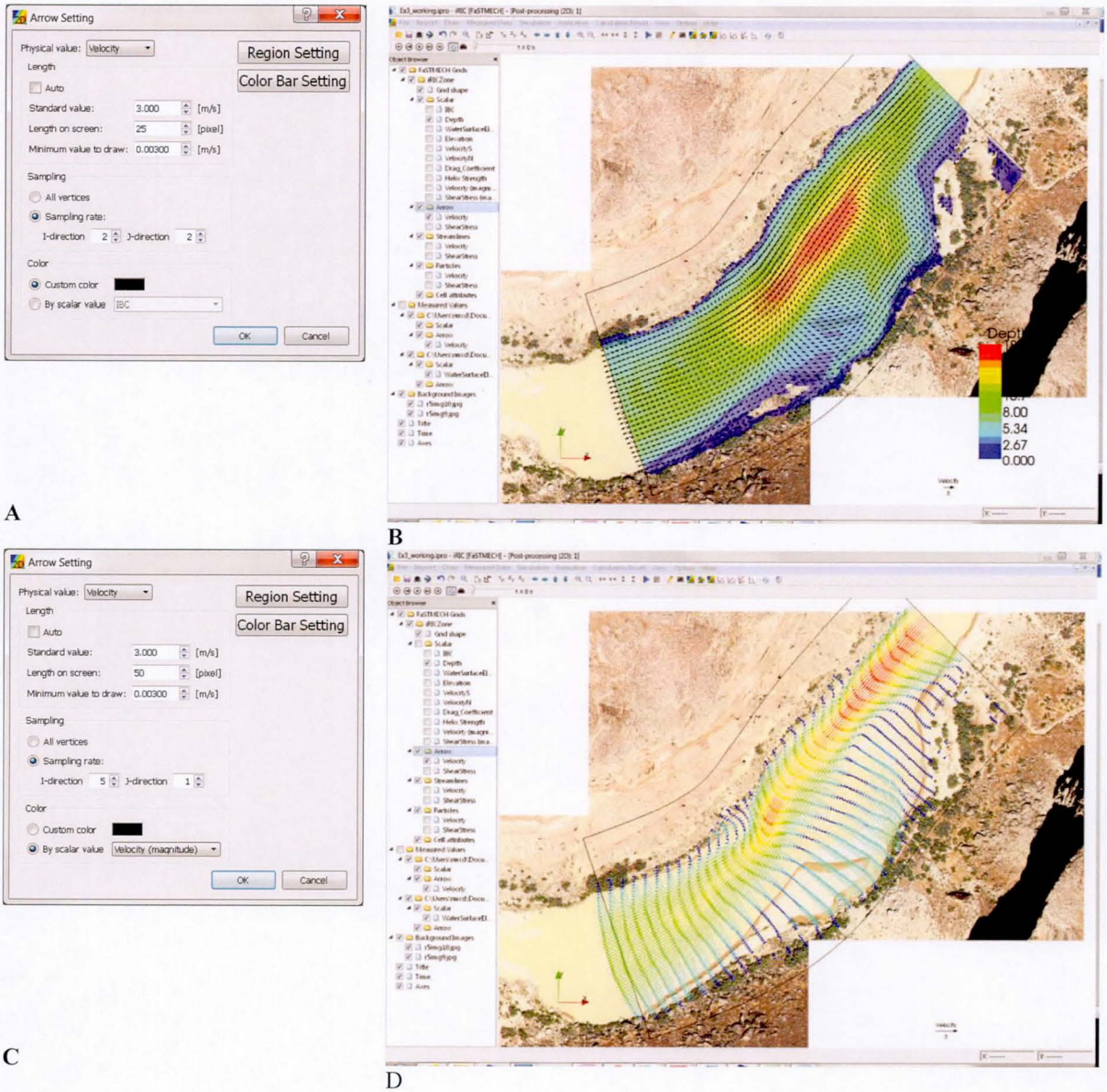


Figure 5. Arrow setting attributes and resulting plot of vectors.

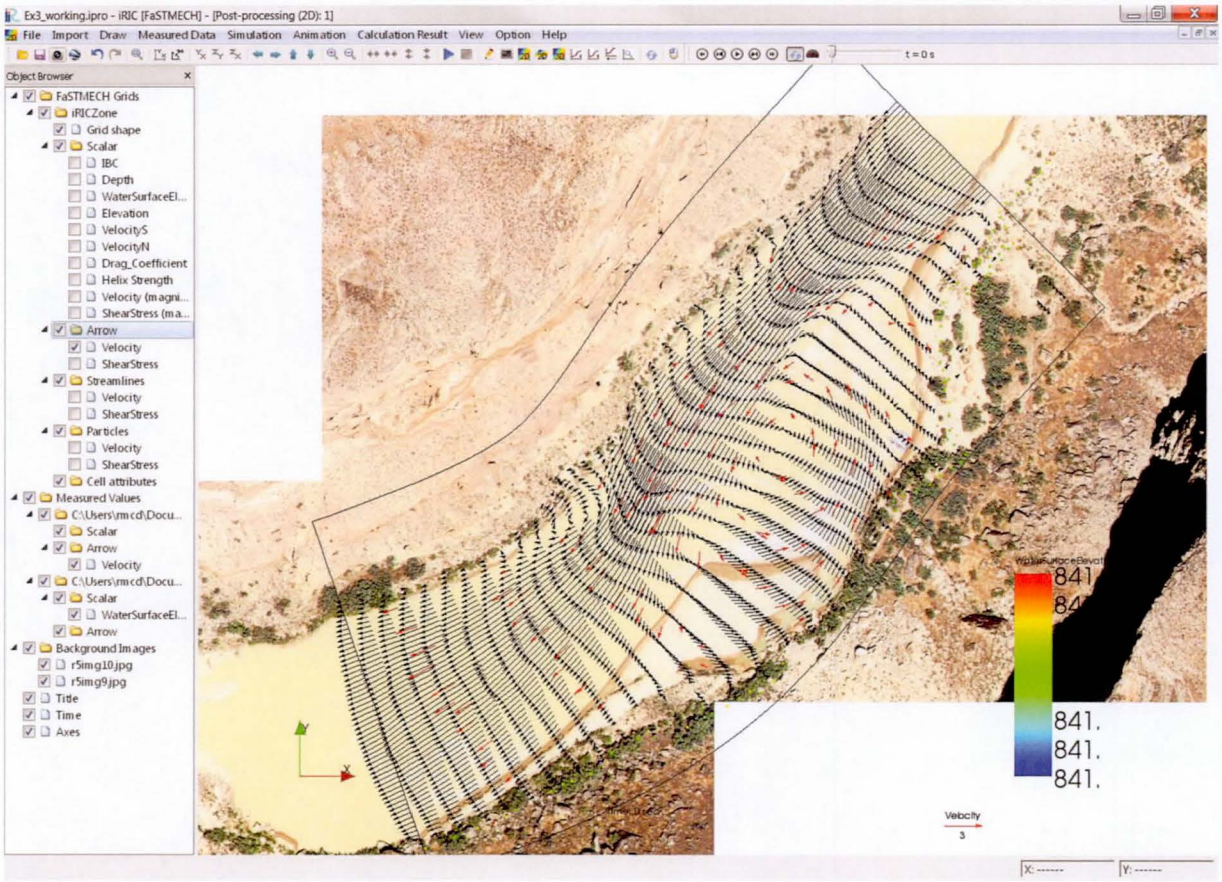
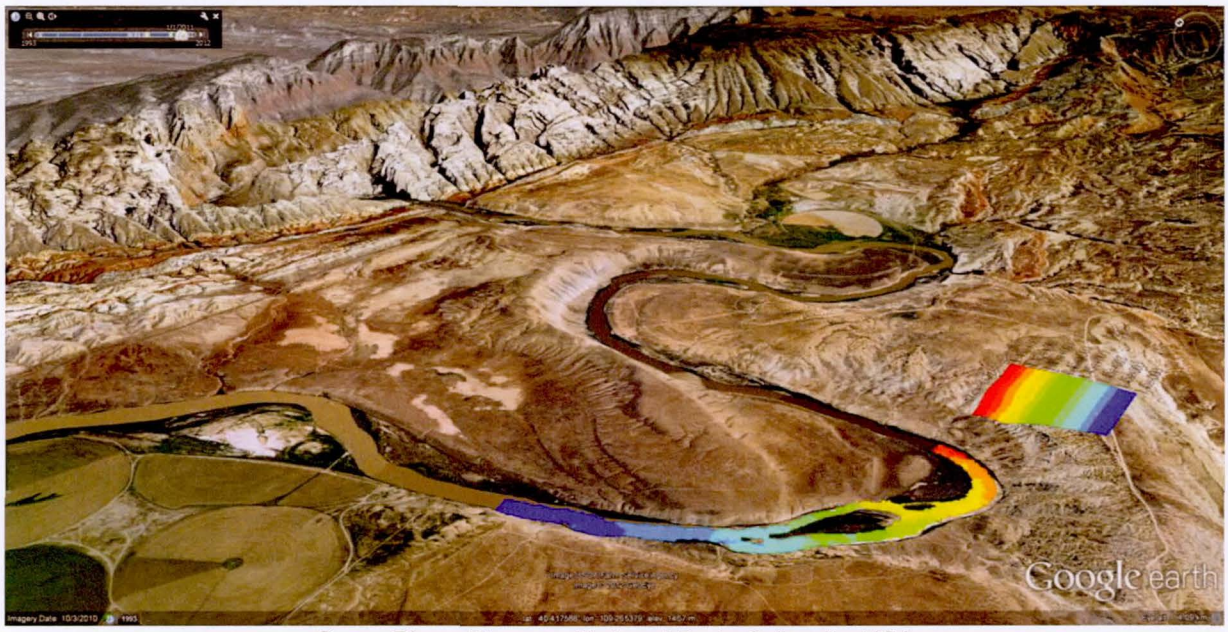


Figure 6. Solution vectors (Black) and measured vectors (Red).

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 6
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
Quasi-3D Model and 3D Graphics 23
Visualize Two- and Three-Dimensional Solution Without and With Streamline Curvature 23
Vertical Structure Continued 25

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

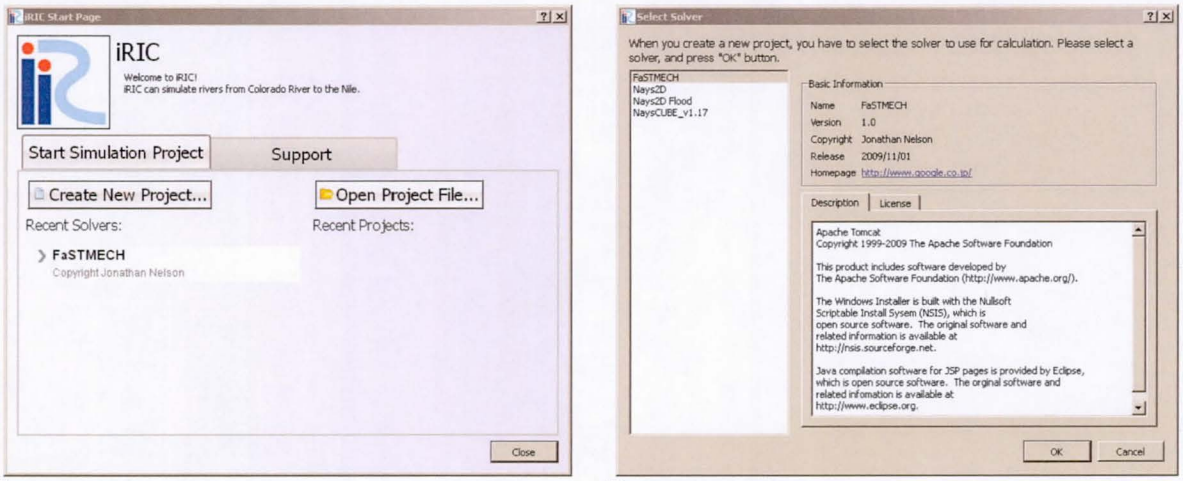
Launch iRIC 2.3 by selecting iRIC 2.3 from the Program Menu list or click on the iRIC icon on the desktop.

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: iRIC Tutorials\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).

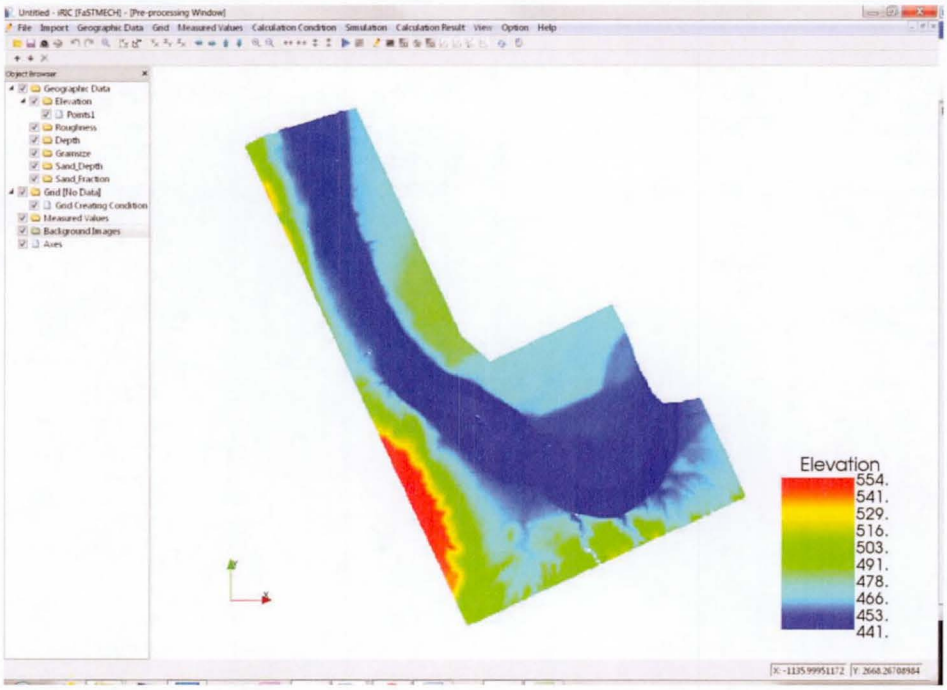


Figure 2. Pre-processing Window display of the Elevation data.

- 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).

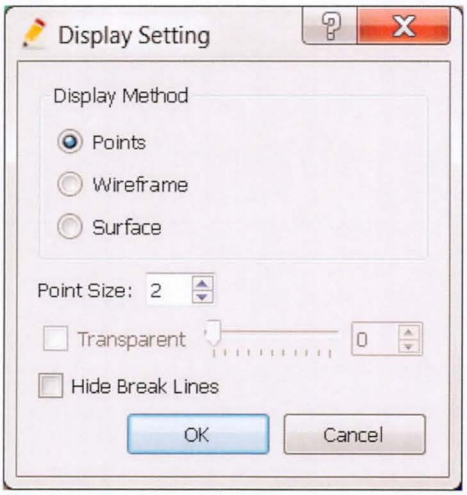


Figure 3. Display Setting dialog to select graphic attributes for the Elevation data

- 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**.

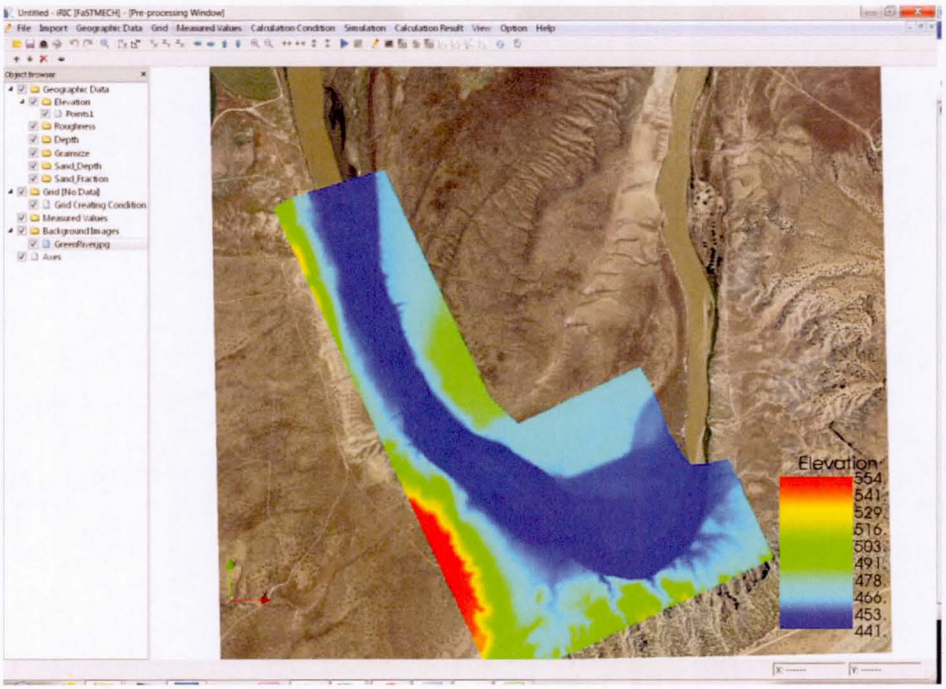


Figure 4. Addition of the background imagery.

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 Algorithm to Create Grid. 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.

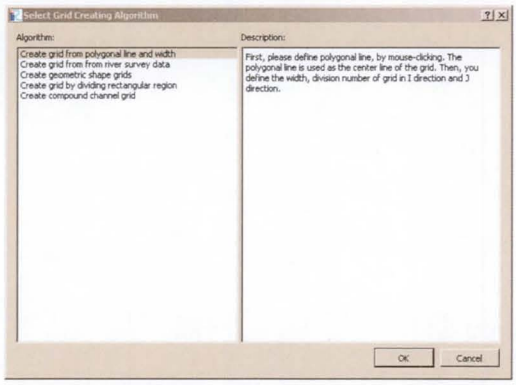


Figure 5. Example of the Select Grid Creating Algorithm dialog.

- 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.

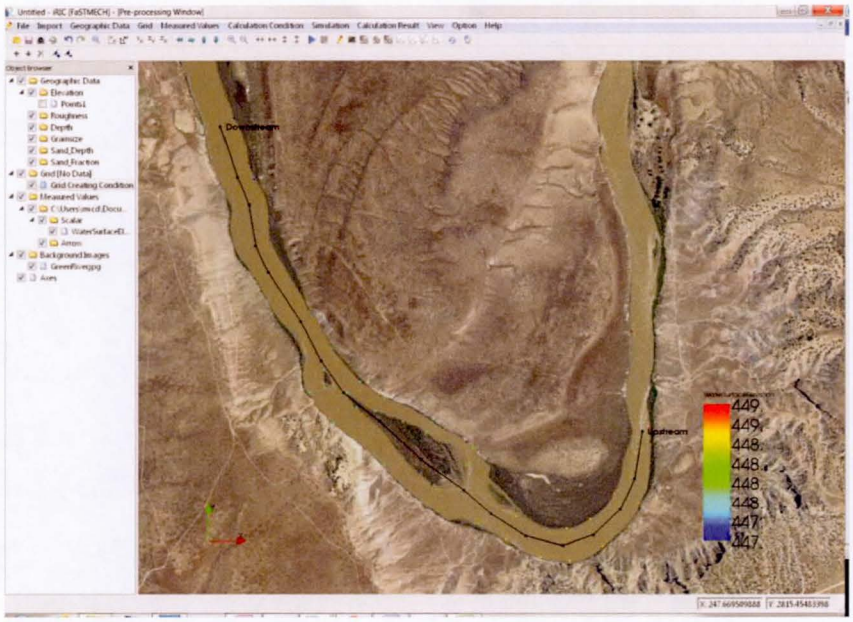


Figure 6. Location to draw centerline. Flow is from bottom to top so start at the lower end of the channel and select points ending near the top of the channel.

- 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 5 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**.

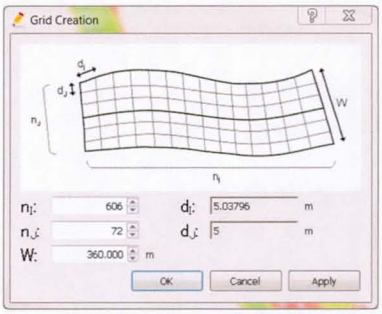


Figure 7. Grid Creation dialog that allows you to specify grid characteristics.

- 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

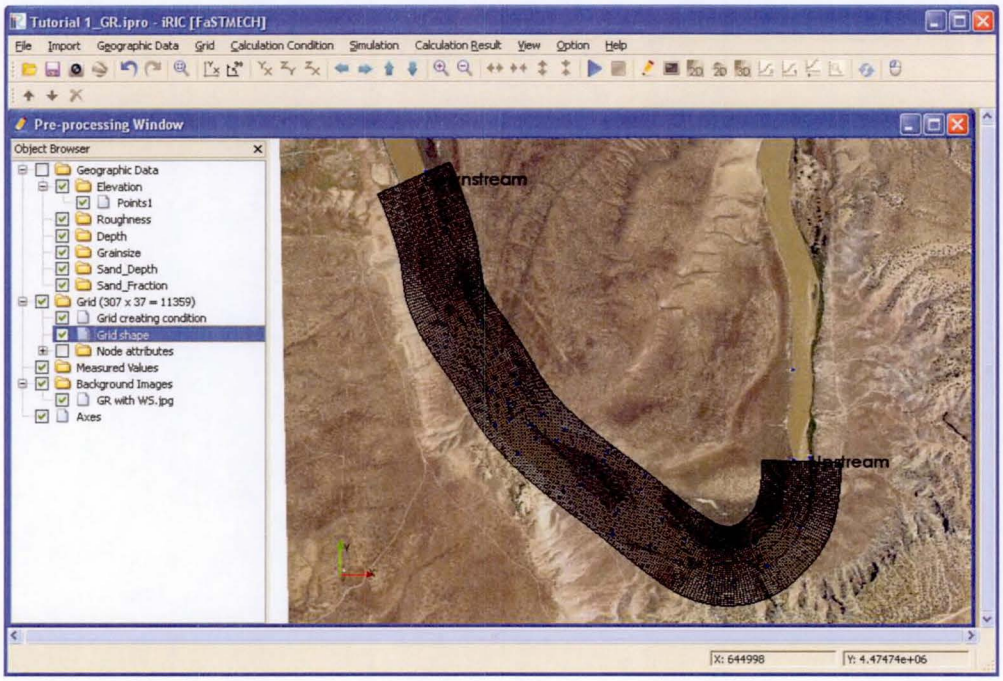


Figure 8. Example grid for the Green River.

- 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.

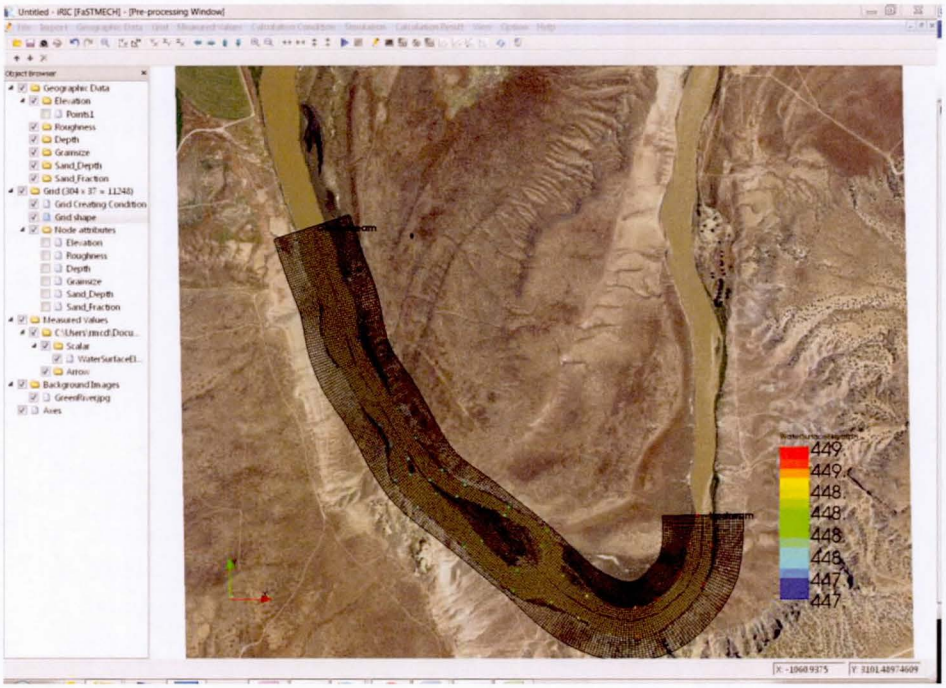


Figure 9. Edited grid location.

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 Settting 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).

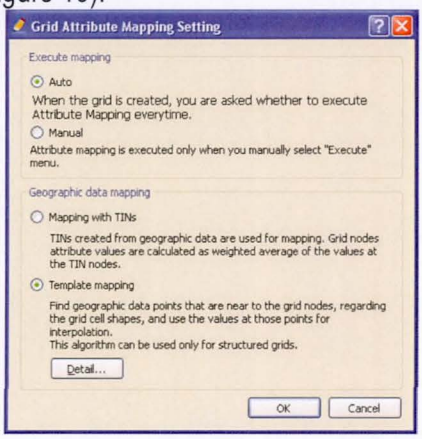


Figure 10. Grid Attribute Mapping dialog

- 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. The topography/bathymetry data set has a high density of points, and we could simply use the option to map with the TIN, however, there is quite a bit of noise in the data so we choose here to slightly smooth out some of the noise by using the template with a size of 25m and 5m in the stream-wise and cross-stream directions respectively.
-

Figure 11. Template Mapping Setting 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.

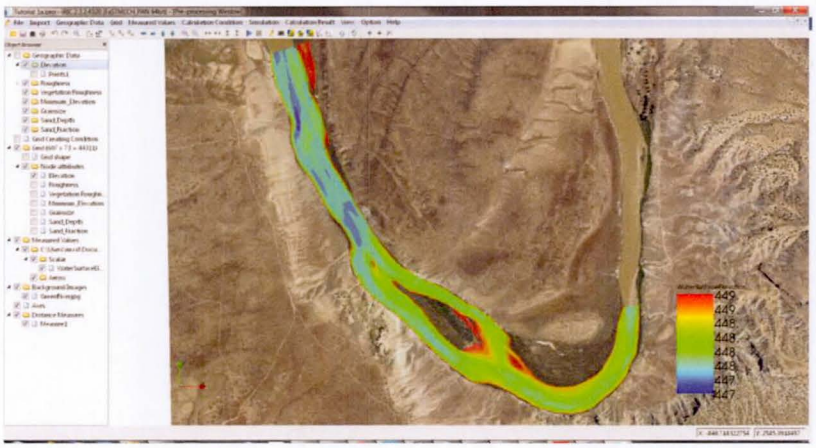


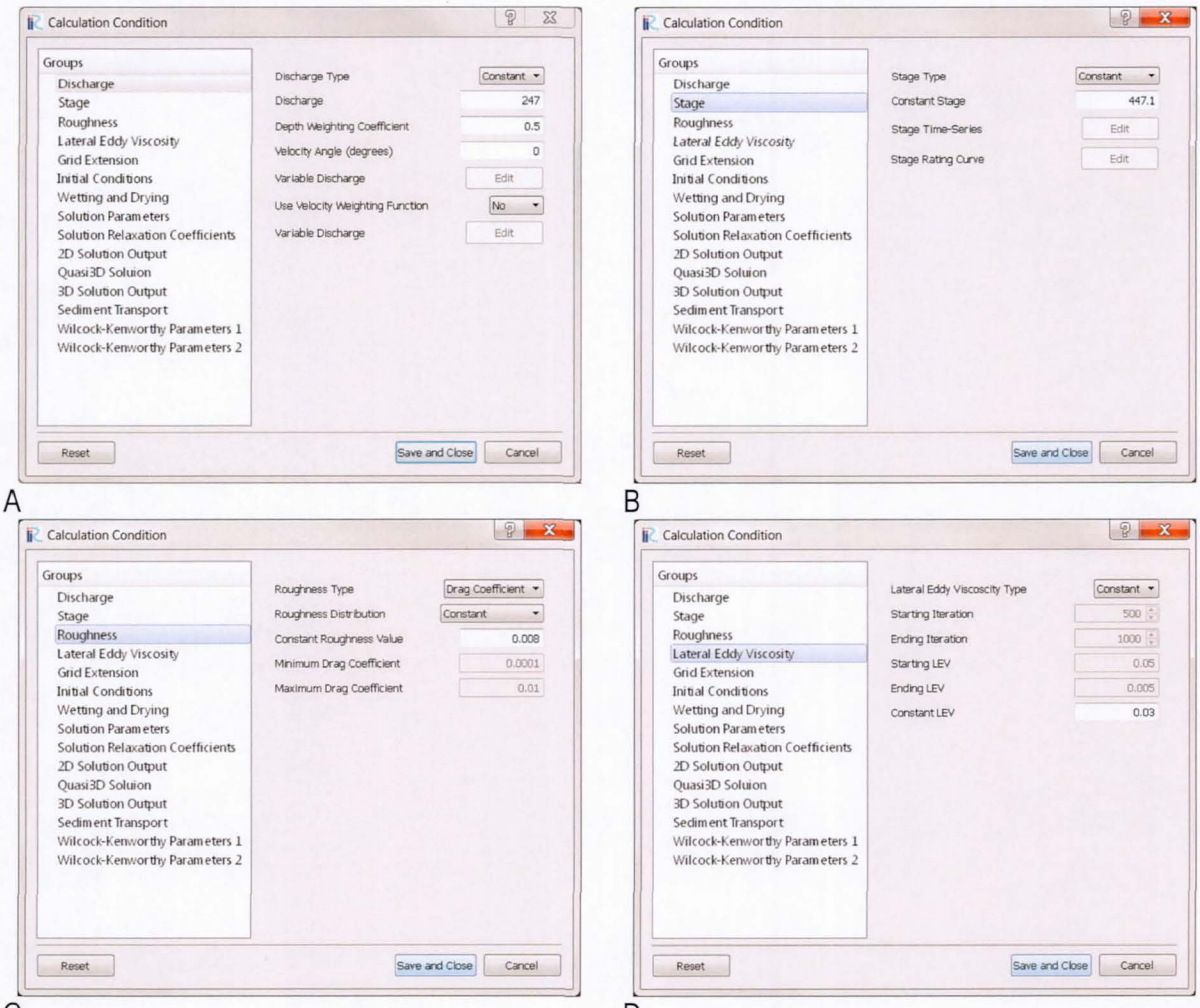
Figure 12. The resulting computational grid with the measured elevations mapped to the grid.

- 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.



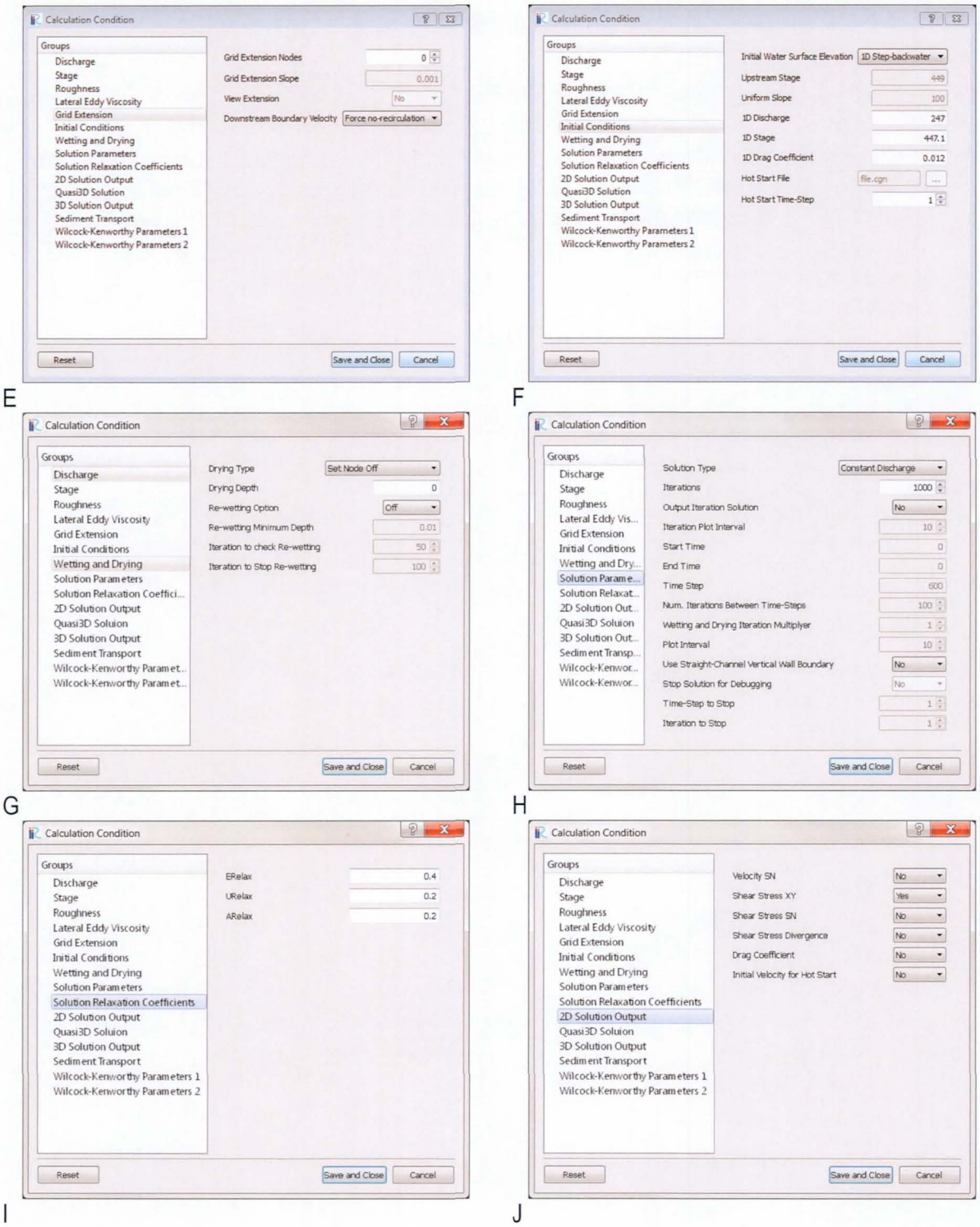


Figure 13. FaSTMECH Calculation Conditions dialog. (A) Discharge, (B) Stage, (C) Roughness, (D) Lateral Eddy Viscosity, (E) Grid Extension, (F) Initial Conditions, (G) Wetting and Drying, (H) Solution Parameters, (I) Solution Relaxation, (J) 2D Solution Output.

- 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).

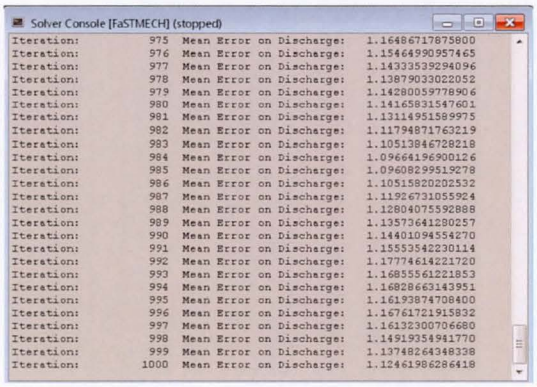


Figure 14. Example of the Solver Console.

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).

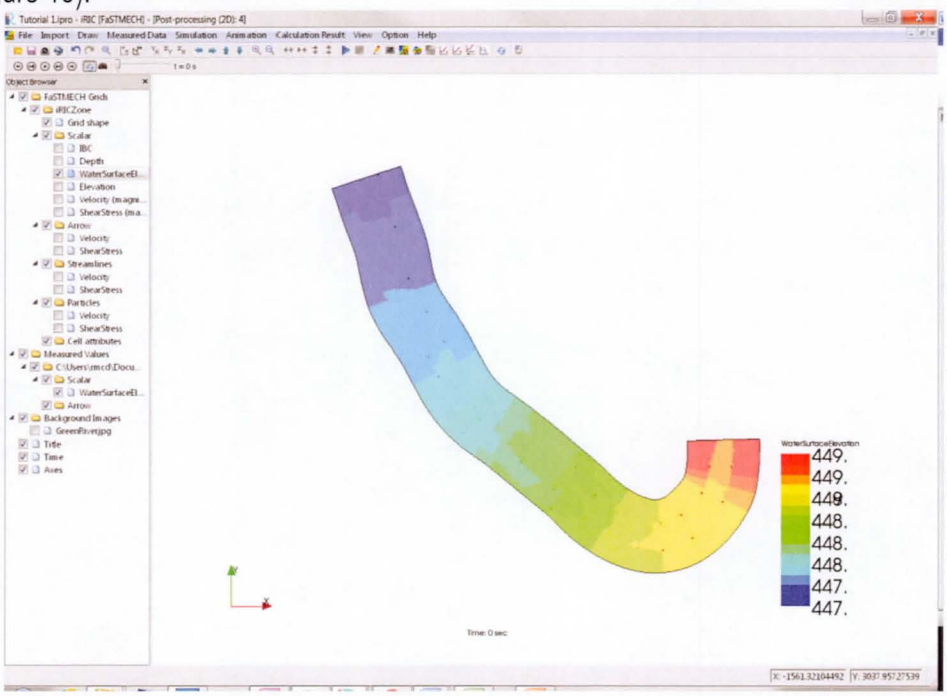


Figure 15. Initial display of the simulated water-surface elevation.

- 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.
-

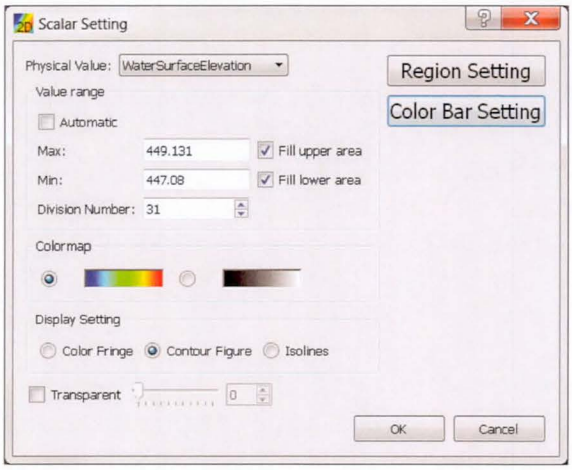


Figure 16. Scalar Setting dialog

- 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).

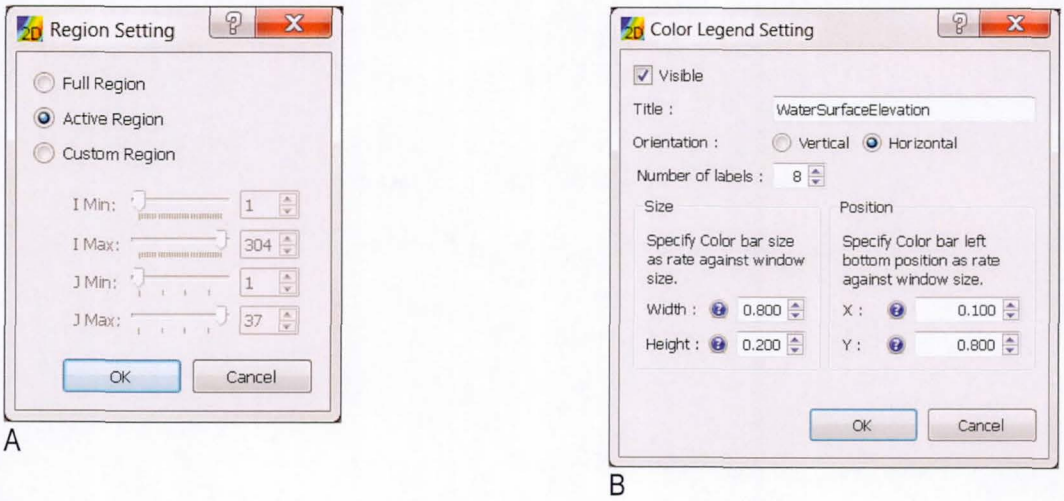


Figure 17. Additional display options available in the Scalar Setting dialog include the (A) Region Setting dialog and the (B) Color Legend Setting dialog (B).

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

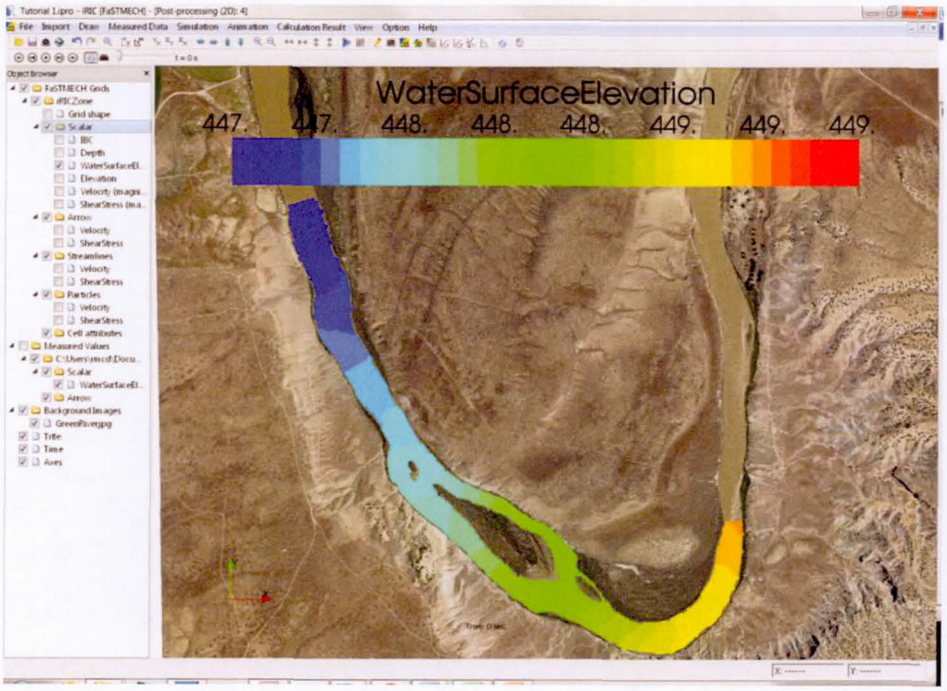


Figure 18. Example display of the simulated water-surface elevation.

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.

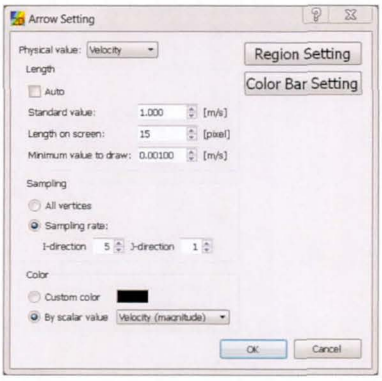


Figure 19. The Vector Attributes dialog.

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.

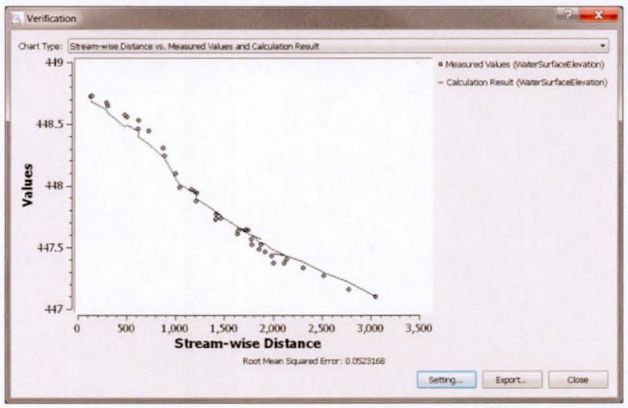
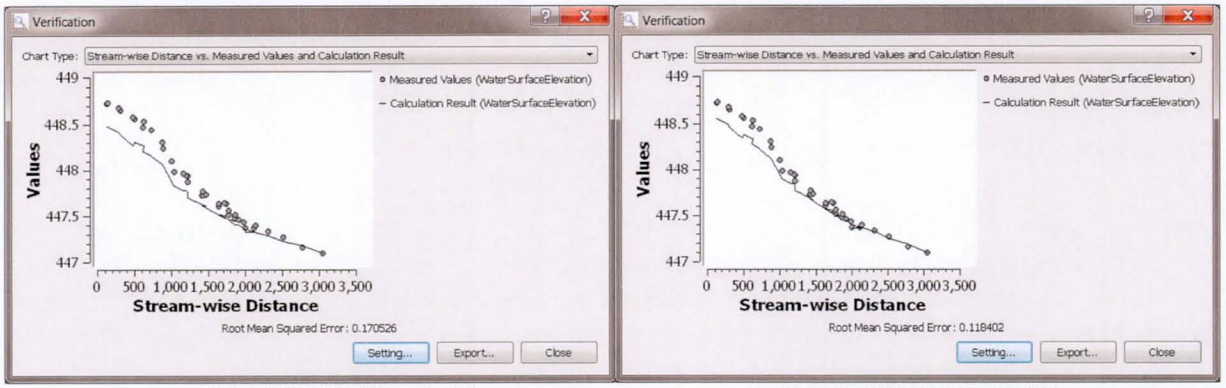


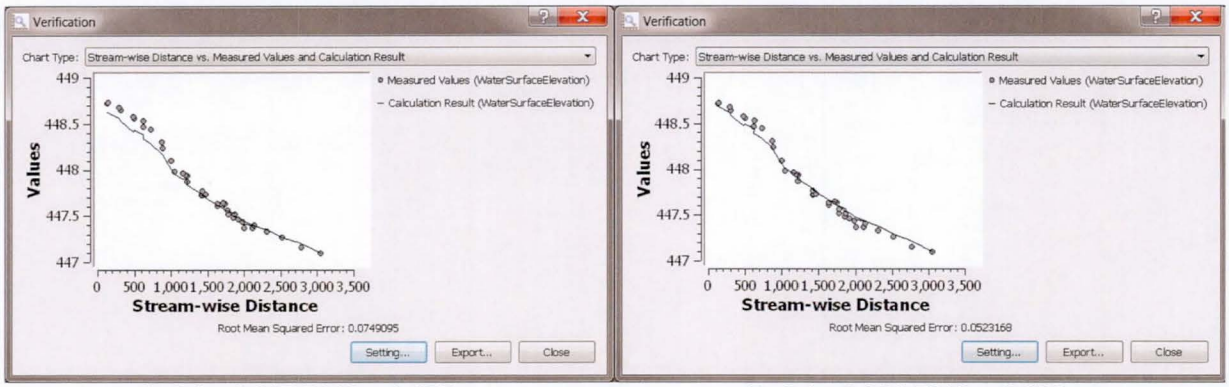
Figure 22. Example verification plot showing the measured and simulated water-surface elevation. Note that the simulated water-surface elevation is too high compared to the measured data at the downstream end of the modeled reach. The Root Mean Squared Error is 0.1.

- 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 that contains the RMS error for a range of simulated roughness values. Looking at these results, it appears that a Cd of about 0.008 yields the lowest RMS error. However, none of the simulations are a good match at all locations.



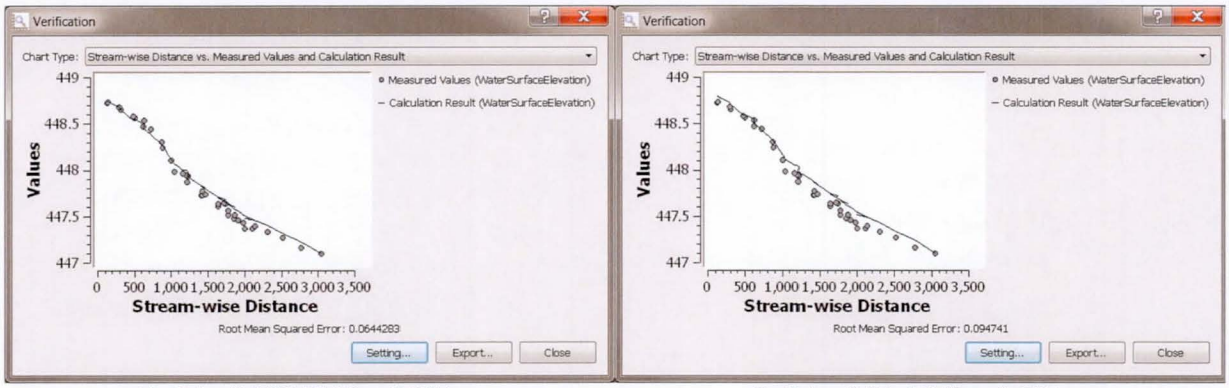
A. Cd = 0.005 RMS = 0.170

B. Cd =0 .006 RMS = 0.118



C. Cd = 0.007 RMS = 0.075

D. Cd = 0.008 RMS = 0.052



E. Cd = 0.009 RMS = 0.064

F. Cd = 0.01 RMS = 0.095

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 0.007 and 0.009.

- 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.

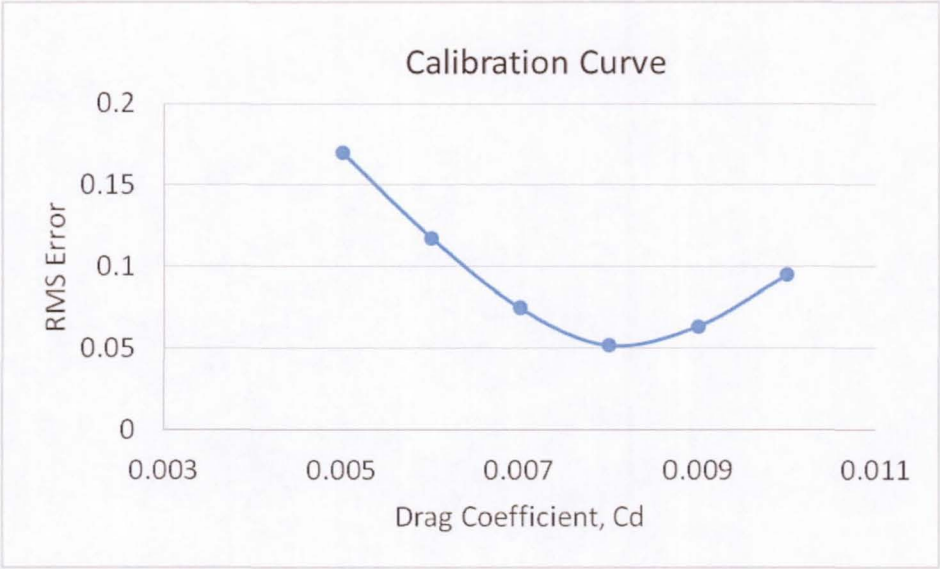


Figure 24. Calibration curve showing the relationship between roughness and the RMS error for the simulated water-surface elevations.

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 0.007 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 0.013.
    - Make sure that both polygons are displayed in the **Object Browser** under Geographic Data | Roughness.

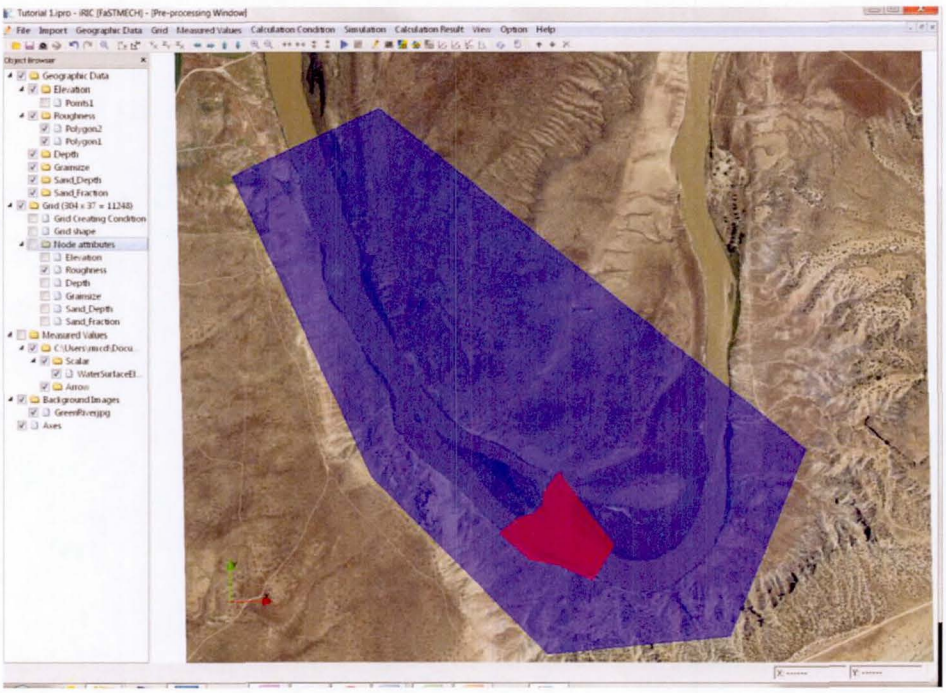


Figure 25. Example of the polygon mapping

- 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 inthe **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.

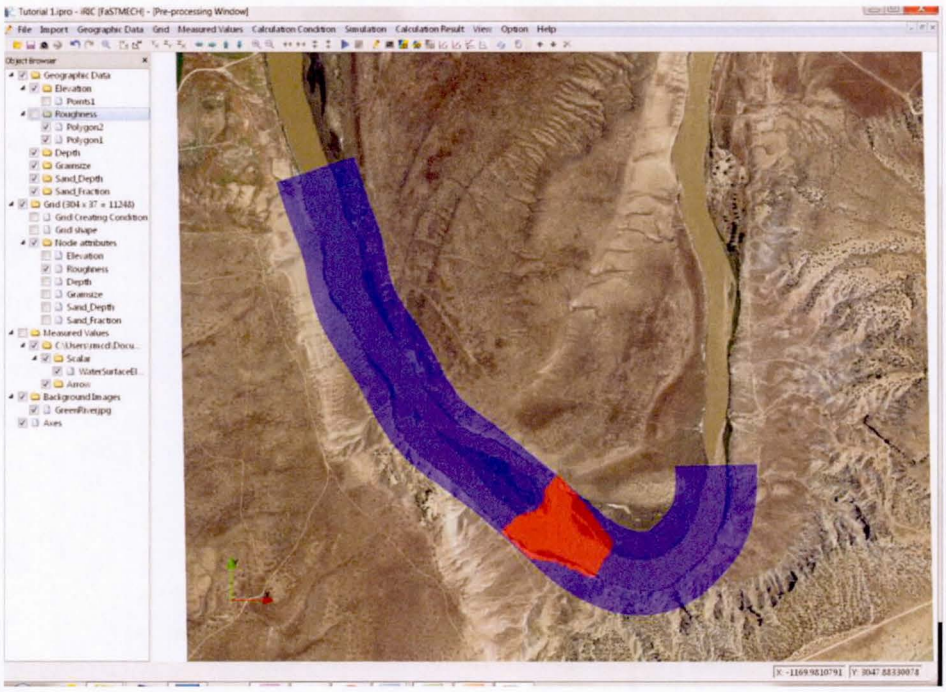


Figure 26. Final roughness attribute mapping

- 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.

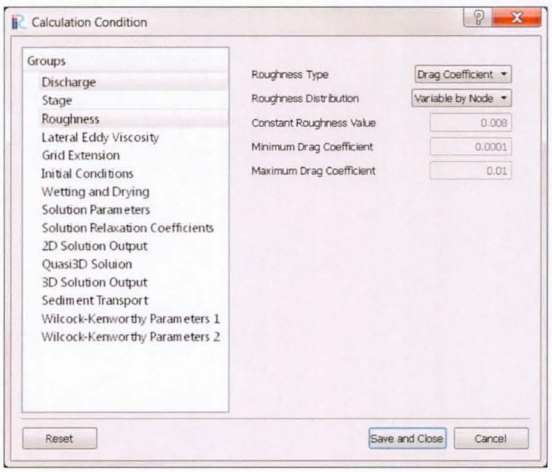


Figure 27. Calculation condition, roughness dialog

- 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).

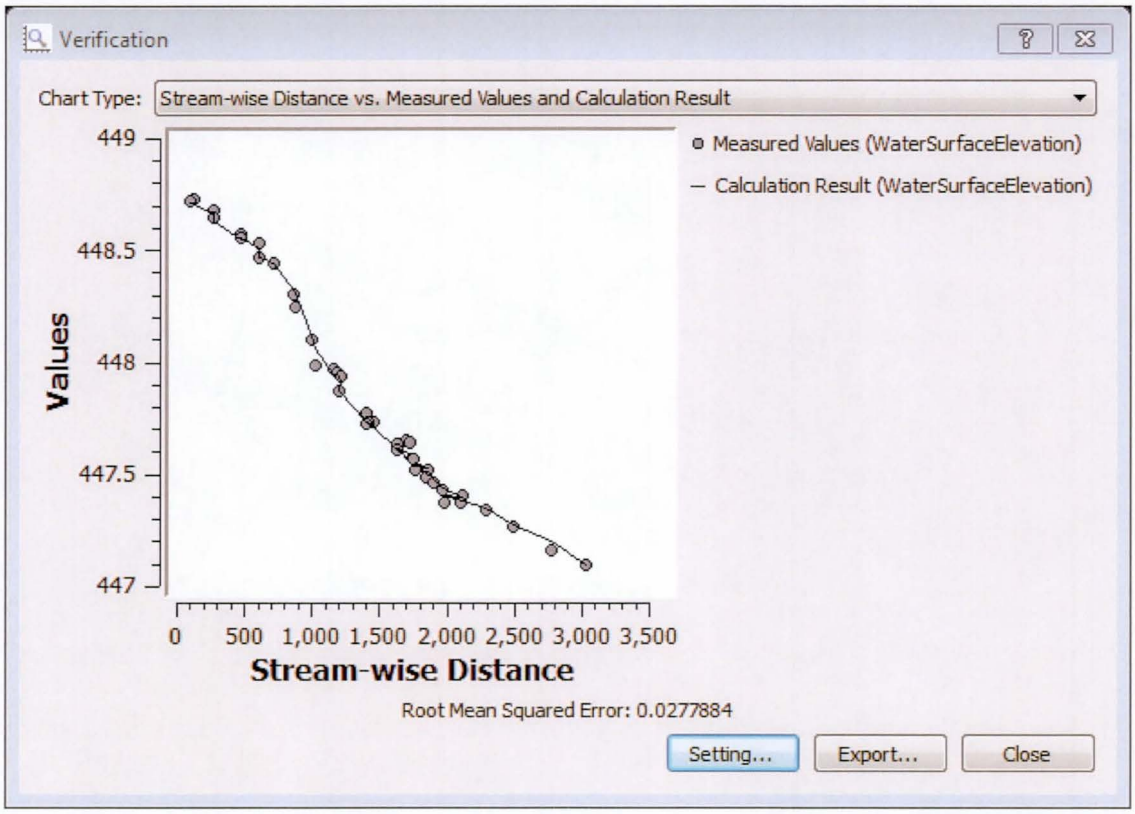


Figure 28. Comparison of the measured and simulated water-surface elevation using a spatially variable drag coefficient. RMS error is 0.028.

Quasi-3D Model and 3D Graphics

In this section the simple vertical structure of shear flows and the generation of secondary flows using the quasi-3D sub-model available in FaSTMECH is briefly explored. A more in-depth discussion of this topic is available in Tutorial 3. In addition, an introduction to the 3D graphics capabilities of iRIC is provided. The model previously developed is used and the qausi-3d model and 3d solution is simply turned on. A nice feature of this function is that the quasi-3d solution is computed upon completion of the 2d solution, such that this additional step requires a minor amount of additional computation time. The quasi-3d model offers the option of using the channel curvature or channel curvature + streamline curvature for calculating secondary flows. We'll use both options in the following sections. First, secondary flow expressed as the helix strength, which is defined as the angular difference between the near-bed flow and the surface flow and is a simple measure of the presence and strength of secondary flows is explored. Second, vertical sections of velocity vectors are plotted to illustrate the rotation of vectors in the vertical due to secondary flow. Finally, cross-sections are examined through the reach to visualize the vertical structure of the flow.

Visualize Two- and Three-Dimensional Solution Without and With Streamline Curvature

- Return to the pre-processing module by selecting View ► Pre-processing Window.
- Select Calculation Condition ► Setting and edit the following tabs in the Calculation Condition Dialog as in Figure 29 and 30. In this first case only channel curvature is used. 3D output including the calculation of the helix strength and the 3D scalar and vector quantities are added through the 3D Solution Output tab.

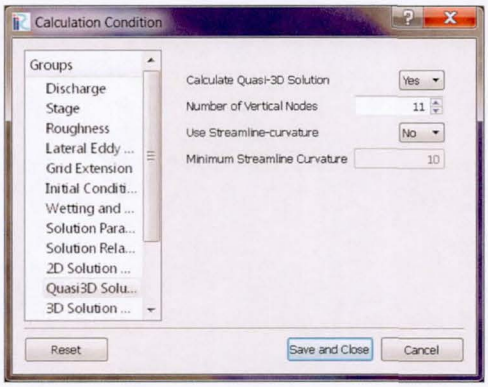


Figure 29. Quasi-3D Solution tab of the Calculation Conditions.

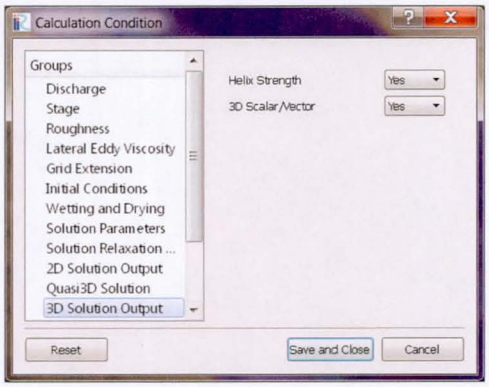


Figure 30. 3D Solution Output tab of the Calculation Conditions

- Re-run the model to obtain the new solution. Notice that in the toolbar the 3D ( ) option is now available.
- Before exploring the 3D graphing options, look at the Helix Strength 2D solution result as in Figure 31 below.
    - Set the min and max Helix strength to -25 and 25 respectively. Also, set the Display Property setting to Contour.

Re-run the model with the Use Streamline-curvature option set in the Calculation Conditions and the Minimum Streamline Curvature set to 10m. The resulting helix strength solution is plotted in Figure 32.

- The helix strength and streamlines are plotted in Figure 33. Note the correlation between curvature of the streamlines and the helix strength.

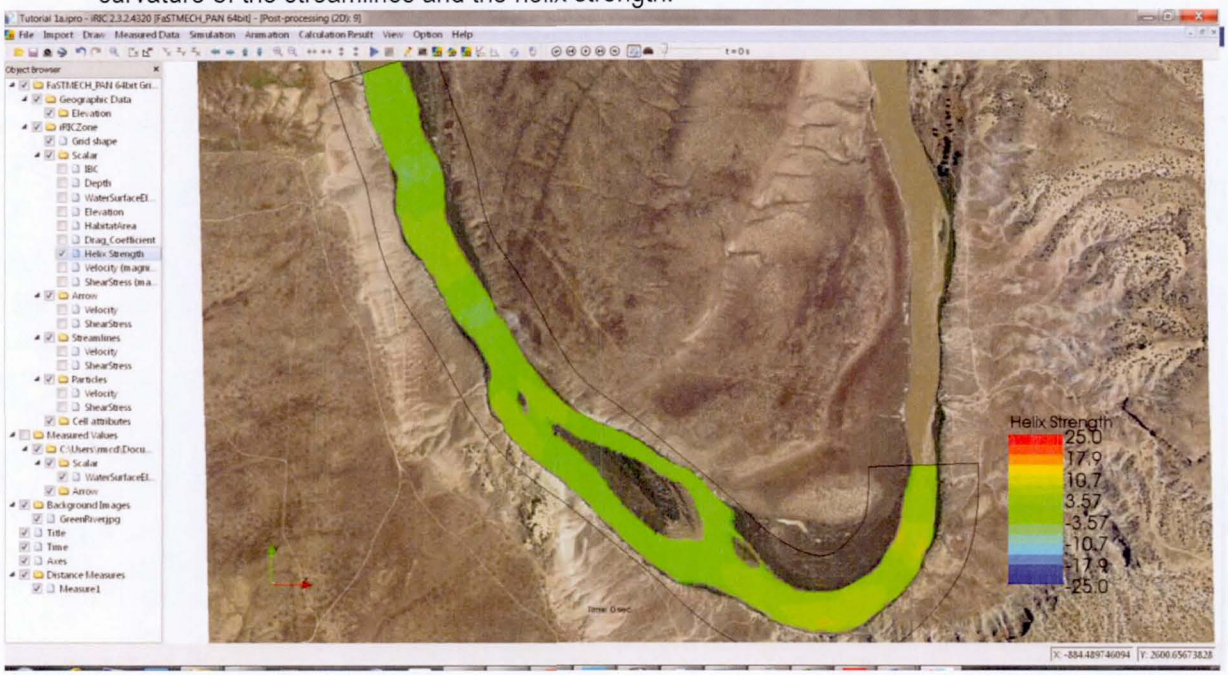


Figure 31. Helix strength calculated using only the grid curvature

Figure 32. Helix strength calculated using streamline curvature

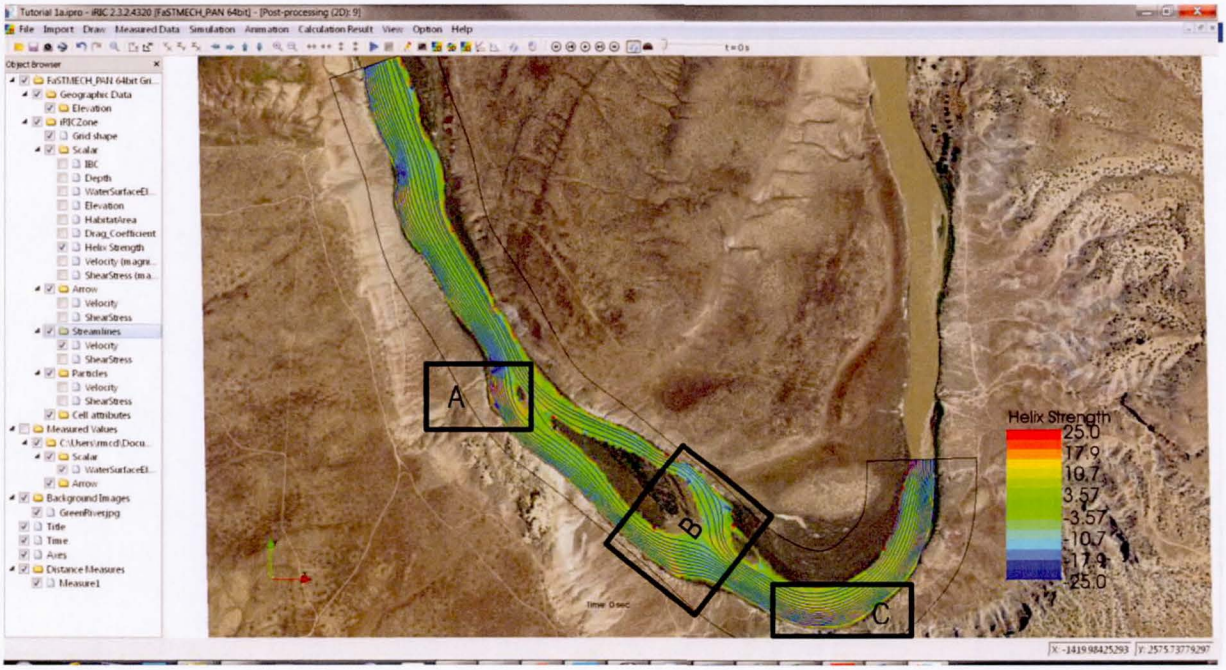
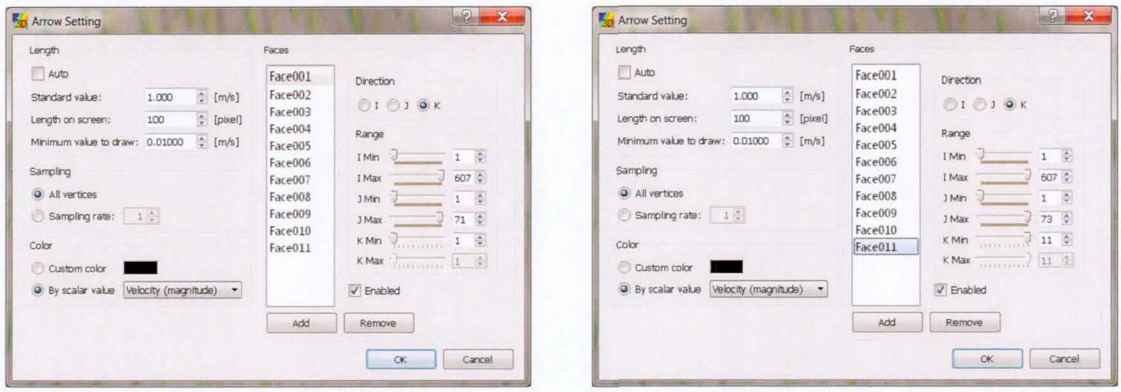


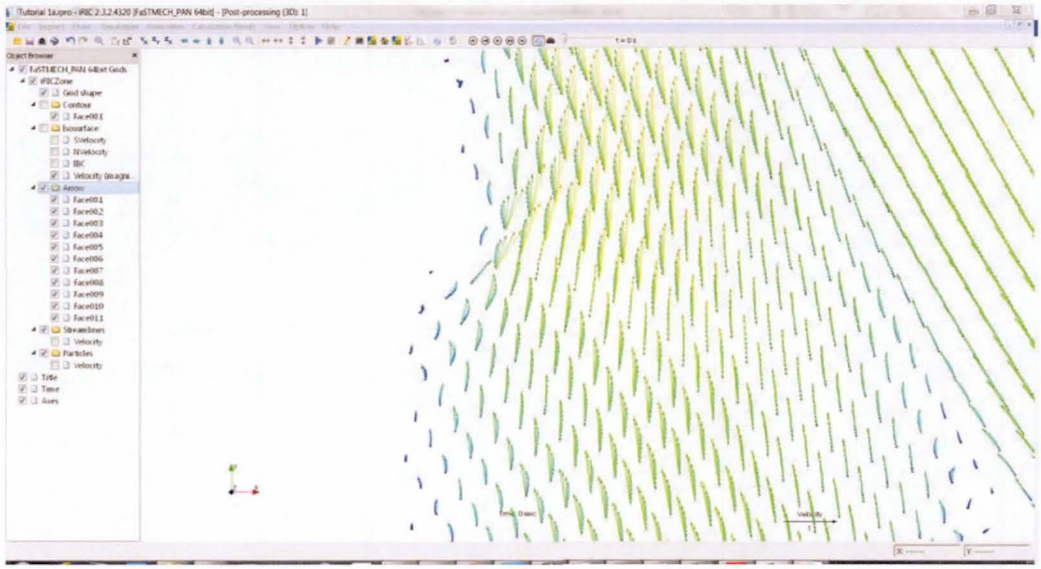
Figure 33. Helix strength and streamlines originating from the upstream boundary. Notice the correlation between curvature in the streamlines and the helix strength. The boxes are the locations of the following figures A) Figure 34, B) Figure 35) and C) Figure 36.

Vertical Structure Continued

1. To create an image as shown in Figure 34:
   - In the 3D Post Processing window right-click *FaSTMECH Grids | iRICZone | Arrow* in the Object Browser and select the Property pop-up menu. Set up the Arrow Setting dialog as shown in Figure 34 (A and B). Where each Face added (Face001, Face002..., Face011) incrementally increases the value of the K index (from 1 – 11). Also, set up the Length, Sampling, and Color fields as shown.
   - By zooming into the downstream end of the main (river-left) channel around the island, where the flow has some of the highest curvature, you should be able to reproduce the graphics as shown in Figure 34 (C).The velocity from every vertical node is shown illustrating the secondary flow where the surface velocities are oriented towards the outside of the bend and the near bed velocities are oriented towards the inside of the bend.



A B



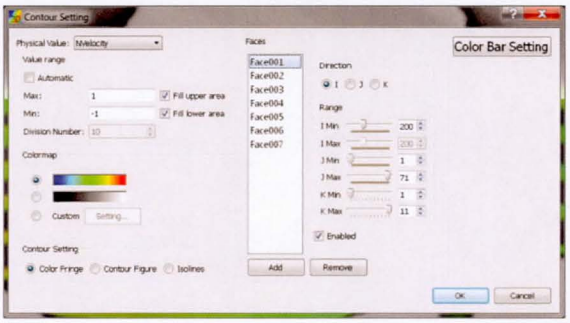
C

Figure 34. The Arrow Setting dialog. Note that in (A) the K index is set to 1 and in (B) the K index is set to 11. The 3-dimensional vector field where the velocities at the surface are highest and the near bed velocities are lowest. Notice how the orientation of the velocities changes as the streamline curvature changes

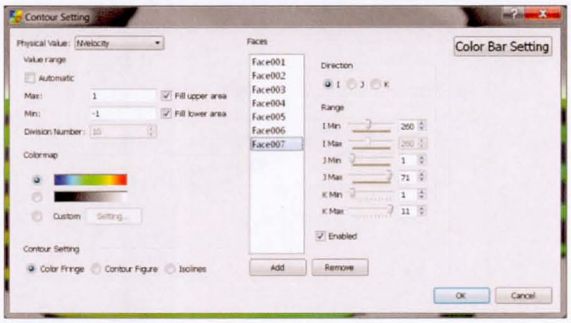
2. To Create an image as shown in Figure 35:
   - In the 3D Post Processing window right-click ***FaSTMECH Grids | iRICZone | Contour*** in the Object Browser and select the Property pop-up menu. Set up the Contour Setting dialog as shown in Figure 35 (A & B). Note that the NVelocity is the cross-stream velocity relative to the grid coordinate system. When the cuvature of the coordinate system (Grid) follows the channel curvature nicely it's easy to see the secondary flow in the plots below (See Figure 36). Seven Faces are added incrementally with the value of the I-index equal to 200, 210, 220, 230, 240, 250, 260.
   - Additionally an iso-surface of the velocity magnitude (0.01 m/s) is added to provide a sense of the channel bottom (Figure 35C)
   - To add some vertical exaggeration so that it's easier to see the cross-sections select View->Z Direction Scale from the Menu and enter 5 in the Z-direction scale. The easiest way to orient

the channel is to first select the ZY.button and then using the Ctrl Key and Right mouse button – drag the mouse from top to bottom. In this case flow is from top to bottom. Convince yourself of this with the knowledge that the coordinate system is right-handed.

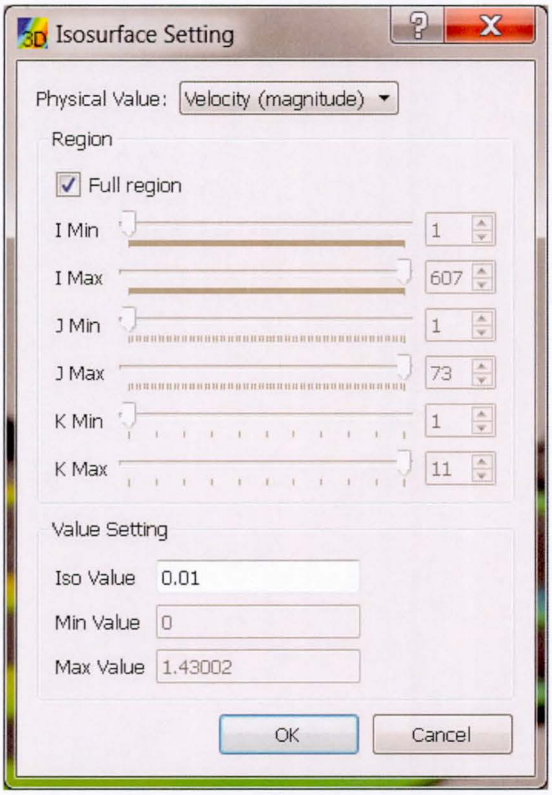
- Figure 36 is created by adding additional contour sections with the I-index values equal to 125, 130, 135, and 140. The max and min NVelocity values are 0.1 to -0.1.



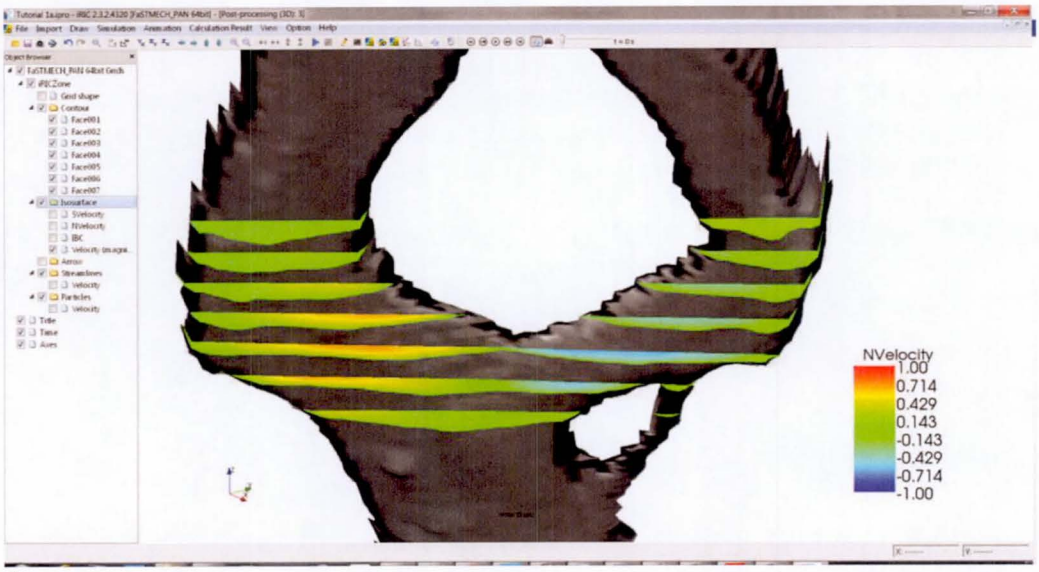
A



B



C



D

Figure 35. The cross-stream velocity is shown for a number of cross-sections through the reach. NVelocity is the magnitude of the velocity oriented perpendicular to the stream-wise direction. When the grid follows the channel curvature well, NVelocity provides a good representation of the secondary flow. Here the grid does not follow the curvature of each channel around the island, so we only capture some of the secondary flow with these plots.

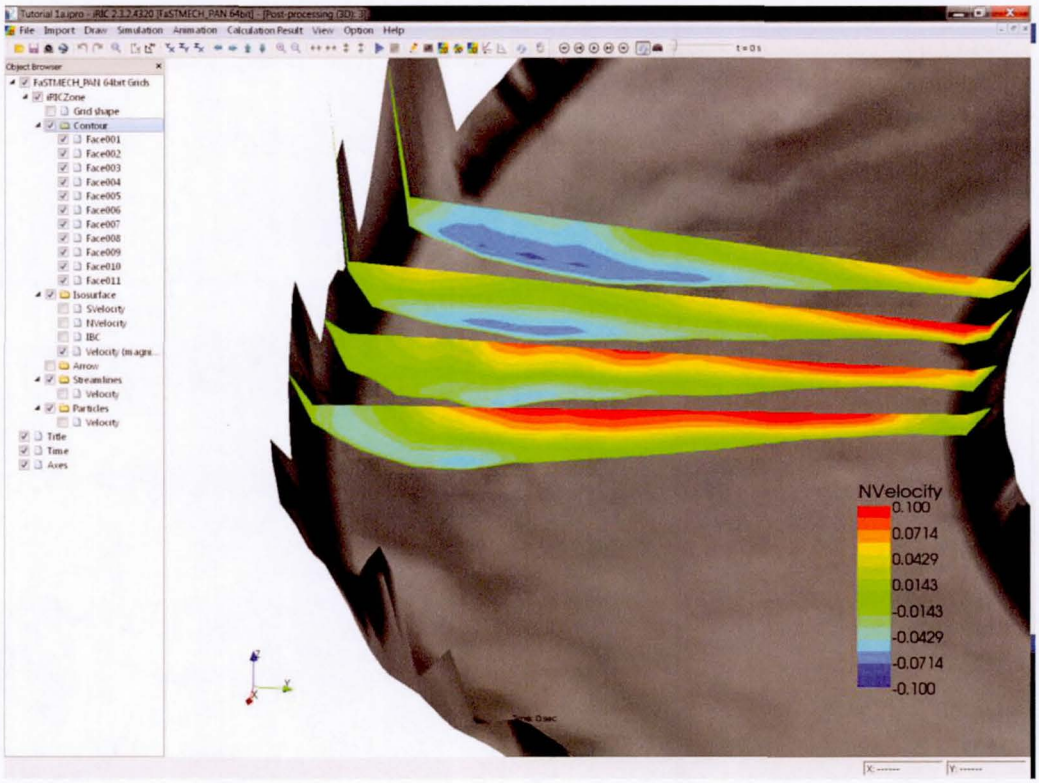


Figure 36. *Vertical Profiles at cross-sections corresponding to I-locations of 125, 130, 135, and 140.*

- For a more in-depth discussion of secondary flows and its effect on sediment-transport, please see Tutorials 3 and 5 respectively.

iRIC Software

Changing River Science

FaSTMECH Tutorial 9

Flood reconstruction and peak discharge estimation

Satellite image of river flow and flood damage on September 14 following the peak discharge on September 13.

Introduction

In September, 2013, a slow-moving, low-pressure system over the Great Basin (southwest of Colorado) pulled a strong plume of monsoonal tropical moisture from the Pacific Ocean off western Mexico and as time progressed, the circulation pattern brought additional moisture from the Gulf of Mexico on easterly and southeasterly flow. These easterly and southeasterly flows resulted in upslope flows in the Colorado Front Range and propelled the moisture against the foothills. The presence of a stalled cold front assisted with additional lift, resulting in rainfall over a very large area[1] (roughly from Colorado Springs north to Fort Collins) (Figure 1). Rainfall amounts exceeded 38.1 cm (15 in) in some locations, with the majority falling in a 36-hour period from the afternoon of Sept. 11 through early morning Sept. 13. Boulder County, where the CODWR gage on St. Vrain Cr.at Lyons is located, received over 22.9 cm (9 in) on September 12 and cumulatively up to 43.2 cm (17 in) for the week[2]. Boulder County has an average annual precipitation of 52.6 cm (20.7 in)[3].

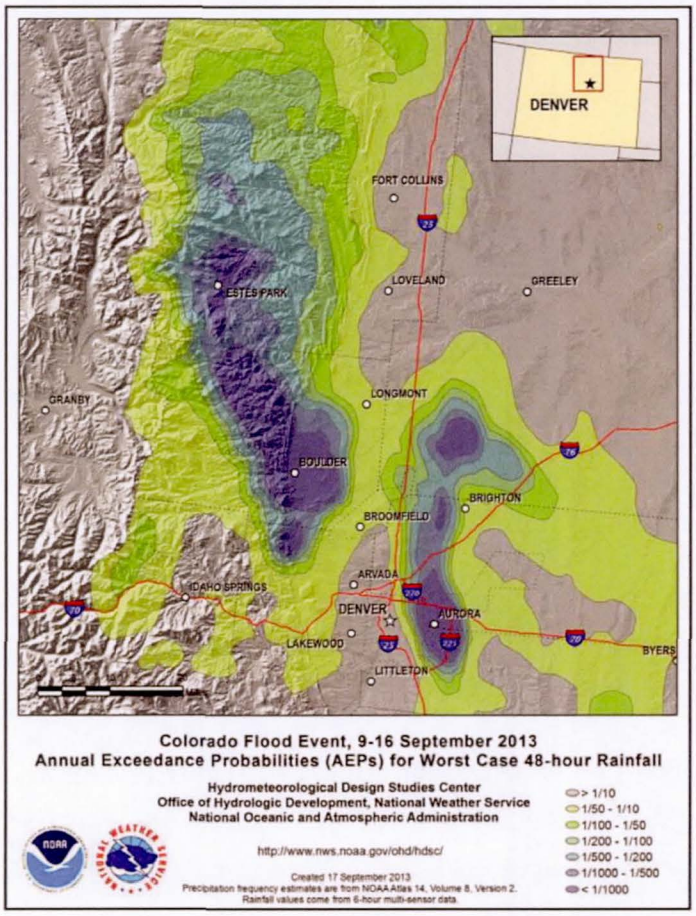


Figure 1

[1] "Severe Flooding on the Colorado Front Range, September 2013": A Preliminary Assessment from the CIRES Western Water Assessment at the University of Colorado, NOAA ESRL Physical Science Division, and the CSU Colorado Climate Center, released September 25, 2013 Lead author: Jeff Lukas CIRES/WWA; lukas@colorado.edu

[2] Andy Freeman, "Flood-Ravaged Boulder, Colo., Sets Annual Rainfall Record" Climate Central published on September 16, 2013 http://www.climatecentral.org/news/flood-ravaged-boulder-colorado-sets-annual-rainfall-record-16481

[3] "Average Annual Rainfall for Colorado" Current Results Research News and Science Facts http://www.currentresults.com/Weather/Colorado/average-yearly-precipitation.php#b

The flood peak on St. Vrain Creek at Lyons occurred sometime in the evening of Sept. 12 or early morning of Sept. 13. The State gaging station was destroyed by the flood sometime after 21:15 hrs when it last transmitted data (Figure 2). Data collected by the gage in the final hours should be considered unreliable as undermining of the road and gagehouse structure was likely occurring and would have affected gage recordings. Within the town of Lyons, many homes and businesses in the central part of the floodway along the Hwy 36/66 corridor were inundated with water, and flood damage to infrastructure was extensive.

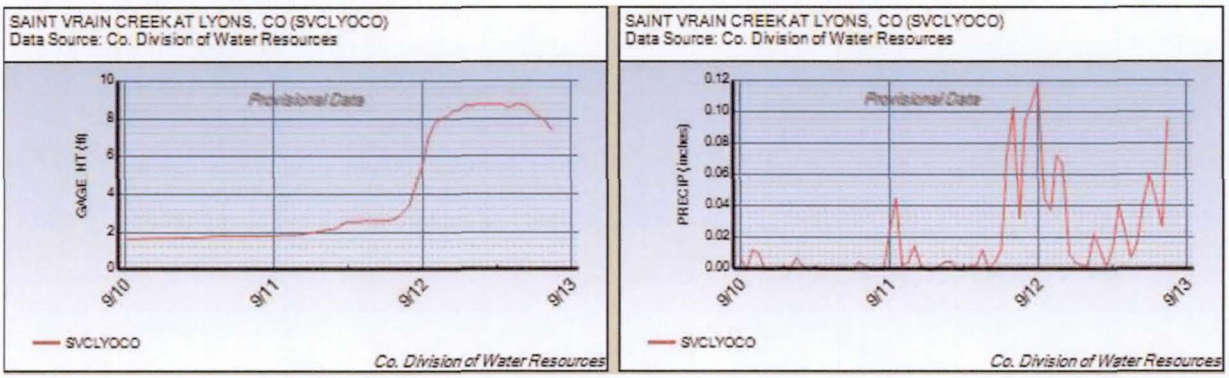


Figure 2 Colorado Division of Water Resources gage record for Saint Vrain Creek at Lyons, CO (http://www.dwr.state.co.us/SurfaceWater/data/detail_graph.aspx?ID=SVCLYOCO&MTYPE=DISCHRG)

Following the flood, USGS personnel surveyed High Water Marks in the region of interest, and shortly following the flood LiDAR was flown. You are provided with Topography as measured by airborne LiDAR following the flood and the measured HWMs and your job is to determine the peak discharge during the flood. In this tutorial you will perform the following tasks:

1. Import elevation, measured high water marks, and background imagery
2. Process topography to account for flow under the roads
3. Build a grid
4. Determine peak discharge:
    a. For a given single roughness value.
    b. For a range of roughness values.

Note: There are two general ways the user instructs iRIC to perform an operation: 1) through the menu at the top of the application, or through the Object Browser on the left of the application. We use two formats to indicate a menu operation or an Object Browser operation. For Example:

Menu operation: **File -> Import -> Geographic Data -> Elevation...** The menu operation is show in Bold font, and each successive menu is separated by an arrow (->).

Object Browser: ***Geographic Data | Elevation | Points1*** The object browser is essentially a tree view with set of branches and sub-branches. Each sub branch is separated by a (|) sybol, and the font is Bold + Italic.

Task 1: Import Data

1. **Launch iRIC:** Start iRIC and select FaSTMECH as your solver (Figure 3A).

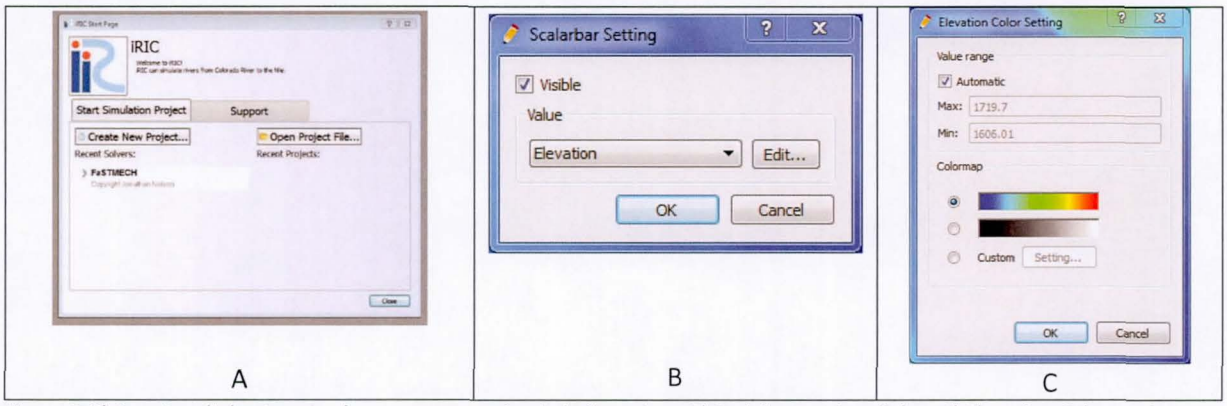


Figure 3 A) Opening dialog to set solver or open previously built project. B) Scalebar setting dialog. C) Elevation Color setting dialog.

2. **Import topography:** Select from the menu, **Import->Geographic Data -> Elevation**. In the folder containing the data for this tutorial select the file StVrain_1_0m_477700_4452000_shifted.tpo. When the file is selected the Filter Setting dialog will appear which allows you to simply decimate your data set. For example setting the filter value to 1 imports every point, setting the filter value to 2 imports every other point, etc. This data is a subset of available LiDAR data for the reach (Figure 4A). It has been decimated to 1 meter resolution, and shifted by 477700 meters in the X-direction and 4452000 meters in the Y-direction.
3. **Add a scalebar:** Select **Geographic Data -> Set Up Scalebar** from the menu (Figure 3B). In the resulting dialog make sure the dropdown list is set to Elevation and select OK. You can always change the color setting by selecting **Geographic Data -> Color Setting -> Elevation**. For now keep the default setting by selecting OK (Figure 3C). Your project should now look like Figure 4A.
4. **Import background image:** From the menu select **Import -> Background Image** and select PostFloodImage_Shifted.jpg.
5. **Import the measured high water marks (HWMs):** Select **Import -> Measured Values**, and select the file HWMarks3_SAC_RIGHTBANK.csv. By default, iRIC has placed a data legend for the HWMs over the Elevation data legend. Move the Elevation data legend by first selecting ***Geographic Data | Elevation*** in the Object Browser tree. Next, click your mouse on the legend and drag it to a new location.
6. **Increase point size of HWMs:** Turn off the elevation point set by de-selecting the ***Geographic Data | Elevation*** checkbox in the Object Browser. To make the HWMs more visible, first select ***Measured Values | File path and name | Scalar*** in the Object Browser with your mouse. Then right-click on the same path in the Object Browser and select Property from the pop-up menu. In the resulting dialog set the Point Size to 7 and select the OK button. Your project should now look like Figure 4B.

Now you have imported all the data necessary to begin creating a flow model and in the process you have learned some of the basic functionality of the iRIC application. You can import different data sets,

add a data legend for the different data sets, and change the graphic properties and view of imported data.

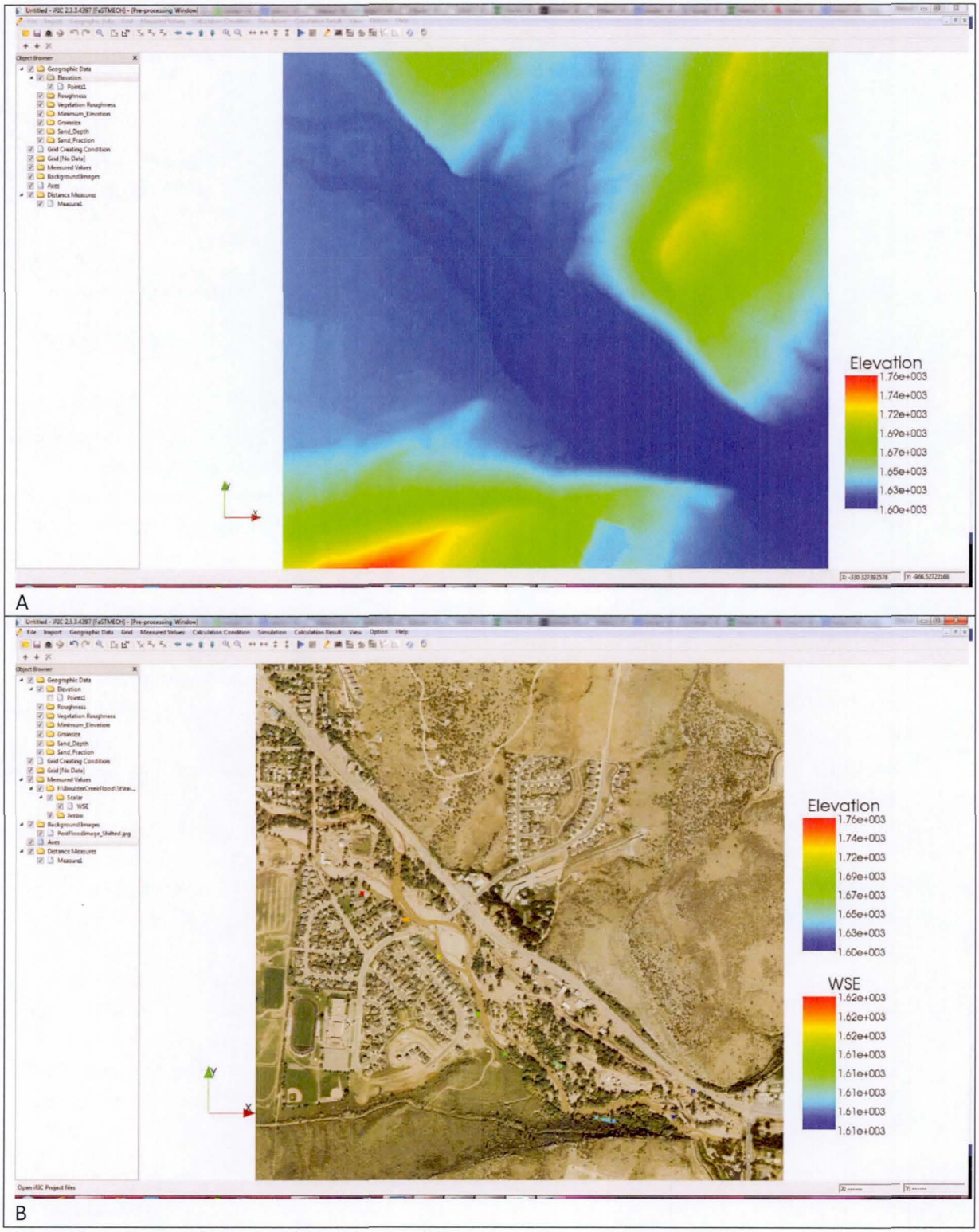


Figure 4 A) graphic view of imported LiDAR data set and B) Background image and measured HWMs.

Task 2: Preprocess the topography

1. **<u>Set the color map of the topography to better visualize the elevations of interest.</u>**
    a. Turn off the measured data by deselecting Measured Values in the Object Browser and then turn the Elevation data back on. You will edit the elevation data to simplify the view. In particular, we want to delete those points higher than are necessary to model this reach, and second we will change the color setting.
    b. Select ***Geographic Data | Elevation | Points1*** in the Object Browser. Doing so enables the tool bar for editing the point set data. Notice that if you select some other branch in the Object Browser, the toolbar is no longer visible. In iRIC, whenever you want to operate on a specific data set, you should first select that data set in the Object Browser, and any available tools for interacting with that data will be visible in the toolbar near the top of the application. The Select Point button (4th from left) allows you to select a single point, shift-select multiple points, or drag-select a rectangle of points. To see how this works, drag-select a rectangle of points. The point set is very dense, so unless you zoom into a specific area, it will be hard to see the points that are selected. That is not important for now. Following the selection, right click on the graphic window and select **Edit Selected Points** from the pop-up menu. If you did not effectively select a point then the **Edit Selected Points** option will be greyed out. If so, try again. In the Edit Points dialog notice that you will see the number of points selected and the minimum and maximum value of those points. This provides a quick and easy way to probe the data and we will use this technique again later in the tutorial. Also notice that X-location and Y-location are blank. If you had selected a single point then you could change its x,y position if desired. When multiple points are selected only the Value is available to change. In this case we are not interested in editing the data; we only use this tool as a quick probe of the values, specifically the minimum and maximum values. Select the Cancel button. If you select the OK button then all the selected points will be set the Value specified in the Edit Points Dialog. Don't do that here, again we are just probing the points to determine their values. If you do this by accident then simply select the Undo button ( ) in the main toolbar at the top of the application.
    c. We are going to delete points greater than a defined value. This will greatly reduce the number of points in the data set to speed up some of the operations we will perform later and also provide more color resolution to the data of interest in the channel. Using the select point tool described above select a point or region of points somewhere in the light blue range to determine the elevation value (s) specifically. You should find values somewhere near 1634. Zoom out so the entire elevation data set is visible, then using the select point tool, drag-select then entire data set. Right click on the graphics window and select Delete Points Greater Than Value, and in the resulting dialog enter 1634 and select the OK button. The results should look like Figure 5A.
    d. To change the color map of the data first make sure you have selected ***Geographic Data | Elevation | Points1***, since that is the data set we will be operating on. Next select **Geographic Data -> Color Setting -> Elevation** from the menu. In the Elevation Color

Setting dialog, de-select the Automatic checkbox and then enter 1634 for the Max value. In the Colormap subbox, choose the Custom button and then select the Setting button and set the Type to Three Colors, the Medium Value color to white (click the color field to get the Select Color dialog) and then enter 1620 for the Value. The resulting data and the three dialogs described in this operation are shown in Figure 5B.

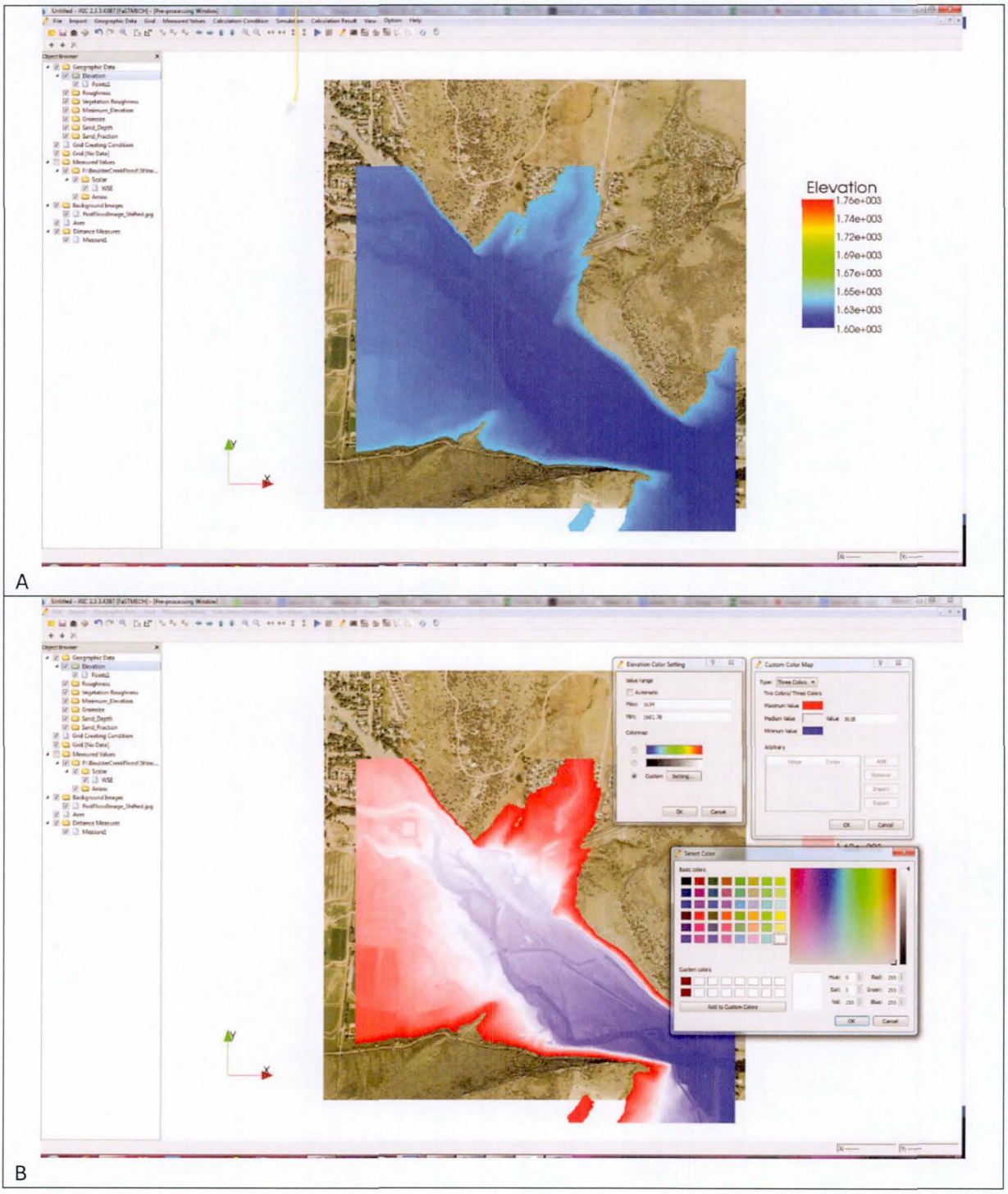


Figure 5 A) Elevation data below 1634 meters. B) Modified Color map of elevation data. Insets are in clockwise direction 1) Elevation Color Setting dialog, 2) Custom Color Map dialog, and 3) Select Color dialog.

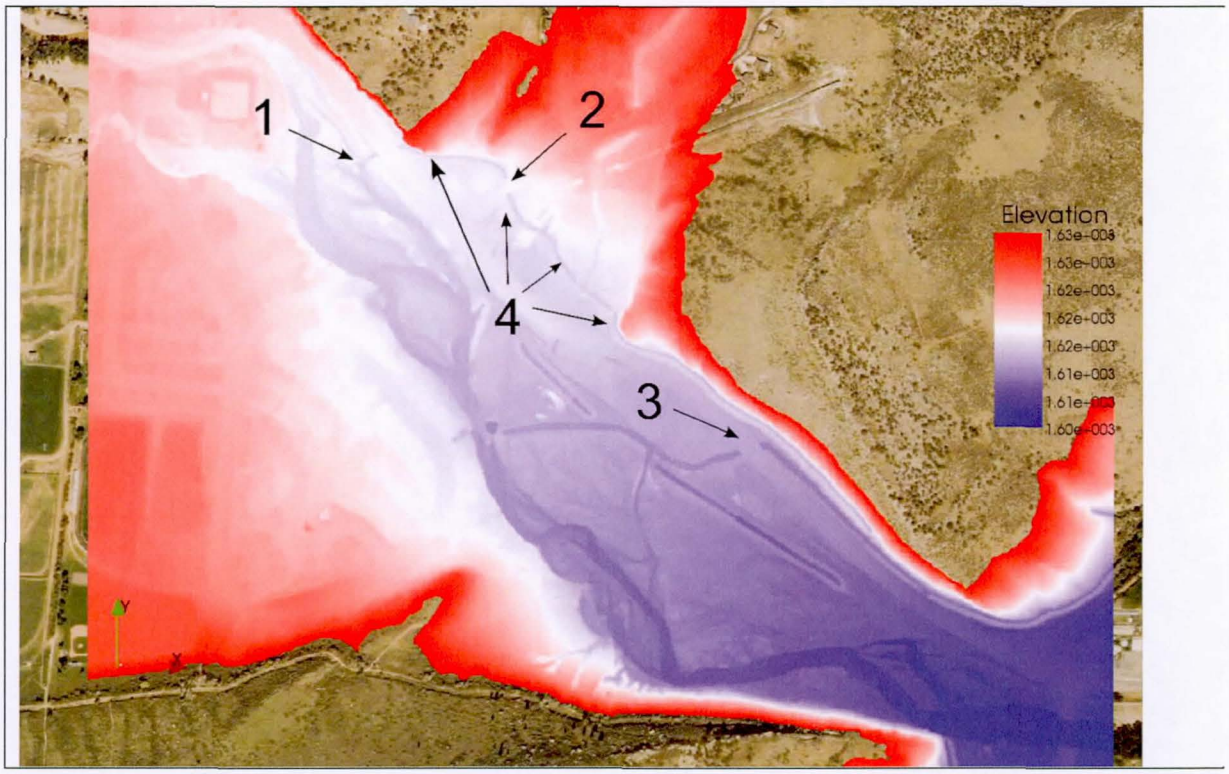


Figure 6 Location of flow diversions under roads and irrigation ditch which require some modification of the existing topography

2. **<u>Modify the topography to account for flow under the roads.</u>** In iRIC you can modify any of the imported Geographic Data such as elevation by drawing polygons over the point set and defining an elevation value to represent the topography. Then, when the Geographic Data is mapped to the grid you will create later, wherever a polygon exists it will over-ride the measured values below it. We will add a series of polygons to approximate the location of the channel under the roads.
    a. In Figure 6, four locations have been identified that require some modification to the elevation data set. At 1, 2, and 3, a road crosses over the channel, and at 4, there are constrictions in an irrigation ditch likely due to vegetation obscuring the ditch in the LiDAR topography. In the following steps, you will add polygons on top of the elevation data. Each polygon has a user defined value that will override the underlying topography and approximate the elevation of channel bottom underneath the road. In this way, modifications to the topography can be imposed without losing the original measured data. If you don't like the modification, just delete the polygon.
    b. To modify the height of the topography at a road crossing, follow the steps below. You will add a polygon yourself for the road crossing identified by the number 1 in Figure 6, and for the rest of the road crossings and modifications to the irrigation canal you will import the polygons which have been built and saved in a shapefile (.shp) for you.
        i. Adjust the color setting to add more detail to the topography so it's easier to define the location of the road crossing #1. We learned how to adjust the color

setting of legend in section 1d above. Set the color setting to a minimum of 1615 and maximum of 1617.

ii. Set the size of the elevation point set to 7 which will make the data easier to see when zoomed in. To do this select ***Geographic Data | Elevation | Points1*** and the right-click and select **Property** in the pop-up menu. Then set the point size to 7 and select the OK button.

iii. Probe the elevation values on either side of the road to determine the best single value to assign to the elevation of the polygon you will create next. You learned how to probe values in section 1b above.

iv. To create a polygon that approximates the spatial location of the channel below the road be sure to first select the ***Geographic Data | Elevation*** in the Object Browser with a left-click, and then right click the same and select **add -> Polygon** in the pop-up menu. Draw a polygon that crosses the road and connect to the topographically low channel on either side. When finished select Enter on the keyboard to finish drawing the polygon and in the resulting Edit Elevation value dialog, enter the elevation for the polygon based on the values you probed in section iii above (Figure 7A).

v. Import polygons created for you to improve the elevation of the remaining two diversions and the irrigation canal. In the menu select **Import -> Geographic Data -> Elevation** and browse to the IrrigationCanal2.shp file. Make sure that in the Polygon Import Setting dialog you have selected Name from the drop-down menu for the Name field, and you have selected Value in the drop-down menu for the Value field (Figure 7B).

vi. Turn off the ***Elevation | Points1*** in the Object Browser and the result of the drawn and imported polygons should look something like Figure 7C.

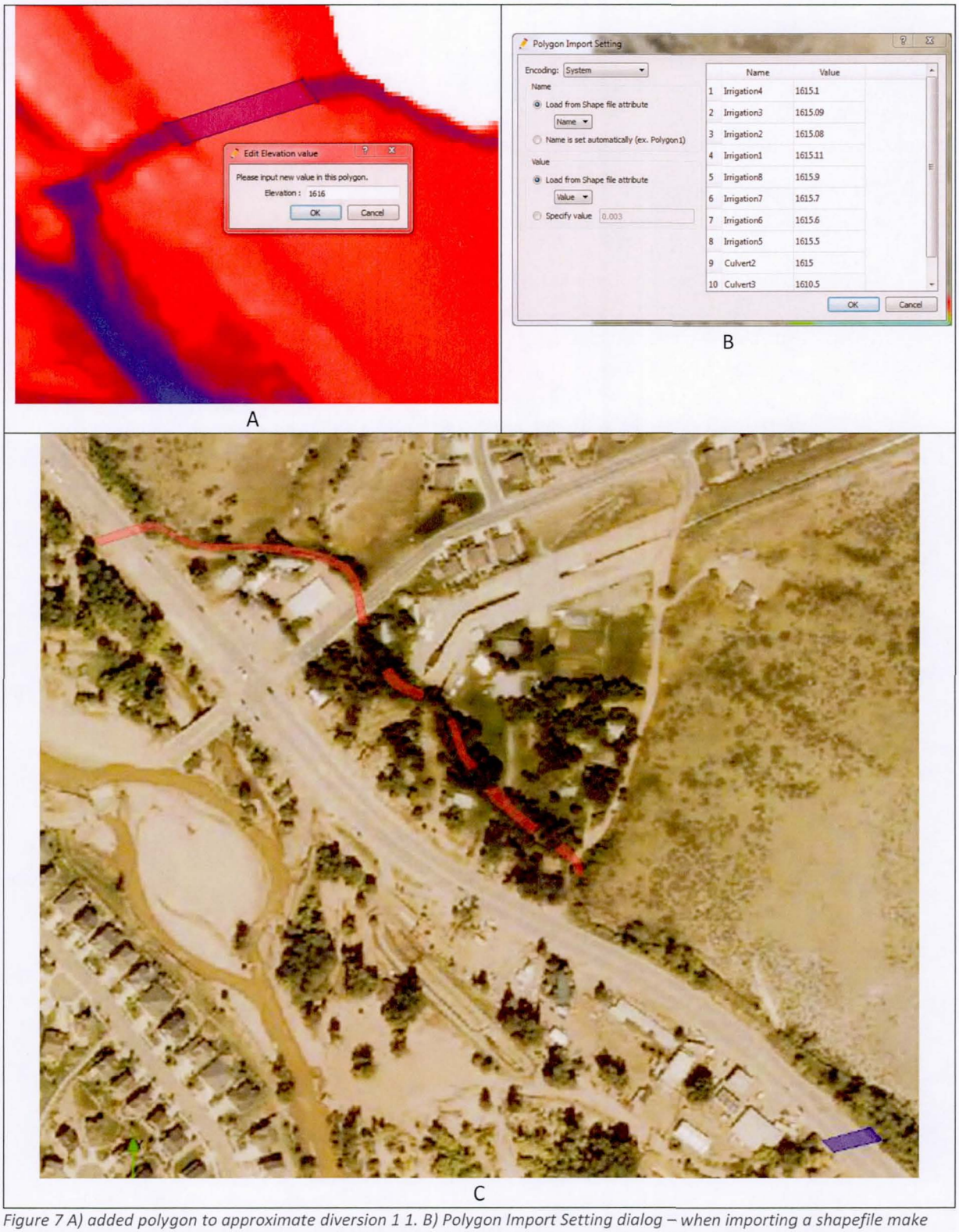


Figure 7 A) added polygon to approximate diversion 1 1. B) Polygon Import Setting dialog – when importing a shapefile make sure you have selected the correct values in the drop-down list for the Name and Value fields. C) The complete set of polygons to enhance the topographic representation of diversions and irrigation canal.

Task 3: Create Grid

In this section we'll create a grid and map or interpolate the measured topography values, and the overriding user defined polygon values, to the grid. It might be easiest to read this entire section before you begin. We will be performing many edits to the grid during this section, to improve the user experience we will turn off some of the automated features of iRIC.

1. **Set mapping attributes:** Select **Grid -> Attributes Mapping -> Setting** from the menu and in the resulting Grid Attribute Mapping Setting dialog, select manual in the Execute mapping section of the dialog. This will prevent iRIC from mapping data to the grid following every edit to the grid.
2. **Considerations:** In step 3 below you will select a grid generator and draw a centerline, but before doing this it's important to consider the following points.
    a. Note that the centerline is drawn from upstream to downstream, in this case from upper left to lower right.
    b. In Figure 8, a simple grid is shown for context only - You won't see this as you draw your centerline. Each of the small black squares on the centerline are the points used to define its path. In this case 6 points are defined. Usually only a few points are desired. You can always add or delete centerline points later if necessary.
    c. The downstream boundary is located such that the furthest downstream measured water-surface elevation is at the boundary because we will use its value as a boundary condition for our simulation.
    d. You will want to adjust the curvature of your grid such that the boundaries are as perpendicular as possible to the main flow direction. See the insets in Figure 8. Here we plot the vectors from a simulation for illustration, but it is generally easy enough to guess the flow direction. It's possible that after an initial solution you may want to go back and adjust the location of the boundaries.
    e. Additionally, it's important to place the upstream boundary such that there is not a narrowing of the channel immediately downstream. The flow direction at the upstream boundary is set perpendicular to the boundary by default (note it is possible to adjust the flow direction at the boundary, but for now it's best and easiest to assume a perpendicular flow direction). If the channel decreases in width immediately downstream, then the flow will be directed into the bank. Looking at the upstream boundary inset in Figure 8, if the boundary was placed further downstream then you might imagine the flow at the upstream boundary would be directed into the bank downstream; the common result is for the model to crash.
3. **Select grid generator and draw centerline:** Select the grid generator by selecting **Grid -> Select Algorithm to Create Grid** in the menu and in the resulting dialog select "Create grid from polygonal line and width" in the list of Algorithms and select the OK button. An information dialog will appear. It provides a quick description on how to create the grid using this tool. Select the OK button. Using your left mouse button define the centerline for the grid. It should look something like that in Figure 8. It's not important to be precise at this point. As you will see later it's important to get the boundaries of the grid perpendicular to the main flow direction and that is easily done once you have a basic grid. The aim here is to draw your centerline in a location that is similar to that in Figure 8. The centerline is completed by selecting the Enter button on your keyboard.

4. **Create simple grid to help placement of grid and boundary conditions:** Create simple grid by selecting **Grid -> Create Grid** from the menu. In the Grid Creation dialog enter 11 for ni, 12, for nj, and 450 for the width and select the OK button (Figure 9A). We'll create a finer grid once we have edited our initial guess of centerline location.
5. **Adjust your centerline and grid curvature:** In this step you will adjust the centerline point locations until you have a grid that encompasses the flow area, good boundary angles to the incoming and outgoing flow direction, and a downstream boundary at a measured water-surface elevation which we'll use as a boundary condition By adjusting the centerline points of the grid you can adjust the grid curvature, and the orientation of the upstream and downstream boundaries. First, select ***Grid Creating Condition*** in the Object Browser. This will turn on the centerline you created in step 3 above. You can adjust the location of each centerline point by moving the mouse over the point and using a mouse left-click and drag to move it. If you don't like your move select the undo button (↶) in the main toolbar.
6. **Rebuild grid to final 2 meter resolution:** Once you are satisfied with the location of your grid, rebuild the grid such that it has a 2 X 2 meter spacing between nodes. From the menu select **Grid -> Create Grid** and enter values for ni and nj, such that the resulting values of di and dj are approximately 2 meters. You can iterate on these values by entering your guess into ni and nj, then selecting the Apply button to see the resulting values of di and dj (Figure 9A).
7. **Map measured values to the grid:** Map or interpolate all geographic data to the grid, in this case just the elevation data. We will map values to the grid using the TIN of the values. This is the default interpolation method. This value is set from the menu by selecting Grid -> Attributes Mapping -> Setting. Note that Geographic Mapping is set to Mapping with TINs in the Grid Attribute Mapping Setting dialog (Figure 9B). Next we will execute the mapping. From the menu select **Grid -> Attributes Mapping -> Execute**. If you turn on the ***Grid | Node attributes | Elevation*** in the Object Browser, your project should look similar to that in Figure 9C.

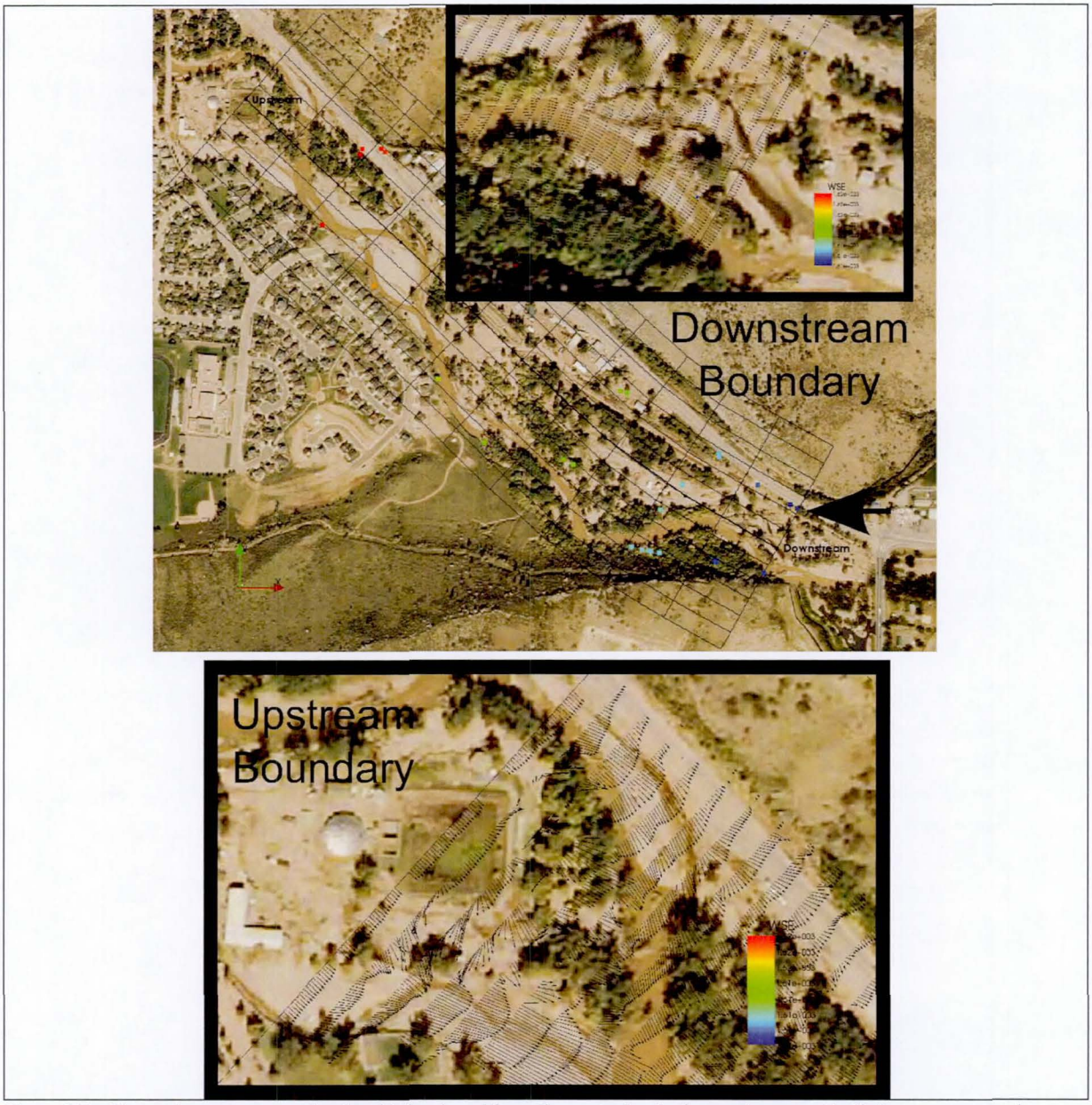


Figure 8 For illustration purposes a simple grid developed from the centerline is shown. Importantly 1) the upstream and downstream grid boundaries are aligned nearly perpendicularly to the incoming and outgoing flow directions, 2) there are no constrictions in flow immediately downstream of the boundary (see the Upstream Boundary inset). Finally the width of the channel encompasses the potential flow area.

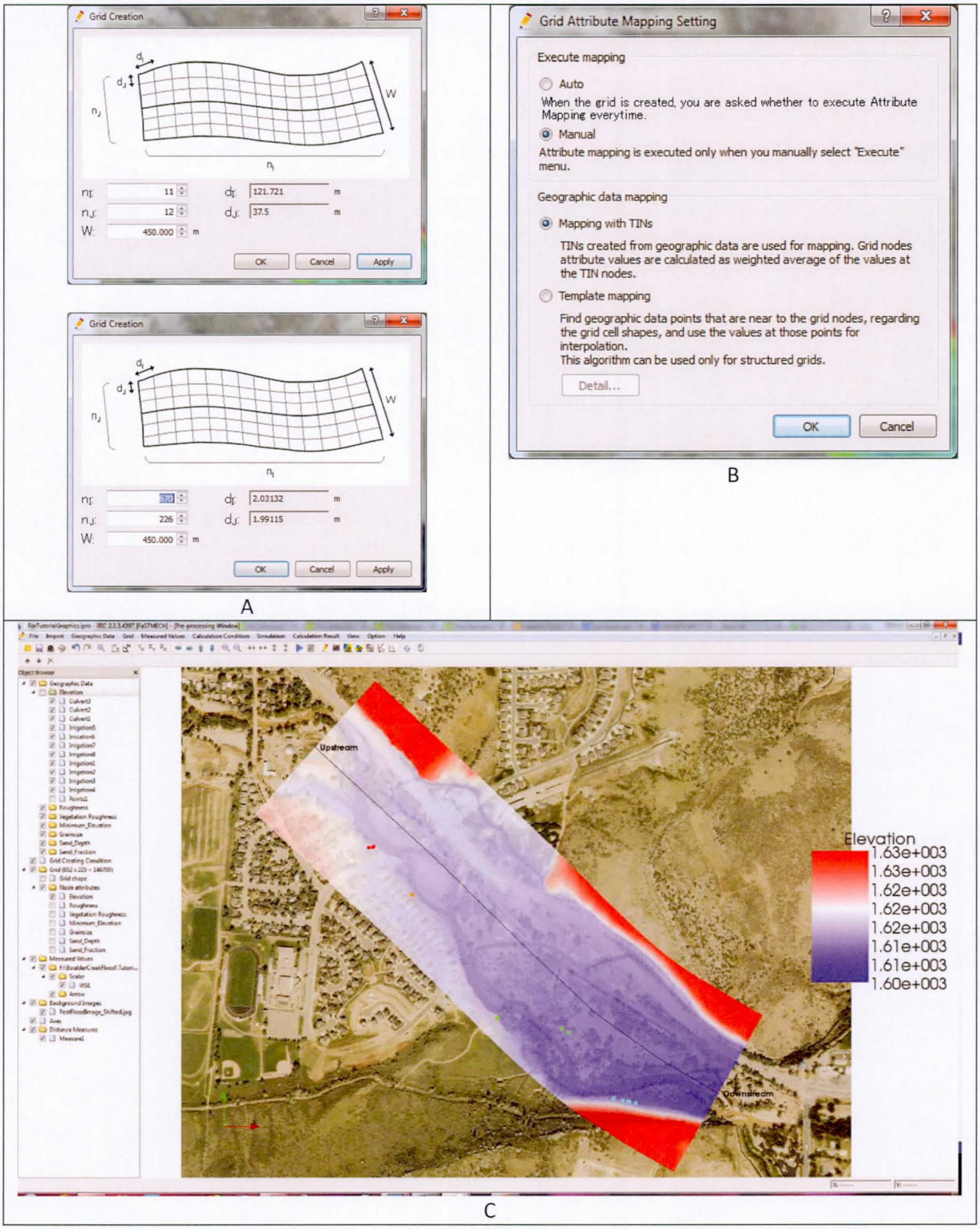


Figure 9 A) Grid Creation parameters for the simple grid in Step 4 (top) and for the final refined grid (bottom). B) The Grid Attribute Mapping dialog with the mapping method set to use the TIN. C) The elevation as mapped to the computational grid. Note that the location of the weirs are approximated by channels across the roads. In this way you can see the result of the polygons we drew earlier.

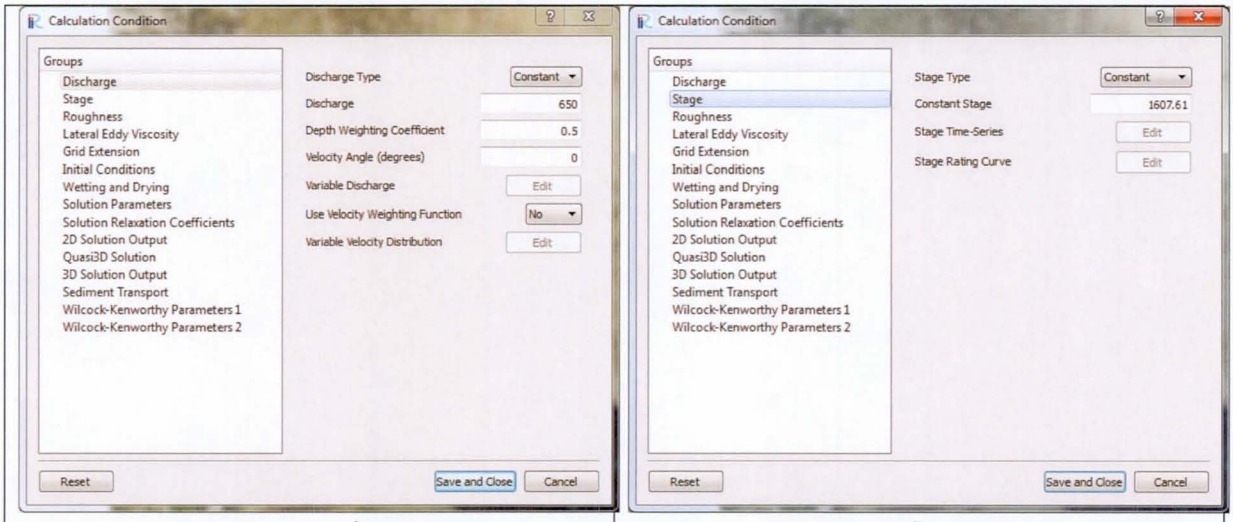
Task 4: Determine the peak discharge

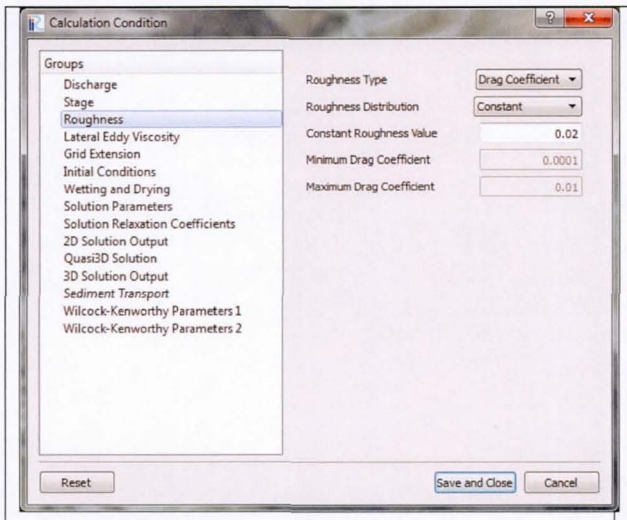
Now that you have created your grid and mapped the elevations values on to it you are ready to begin simulating flow through the reach. The goal here is to estimate the peak discharge using the available topography and measured HWMs. Both discharge and roughness are unknown. To begin in Part A, we will first determine the best discharge corresponding to a single value of roughness. To estimate this peak discharge we will run a series of simulations that sweep through a reasonable set of discharge values. (Table 1). The resulting simulated water-surface elevations will be compared to the measured HWMs and a root mean squared error (RMSE) will be determined for each simulation. We will record those RMSE values in Table 1 and then estimate the peak discharge from the discharge value that results in the best prediction to the measured HWMs.

Part A

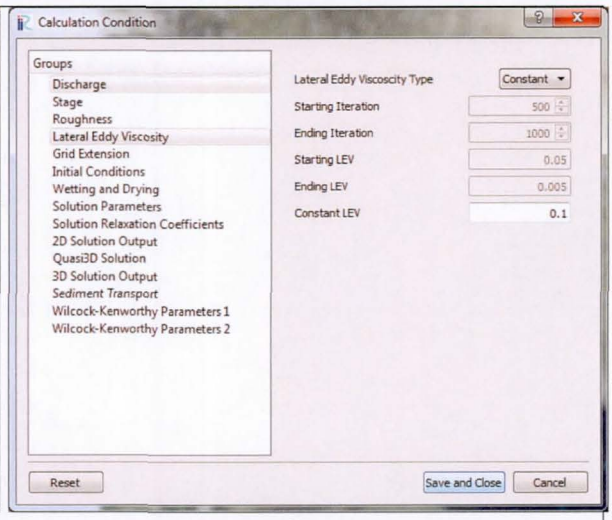
To help you get started, we provide the input conditions for the 650cms flow with a drag coefficient of 0.02. The input conditions are provided below in Figure 10. A few notes before you begin.

a. Note that we are using a 1-dimensional model (Figure 10F) to determine the initial water-surface elevation through the reach. For each new discharge you simulate you will have to change the discharge in the Discharge and Initial Conditions tab (Figure 10 A & F respectively).
b. We have provided you with the stage of the most downstream measured HWM which is 1607.61 meters. It will stay constant for all the simulations.

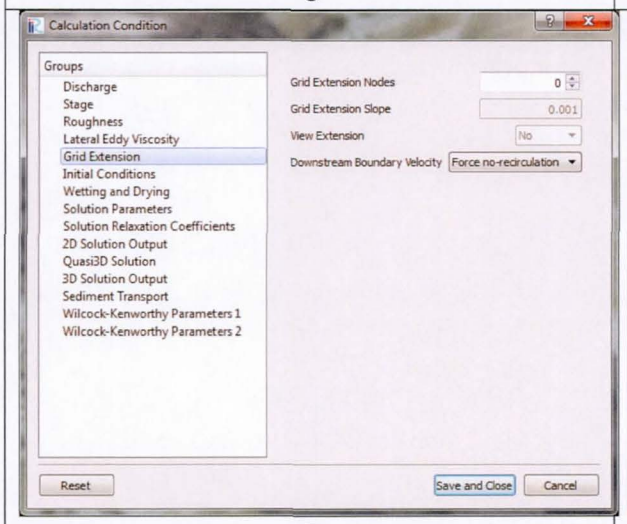



Calculation Condition
Groups
Discharge
Stage
Roughness
Lateral Eddy Viscosity
Grid Extension
Initial Conditions
Wetting and Drying
Solution Parameters
Solution Relaxation Coefficients
2D Solution Output
Quasi3D Solution
3D Solution Output
Sediment Transport
Wilcock-Kenworthy Parameters 1
Wilcock-Kenworthy Parameters 2
Roughness Type
Drag Coefficient
Roughness Distribution
Constant
Constant Roughness Value
0.02
Minimum Drag Coefficient
0.0001
Maximum Drag Coefficient
0.01
Reset
Save and Close
Cancel

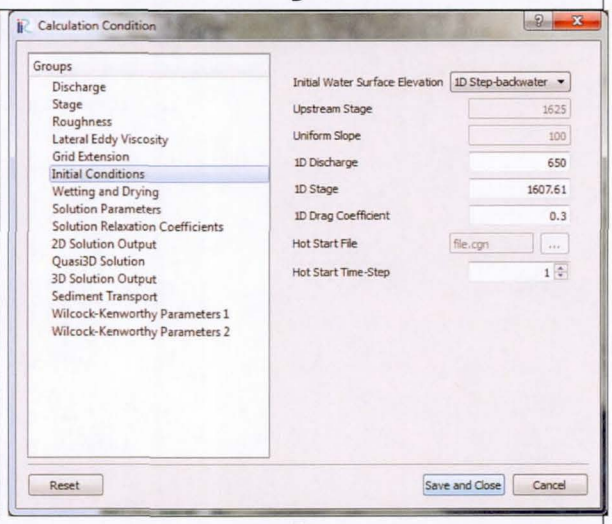
C


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 Viscosity Type
Constant
Starting Iteration
500
Ending Iteration
1000
Starting LEV
0.05
Ending LEV
0.005
Constant LEV
0.1
Reset
Save and Close
Cancel

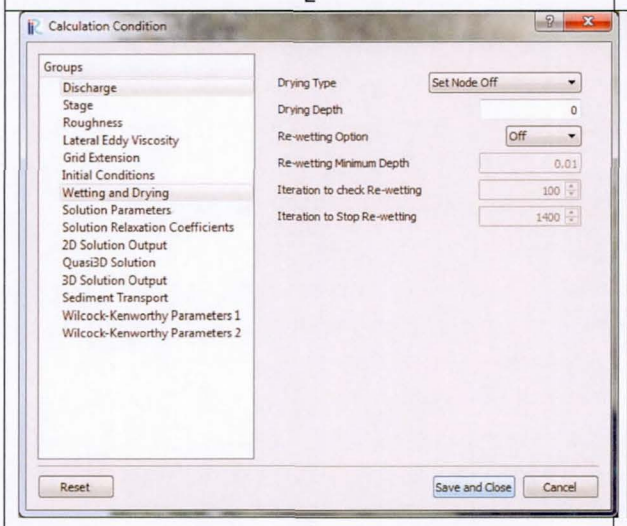
D


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

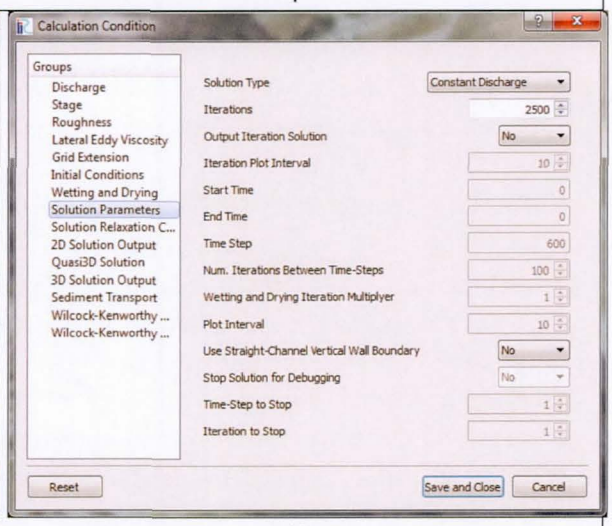
E


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
1D Step-backwater
Upstream Stage
1625
Uniform Slope
100
1D Discharge
650
1D Stage
1607.61
1D Drag Coefficient
0.3
Hot Start File
file.cgn
Hot Start Time-Step
1
Reset
Save and Close
Cancel

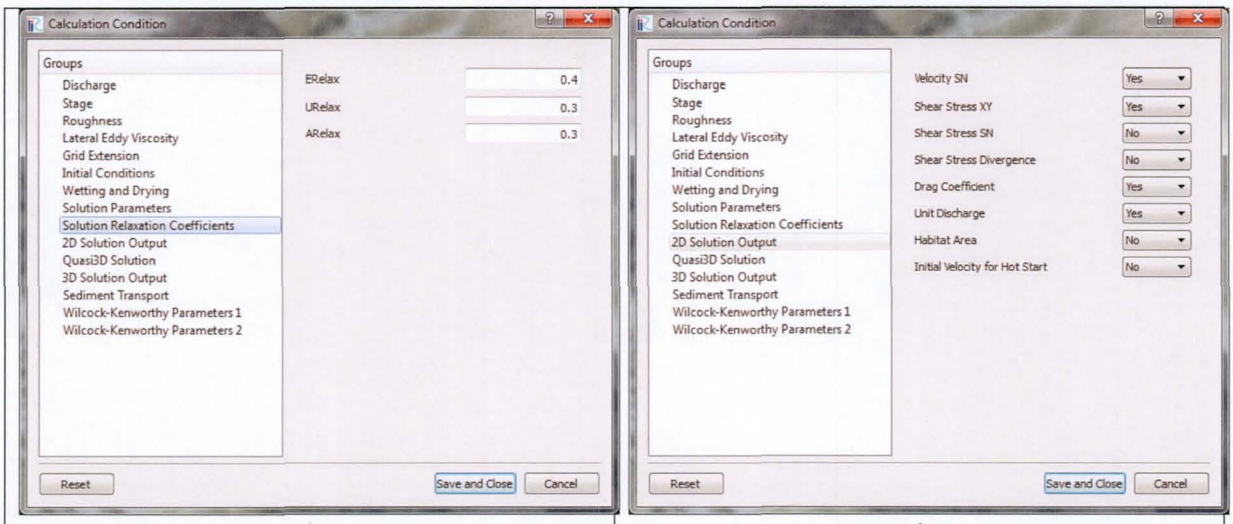
F


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
Drying Type
Set Node Off
Drying Depth
0
Re-wetting Option
Off
Re-wetting Minimum Depth
0.01
Iteration to check Re-wetting
100
Iteration to Stop Re-wetting
1400
Reset
Save and Close
Cancel

G


Calculation Condition
Groups
Discharge
Stage
Roughness
Lateral Eddy Viscosity
Grid Extension
Initial Conditions
Wetting and Drying
Solution Parameters
Solution Relaxation C...
2D Solution Output
Quasi3D Solution
3D Solution Output
Sediment Transport
Wilcock-Kenworthy ...
Wilcock-Kenworthy ...
Solution Type
Constant Discharge
Iterations
2500
Output Iteration Solution
No
Iteration Plot Interval
10
Start Time
0
End Time
0
Time Step
600
Num. Iterations Between Time-Steps
100
Wetting and Drying Iteration Multiplyer
1
Plot Interval
10
Use Straight-Channel Vertical Wall Boundary
No
Stop Solution for Debugging
No
Time-Step to Stop
1
Iteration to Stop
1
Reset
Save and Close
Cancel

H



I | J

Note: The last 5 groups, Quasi3D Solution, 3D Solution Ouput, etc. are not used in this simulation because we are only running a simple flow solution. For more information on the Quasi-3D solution look at Tutorial 3. For more information on sediment-transport look at Tutorial 5 & 6.

*Figure 10 The calculation conditions for the FaSTMECH model accessed by selecting **Calculation Condition -> Setting** from the menu. Each tab for a flow solution (without sediment transport) is shown including: A) Discharge, B) Stage, C) Roughness, D) Later Eddy Viscosity, E) Grid Extension, F) Wetting and Drying, G) Solution Parameters, and H) Solution Relaxation Coefficients.*

1. **Enter the calculation conditions for a simulation:** From the menu select **Calculation Condition -> Setting**, and Fill in the values for each tab of the Calculation Condition dialog as those in Figure 10. Except as explained above in (a) enter the discharge corresponding to the discharge you are modeling in the Discharge and Initial Conditions tab.
2. **Run the simulation:** From the menu select **Simulation - > Run**. The Solver Console view will be activated in iRIC and you will see the progression of the simulation showing the Iteration number and the mean error on discharge. The mean error on discharge is the root-mean squared error between the specified discharge and the discharge calculated at each of the cross-sections in the grid. Watching these values provides some insight to the simulation convergence. Ideally the values should converge to values less than 3 (usually much smaller) and should be relatively steady or unchanging at the end of the simulation. However, some small change in values is expected. If the simulation is successful you will see the Solver Finished dialog. Select OK and proceed to the next section.
3. **If the simulation stops before the number of iterations specified:** You will see an error message in the Solver Console view. **Note the last iteration number** and select the button from the toolbar to end the simulation. The best way to begin investigating why your simulation did not complete is to run the simulation for approximately 5 iterations before it stopped (which is why you noted the iteration the solution stopped) and look at the solution. Running the simulation for 5 iterations before the solution stopped is a good rule of thumb but it may be anywhere from 1-10 iterations to get a solution which will provide some clue as to why the solution stopped. Generally viewing the water surface elevation and/or the velocity vectors will provide some hints to why the solution stopped. Discuss with your neighbor or ask an instructor for help.
4. **Compare the simulated water-surface elevation to the measured HWMs and record the RMSE value:** When the simulation is finished, select **Calculation Result -> Compare with measured values**

from the menu and you will launch the Verification dialog. Initially it starts with the Verification Setting dialog (Figure 11A). Set the Physical Value in the drop-down list to WaterSurfaceElevation and press the OK button.

a. In the Verification dialog you will see a comparison between the measured data (circles) and simulation (line) as a function of stream-wise distance (Figure 11B). Note that in the Verification dialog the root mean squared error, a measure of the mean difference between the measured and simulated data is displayed in the dialog below the comparison plot. Write this number down in the appropriate column of Table 1.
b. You now know how to get a comparison between measured and simulated water-surface elevations in iRIC. (Figure 11 B. You will fill your Table 1 with the RMSE value for each discharge and drag coefficient pair.

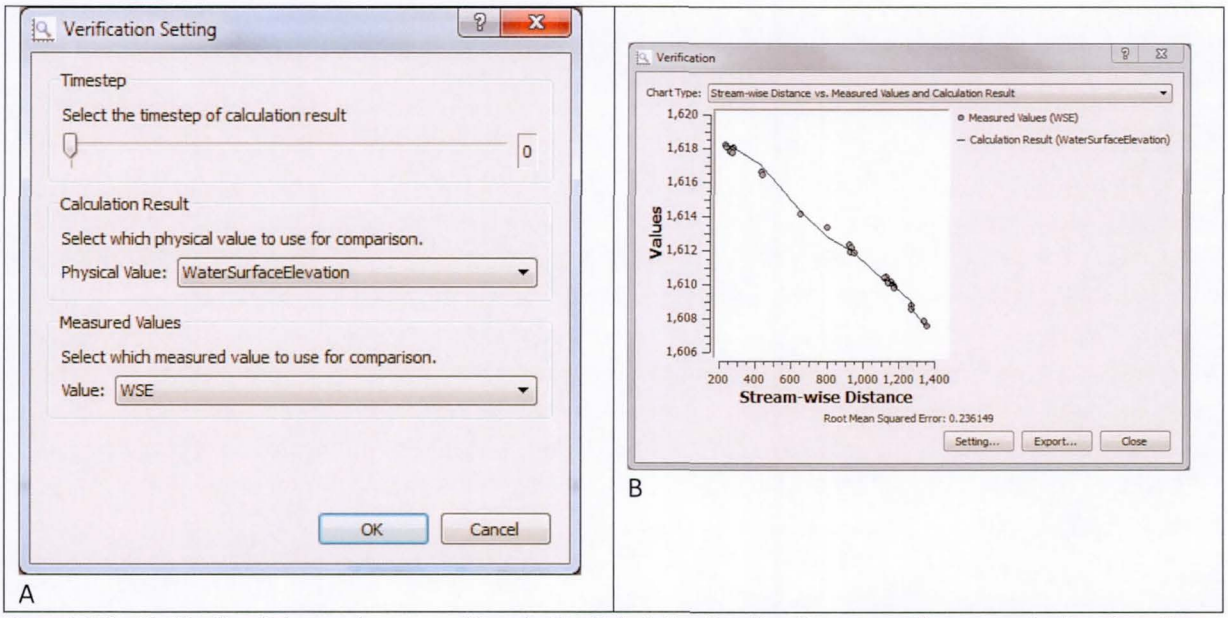


*Figure 11 The Verification dialog can be accessed by selecting **Calculation Result -> Compare with measured values** from the menu. A) The initial dialog provides the setting to choose which simulation scalar set will be used to compare with the measured data, in this case WaterSurfaceElevation is chosen. B) The comparison from the simulation of 650 m^3/s with 2500 iterations.*

Table 1 Your root mean square error, shown in the Verification dialog for each simulation pair of discharge and drag coefficient (Cd) will be recorded in this table.

	Flow m^3/s								
Cd	400	450	500	550	600	650	700	750	800
0.02									

Part B

In Part A you assessed the best discharge value given the measured high water marks, for a drag coefficient of 0.02. However, we really don't know what the drag coefficient is. Generally we calibrate each flow over a range of drag coefficients to determine the best drag coefficient for each different flow magnitude. That is the calibration process. We calibrate in this way because the total roughness is the additive roughness over a range of scales from the individual grains on the bed, to bedforms including ripples and dunes if present, increasing in scale all the way to the plan form shape of the river itself. Between the scale of the computational grid, and the resolution of the measured topography, our simulation will capture some subset of the various scales of roughness. The rest of the roughness is lumped into a single value represented by the drag coefficient.

A good strategy when determining the peak discharge based on surveyed topography and high water marks is to sweep through the parameter space of both discharge and roughness. Comparing the simulated water-surface elevation at each discharge and roughness pair and recording the RMSE of each simulation will result in a table that will provide a good estimation of the peak discharge assessed from the discharge roughness pair that results in the lowest RMSE.

In addition, the resulting table also provides a sensitivity analysis on the peak discharge. Based on the results you may find a well-defined peak discharge where the RMSE values clearly indicate a single discharge value, or you may find that a well-defined discharge is not found and instead the simulations indicate that the peak discharge lies within a range of flows. In other words, you may find that there is a clear lowest RMSE value that indicates the peak discharge, or you may find that there is range of flows that have RMSE errors that are too similar to definitively distinguish a best value. While this method is more time intensive it provides an improvement to the standard 1-dimensional approach where a single roughness value is specified a priori. In Part B we will run simulations for the set of drag coefficients (roughness values) and discharges defined in Table 2. Before you begin your simulations you might find it instructive to look at grids of other students near you and provide each other feedback and make sure you are all on the same page.

Table 2: Fill in the RMSE values for each simulation pair of discharge and drag coefficient (Cd)

	Flow m^3/s						
Cd	500	550	600	650	700	750	800
0.01							
0.0125							
0.015							
0.0175							
0.02							
0.0225							
0.025							

iRIC Software

Changing River Science

Nays2DH Examples

Copyright iRIC Project All Right Reserved.

Introduction to Nays2DH

1. What is Nays2DH?

Nays2DH is a computational model for simulating horizontal two-dimensional (2D) flow, sediment transport, morphological changes of bed and banks in rivers. Although iRIC has provided several 2D solvers, such as Nays2D, Morpho2D, FaSTMECH, etc., we believe that the users sometimes may confuse which solver is preferable for their own case. Therefore, we decided to combine Nays2D and Morpho2D to provide a more powerful and user friendly tool for iRIC users, we called it Nays2DH.

Nays2D, developed by Dr. Yasuyuki Shimizu in Hokkaido University in Japan, is a plane 2D solver for calculating flow, sediment transport, and bed evolution and bank erosion in rivers. By joining many developers to Nays2D project, several functions, for instance, river confluence model, mixed grain size model and hot start function, have been added. Nays2D is attached to iRIC and RIC-Nays which is a predecessor project of iRIC. Nays2D includes several options for simulating river flows such as an unsteady vortex generation in open channel flows and river morphodynamics. River morphodynamics includes the initiation and development of free bars in rivers and the interaction between free bars and forced bars in meandering channels. In addition, Nays2D has been applied to several practical applications: bed evolution process in rivers affected by trees and vegetation, calculation and prediction of inundation on floodplains, sedimentation in river confluences, analysis of bank erosion and flood disasters.

Morpho2D developed by Dr. Hiroshi Takebayashi is a solver to simulate the two-dimensional morphodynamical changes in rivers. Initially, it was attached in the RIC-Nays as 'Mixed grain-size model'. Since iRIC released version 1, this solver has been reformed as Morpho2D. Morpho2D includes several possibilities for simulating the morphological changes of river bed with uniform and mixed sediment sizes, and simulating the development of free bars with sorting of sediment particles on the river bed. Morpho2D has also been applied to several real world applications in river engineering, such as analysis or bed evolution under vegetation effects, sediment transport and bed evolution with the non-erodible bed.

Both solvers have their own advantages, but they include a common part as they are both 2D models. We developed a new and more powerful solver by combining the functions of these models. In this version, the user can choose a sediment transport model based on the functions which were implemented in the both solvers. The user also can combine a river confluence model, bank erosion model, bedload-suspended load simulation in mixture sediment, bedload layer model and fixed bed model, and also able to change the sediment supply rate from the upstream end. However, there are some uncoupled parts in the current version. The combination of the functions explained above may not work in some points, and the seepage flow model used in Morpho2D is not implemented. These points will be improved in the near future.

Nays2DH development team
Yasuyuki SHIMIZU, Hokkaido University, Japan.
Hiroshi TAKEBAYASHI, Kyoto University, Japan.

2. Features of the hydrodynamic model

(1) As a coordinate system, a general curvilinear coordinate system is adopted, allowing direct consideration of complex boundaries and riverbed shapes.

(2) Calculations involving confluences of a main channel and tributaries can be performed.

(3) For the finite differencing applied to the advection terms in the momentum equations, the user can select from among [Upwind difference method (first order)] and [CIP method]*1.

(4) For the turbulent field calculation method, the user can select from among [Constant eddy viscosity], [Zero-equation model] and [k-ε model].

(5) Various settings are possible for boundary conditions of the upstream and downstream ends, including periodic boundary conditions, downstream water surface elevation setting and upstream velocity setting. This makes it convenient to set boundary conditions from limited observational data.

(6) For setting the initial water-surface profile, the user can select from among [Constant slope], [Line], [Uniform flow calculation] and [Non-uniform flow calculation].

(7) The bottom friction evaluation method is set by using Manning's roughness parameter. This parameter can be set to each computational cell.

(8) Any obstacle within the calculation target area can be taken into account on a calculation-cell basis. For each calculation cell, a flag can be determined to define an obstacle. By this means, river structures such as bridge piers can be easily incorporated in calculation.

(9) The effect of vegetation on the flow can be introduced as a drag force. The user can set the density of vegetation and the height of vegetation in each computational cell.

3. Features of the sediment-transport model and riverbed deformation model

(1) The model allows the user to select between performing flow regime calculation only and performing riverbed deformation calculation along with flow regime calculation.

(2) For sediment transport, the user can select between bed load only and bed load + suspended load.

(3) For grain size distribution, the user can select between uniform and non-uniform. When non-uniform grain size is selected, changes in the grain size distribution in the depth direction during calculation can be stored on a multilayer basis.

(4) The sediment supply rate at the upstream boundary condition can be adjusted. The user can set the ratio of "sediment transport supplying rate from upstream " to "equilibrium sediment transport rate".

(5) The sediment transport and mass balance of it on the fixed bed can be calculated by introducing the bed-load layer model. In addition, the user can set either the elevation of river bed or fixed bed.

*1 This is a high-order finite-difference method. By using a cubic polynomial as an interpolation function, numeric diffusion is reduced, thus enabling high-precision local interpolation.

(6) The user can select a total bedload transport formula from among [Meyer-Peter and Muller formula] and [Ashida and Michiue formula]. For calculating the bedload transport vector, the user can select [Watanabe formula] or [Ashida, Egashira and Liu formula].

(7) The user can select an upward flux of suspended sediment from river beds from among [Lane-Kalinske formula] and [Itakura and Kishi formula].

(8) Morphological factor which is an acceleration parameter of bed evolution can be set.

(9) The solver incorporates a slope collapse model. An unrealistic steep slope can sometimes occur in the bed evolution, if it is simulated by using only Exner equation. In this model, if the angle of riverbed exceeds the critical angle, which is a user parameter, the bed is instantaneously corrected to adjust the bed angle (to be not larger than the critical angle). By applying this model, the user can treat the morphologic evolution processes such as bank erosion, which cannot be well captured by using only continuity equation of riverbed.

(10) Bank erosion takes into account of the angle of repose. More specifically, when the riverbed slope exceeds the angle of repose as riverbed deformation proceeds, adjustment is made by exchanging sediment with surrounding cells such that the bed angle slope decreases and fall lower than the angle of repose. In addition, if the channel widens from bank erosion, calculation grids are automatically moved accordingly.

4. Other features

(1) HotStart is provided (starts the calculation from where the previous calculation left off, based on the previous calculation results).

(2) The user can use parallel computing by OpenMP for reducing the computational time. The user can specify the number of the CPUs used for the computation.

5. Calculation steps

You can use Nays2DH in the iRIC for your calculations as following procedures;

Launch Nays2DH

Prepare to use Nays2DH in iRIC

↓

Generating the computational grid

Generate the computational grid by using several geographic data (river survey data, Digital Elevation Mode, etc).

↓

Setting the computational conditions

Set initial and boundary conditions, etc.

↓

Running a simulation

Use Nays2DH to run a simulation.

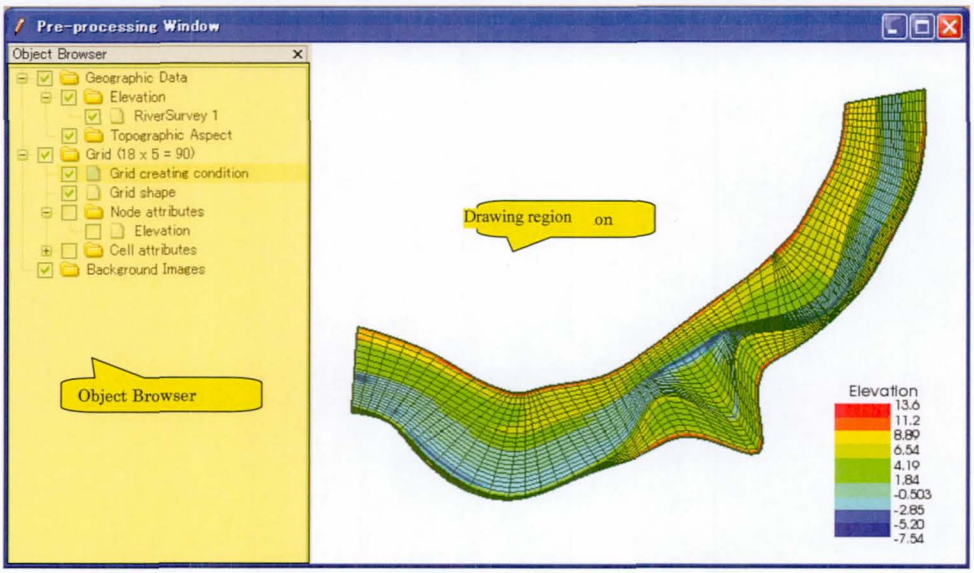
↓

Visualizing the simulated results

Visualize the simulated results by contour maps, vectors, streamlines, particles and etc.

6. Operations in [Object Browser]

iRIC uses [Object Browser] for all sub windows except [Solver Console] and [Graph Window].



[Object Browser] of the [Pre-processing Window]

- **In [Object Browser], the following operations can be made:**

Selecting an operation target:

Left click on the item to select data to be edited. The menu enables operations that are possible under the configuration of items selected in [Object Browser].

Switching between "show" and "hide":

Make a check mark in the box left of the item, to make it shown on the canvas. For items in hierarchical trees, if the check mark for the upper-level item is removed, the item and all the lower-level items are hidden.

Changing the order of items:

An item can be moved up and down to change order. This operation is available only for [Geographic Data] and [Background Images] on [Pre-processing Window]. Select ↑ / ↓ on the Operation Toolbar to change order of items.

Changing a display setting in [Property] dialog:

Open right-clicking menu and click on [Property] to open a dialog and modify settings. Some items do not have [Property] dialog.

Importing / exporting data

Open right-clicking menu on an item to facilitate data import or export. Some items do not have import / export feature.

- **View changing operations can be made with Ctrl key and mouse operations.**

View changing operations on the canvas

Operation	Action	Cursor during action
Rotate	Right drag while pressing Ctrl key.	
Zoom in/out	Drag the scroll button (wheel) while pressing Ctrl key, or scroll up/down the scroll button.	
Drag	Left drag while pressing Ctrl key.	

7. Toolbars

[Main Toolbar] handles file I/O, view setting operations for canvas, solver launching and opening sub windows.

[Main Toolbar]

Functions of the items on the [Main Toolbar]

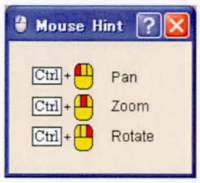
Icon	Name	Description
	[Open]	Opens a project file
	[Save]	Saves the project.
	[Save Snapshot] (N)	Saves a snapshot of the active window.
	[Continuous Snapshots / Google Earth Export]	Saves images of each time step produced by the Post-processing Window/Graph Window into image files, and outputs a KML file for Google Earth.
	Undo	Undoes the most recent action.
	Redo	Redoes the most recent undoed action.
	Fit	Zooms out such that everything is displayed in the window.
	Reset Rotation	Rotates the graphic such that the X-axis is pointing rightward (in the positive direction) and the Y-axis is pointing upward (in the positive direction).
	90-Degree Rotation	Rotates 90 degrees counterclockwise.

	XY-plane	Rotates the graphic such that the X-axis is pointing right-ward (in the positive direction) and the Y-axis is pointing upward (in the positive direction).
	YZ-plane	Rotates the graphic such that the Y-axis is pointing right-ward (in the positive direction) and the Z-axis is pointing upward (in the positive direction).
	XZ-plane	Rotates the graphic such that the Z-axis is pointing right-ward (in the positive direction) and the X-axis is pointing upward (in the positive direction).
	Moves to the left	Shifts the viewpoint to the left.
	Moves to the right	Shifts the viewpoint to the right.
	Moves to the top	Shifts the viewpoint to the top.
	Moves to the bottom	Shifts the viewpoint to the bottom.
	[Zoom in]	Zooms in on the display.
	[Zoom out]	Zooms out from the display.
	Extension in the X-axis direction	Extends only in the X-axis direction.
	Shortening in the X-axis direction	Shortens only in the X-axis direction.
	Extension in the Y-axis direction]	Extends only in the Y-axis direction.
	Shortening in the Y-axis direction	Shortens only in the Y-axis direction.
	[Run]	Starts the solver.
	[Stop]	Stops the solver.
	Display Pre-processor	Opens [Pre-processor Window].
	Display [Solver Console]	Displays [Solver Console].
	[Open New 2D Post-processing Window]	Opens a new [2D Post-processing Window].
	[Open New Bird's Eye 2D Post-processing Window]	Opens a new [Bird's Eye Post-processing Window].
	[Open New 3D Post-processing Window]	Opens a new [3D Post-processing Window].
	[Open New Graph Window]	Opens a new [Graph Window].
	[Compare with measured values]	Opens a dialog to compare calculation results with meas-ured values
	[Reload Calculation Result]	Reloads calculation result.
	[Mouse Hints]	Displays the [Mouse Hints] dialog.

- **Mouse Hint] dialog**

  Menu bar: [Help] (H) ▶ [Mouse Hints] (M)

  Operation Toolbar:



[Mouse Hint] dialog

8. Grid creating algorithms available in Nays2DH

Several types of grids generator can be used in Nays2DH solver.

Grid creating algorithms available in Nays2DH solver

Created grid	Item	Description
Two-dimensional structured grid	[Create grid from polygonal lines and width]	Creates a grid that smoothly follows a polygonal line.
	[Create grid from river survey data]	Creates a grid from [River Survey Data]. In addition to transverse lines being set, division points are set on the transverse lines the river centerline and left / right bank lines.
	[Create grid by dividing rectangular region]	Creates a rectangular grid that is evenly divided in the x and y directions.
	[Create compound channel grid]	Creates a grid that has lower channel, by defining grid creating region and lower channel region.
	Multifunction grid generator	Create a geometric grid such as straight, sine-generated curve, zigzag, or Kinoshita meandering curve
	Simple grid generator	Create a geometric grid with straight or sine-generated curve

Tutorial 1.

Flow Simulation for a Channel with Structures

Purpose of this tutorial

Simulate the flow regime (water depth and flow velocity) by using Nays2DH for a channel with obstacles and with a certain discharge, and verify the simulation results.

Outline:

Creating the calculation grid

Using the grid creating algorithm of iRIC, create a calculation grid for a straight channel (10 m long, 0.5 m wide) that has four obstacles that resemble spur dikes. Create calculation grids: 21 division points in the transverse direction, and 201 division points in the longitudinal direction.

Setting the calculation conditions

Set simulation conditions for steady flow with a discharge of 0.003 m3/s for 600 seconds (10 minutes). Set various other conditions necessary for simulation.

Making a simulation

Visualizing the calculation results

This section explains examples of how the "particles" and streamline are used to visualize the flow field.

1. Generate a computational grid

Operations related to [Geographic Data] are available from [Geographic Data] menu when the [Pre-processing Window] is active. [Geographic Data] types that users can import and edit depend depends on the solver. Currently, iRIC supports the following three types of [Geographic Data].

- River Survey Data
- Elevation data
- Polygon

In Tutorial 1, generating a rectangular channel computational grid with rectangular cross sections, use [Multifunction Grid Generator] provided in iRIC. By using [Multifunction Grid Generator], user can create a geometric channel such as Straight, Sine-generated curve, Zigzag, and Kinoshita meandering curve. The operation to make a straight channel in Tutorial 1 is as below. Fist, select Menu bar -> [Grid] -> [Select Algorithm to Create Grid], and select [Multifunction Grid Generator] in the dialog of [select grid creating algorithm]. After the dialog for grid generation is opened, set the various conditions for generating the computational grid as follows;

[Channel Shape]----- Select Channel Shape : Straight, Cross Sectional Shape : Single Cross Section
[Cross Sectional Shape Parameters]----- Width : 0.5m, Number of Grid in Lateral Direction : 20
[Channel Shape Parameters]----- Wave Length of Meander : 10m, Wave Number : 1
Number of Grids in One Wave Length : 200
[Bed and Channel Shape]----- Initial Bed Shape : Flat (no bar), Channel Slope : 0.001
[Bed Condition] ----- Bed Condition : Movable Bed

The default values are used for the other parameters.

The computational grid will be generated as follows;

By using the mouse to move the grid and to zoom in and out, the grid becomes easier to see.

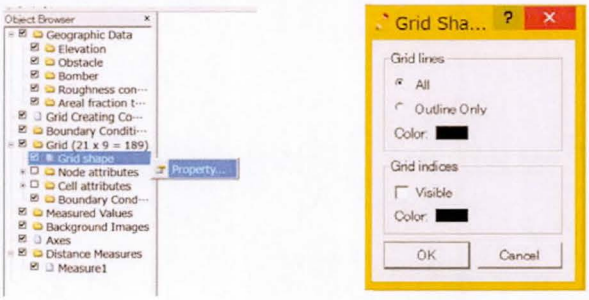
Display settings for grids in Object Browser as below.

Menu items in [Display Settings]

Menu	Description
[Grid shape] (S)	Edits display settings on grid shape.
[Node Attribute] (N)	Edits display settings on grid node attribute.
[Cell Attribute] (C)	Edits display settings on grid cell attribute.
[Set Up Scalarbar (U)	Edits display settings on color bar.

a) [Grid shape] (S)

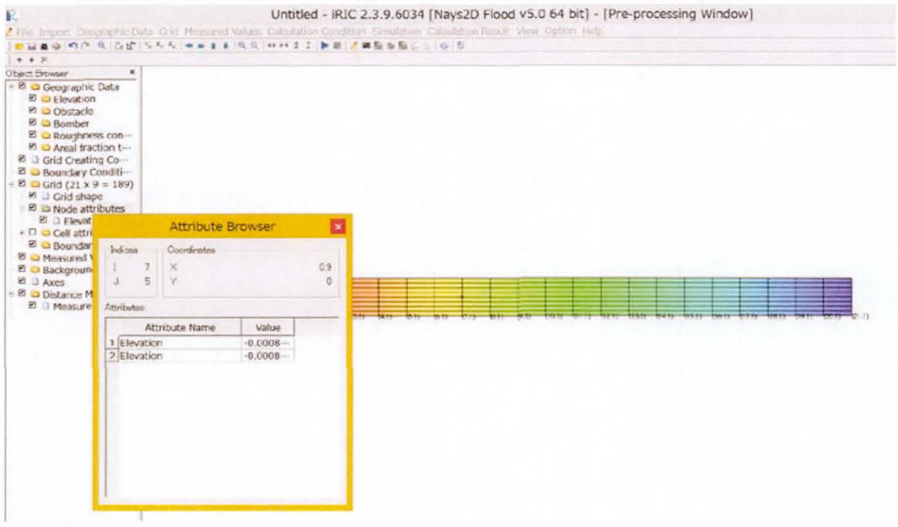
Right click on [Grid Shape] and select [Property], then [Grid Shape Display] dialog will appear. In the dialog, you can change the setting to visualize the calculation grid in the pre-processing window in iRIC.



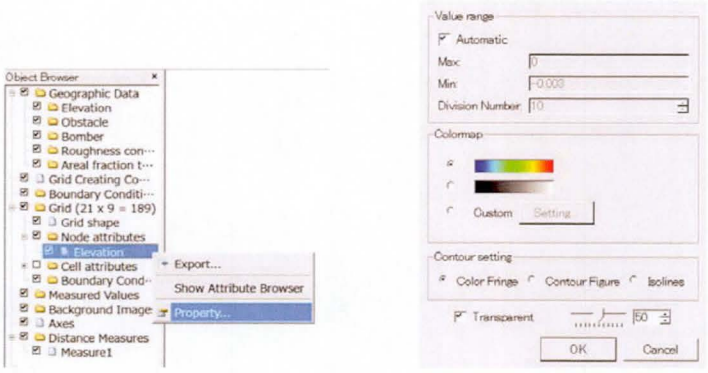
Example of grid shapes

(b) Node Attribute (N)

Right click on [Node Attribute] in the object browser and select [Show attribute browser], Attribute Browser window will appear as below. In this dialog, user can confirm the grid number of I and J, X and Y coordinate valuables, and Elevation value at the selected location. I is the grid number of the longitudinal direction, increasing toward to the downstream direction. J is the grid number of the transverse direction, increasing from left bank to right bank direction.



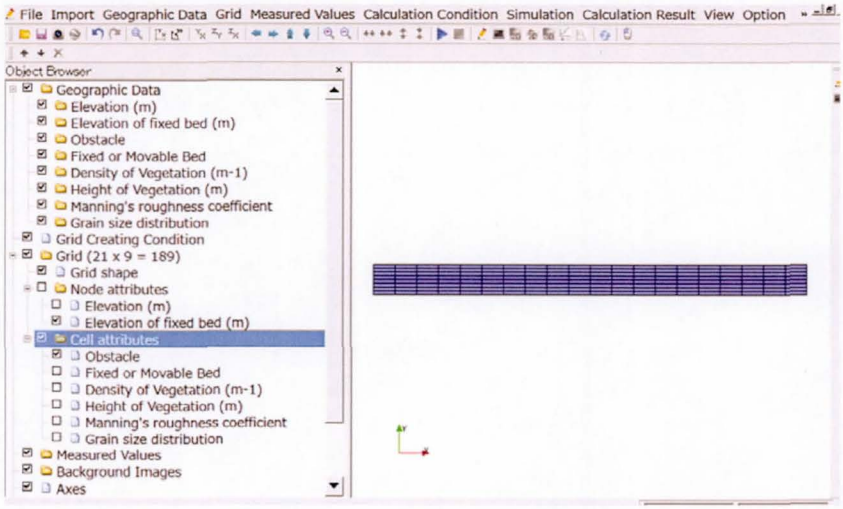
In addition, when you right click on [Node Attribute] -> [Elevation] in the object browser and select [Property], Grid Node Attribute Display will appear as below. In this dialog, you can change the maximum and minimum scale value of the elevation contour map, contour color setting, transparency of the contour map, and so on.



(c) Cell Attribute (C)

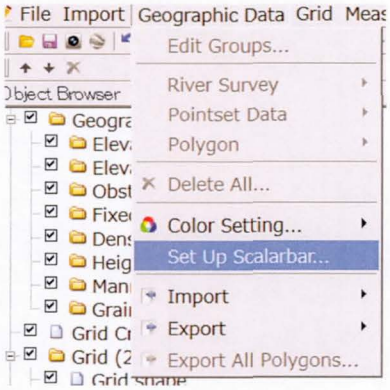
When you select [Cell Attribute], you can confirm the location of Obstacles, Fixed and Movable bed, and Height of Vegetation if you have set such valuables to the grid, and also confirm Manning's roughness coef-

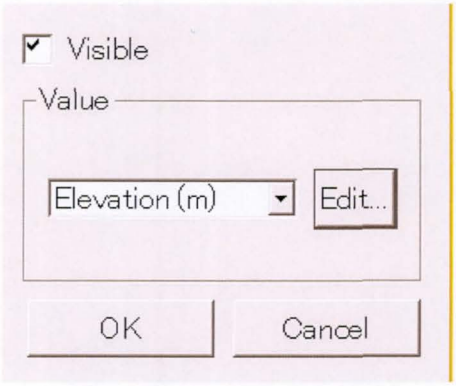
ficient and Grain size distributions in each grid cell.



(d) Set up Scalar bar (U)

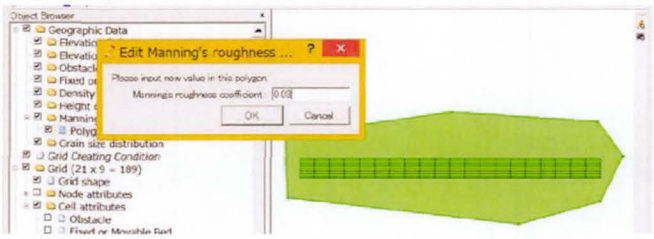
When you select [Geographic Data] -> [Set Up Scalarbar] on Toolbar, the [Scalar bar Setting] dialog will open. In the dialog as below. Select [Elevation (m)] here, then scale bar of Elevation will appear in the pre-processing window in iRIC. If you want to change the scale value of the Elevation, select [Geographic Data] -> [Color setting] on Toolbar.





2. Set Manning's roughness coefficient

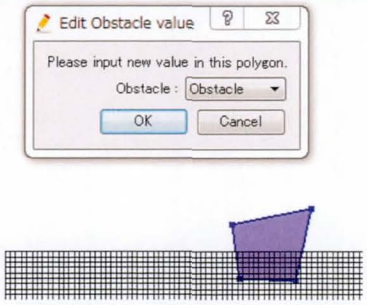
Set Manning's roughness coefficient on calculation grid. Select [Geographic Data] -> [Manning's roughness coefficient] in the Object Browser and select [Add] –> [Polygon] by right clicking on the [Obstacle]. Make a polygon by PC mouse as like figure below. If you want to add different Manning's roughness coefficient in each grid cell, make several polygons respectively.



3. Set Obstacles

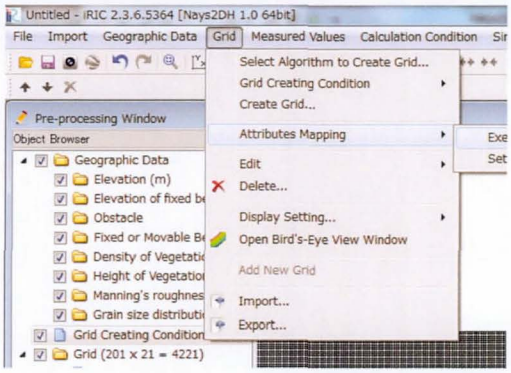
Set obstacles on calculation grid. Select [Geographic Data] -> [Obstacle] in the Object Browser and select [Add] –> [Polygon] by right clicking on the [Obstacle]. Make a polygon by PC mouse to select the area where an obstacle is to be set.

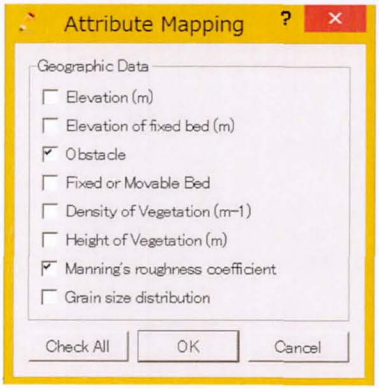
After making polygon, [Edit Obstacle value] window will appear.



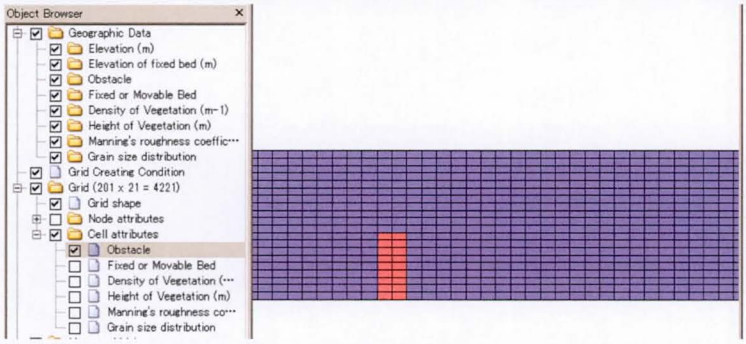
Select [Obstacle] and click on [OK].

Select [Grid] – [Attribute Mapping]. Then, [Attribute Mapping] window will appear. In the window, check graphical data which you want to give to the grid as attributes. Now, select only [Obstacle] and [Manning's roughness coefficient], and then click on [OK].



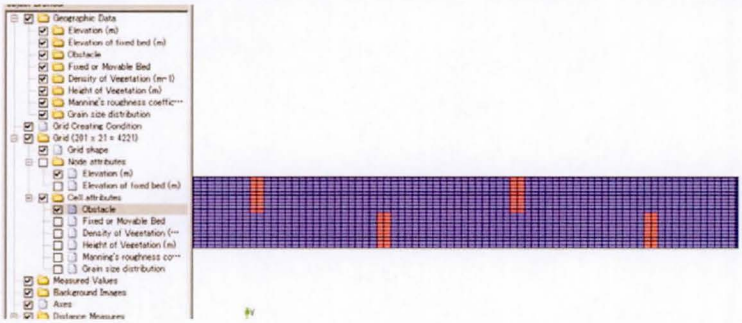


To check the results of attribute mapping, select [Grid] -> [Cell Attributes] in the Object Browser and make a check mark ☑ in these boxes; also, select [Grid] -> [Cell Attributes] -> [Obstacle] and make a check mark ☑ in these boxes.



Cells that share an attribute are displayed in the same color. The blue ones are fluid cells. The red ones are obstacle cells.

Set four obstacles on the calculation grid in the same way.



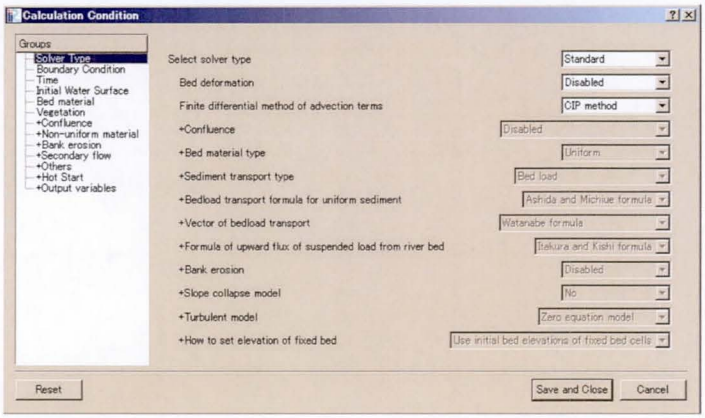
Example: Set four obstacles.

Note: Setting obstacles near inlet or outlet areas of the calculation flume will cause instability of your calculation.

4. Setting calculation conditions

Select [Calculation conditions] -> [Setting] on menu bar and set calculation conditions as followings. The default values are used for the other parameters.

Solver Type:



- Select solver type---- Standard
  - Bed deformation---- Disabled
  - Finite difference method of advection terms---- CIP method

Note) You can select the items with mark [+], in case that you select [Advanced] in [Select Solver Type].

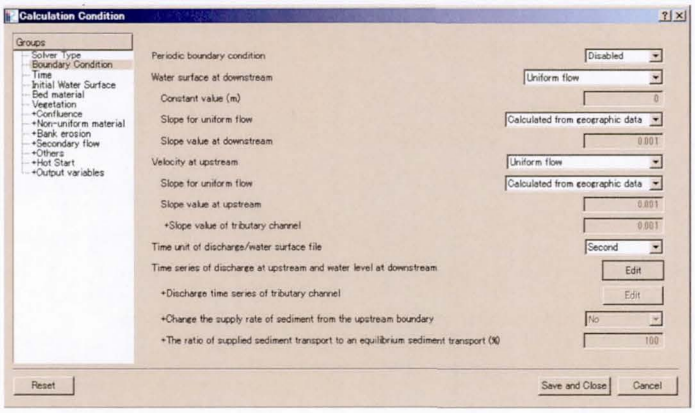
Nays2DH has two types of solvers: the standard edition and the advanced edition. The standard edition allows you to perform flow regime analysis of general river segments and analysis of bed deformation with uniform sediment material. The advanced edition allows you to deal with more complex boundary conditions and initial conditions such as the following: non-uniform grain size multilayer model, river confluence model, Hot start, and so on.

.

Description of Solver Type settings

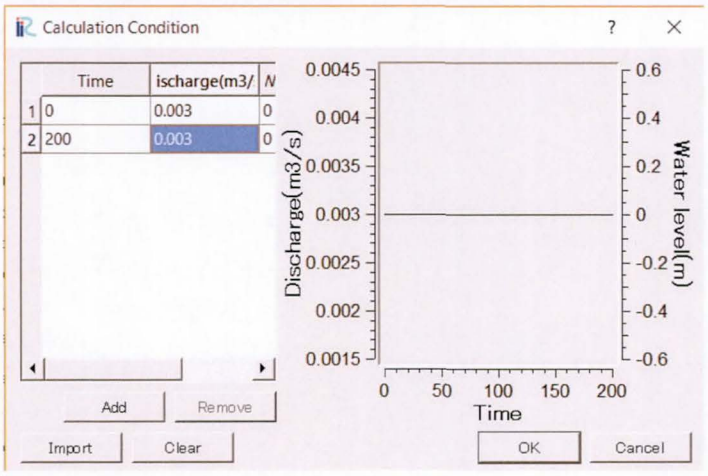
#	Item	Description	Remarks
1	Solver type	Sets the solver type. Choose [Standard] or [Advanced].	When [Standard] is selected, items marked [+] cannot be set.
2	Bed deformation	Sets whether or not bed deformation is calculated.	
3	Finite-difference method of advection terms	Select the finite-difference method of the advection terms between [Upwind difference method] or [CIP method].	

Boundary Condition:



- Periodic boundary condition
  ---- Disabled
- Water surface at downstream
  ---- Uniform flow
- Velocity at upstream
  ---- Uniform flow
- Time--- Second
- Click on [Edit] to set discharge

After click on [Edit], the window for setting the discharge will open as below. In the window, click on [Add] and input the following settings.



- Time---- 0 - 200 sec
- Discharge---- 0.003 (m3/s)

Descriptions of boundary condition settings

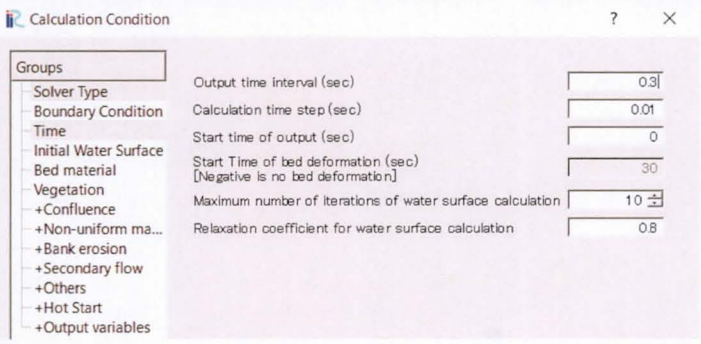
#	Item	Description	Remarks
1	Periodic boundary condition	Enables/disables the periodic boundary condition*2.	
2	Water surface at downstream end	Sets water surface at downstream end. [Constant value], [Uniform flow] or [Given from time series data] can be selected.	When set to [Given from time series data], the user should set the water level changes in [Time series of discharge at upstream and water level at downstream]
3	Constant value (m)	Enter a value if you have selected	

*2 [Periodic boundary condition] gives the hydraulic and sediment transport conditions at the downstream end to the upstream end. It is used when hydraulic conditions, channel conditions and sediment feeding conditions are of periodic nature, such as in the case of experiments. Before using it for an actual river, sufficient verification of periodicity is necessary.

		[Constant value] for [Water surface elevation at downstream end].	
4	Slope for uniform flow	Enter a value if you have selected [Uniform flow] for [Water surface elevation at downstream end]. Select [Calculated from geographic data] or [Constant value].	If [Calculated from geographic data] is selected, uniform flow calculation is done using the slope of the downstream end calculation grid.
5	Slope value at downstream end	Enter a value if you have selected [Constant value] for [Slope gradient for uniform flow simulation].	
6	Flow velocity distribution at the upstream end	Sets the velocity distribution at the upstream end. Select [Calculate from water depth at upstream end] or [Uniform flow].	
7	Slope for uniform flow	Set this if you select [Uniform flow] for [Flow velocity distribution at the upstream end]. Select [Calculated from geographical data] or [Constant value].	If [Calculated from geographic data] is selected, uniform flow calculation is done using the upstream toe slope of the calculation grid.
8	Slope value at upstream end	Enter a value if you have selected [Constant value] for [Slope gradient for uniform flow simulation].	
9	+Slope value of tributary channel	Enter a value when using the confluence model when you select [Constant value] for [Slope gradient for uniform flow simulation].	
10	Time unit of discharge/water surface elevation file	Sets the unit of time for the time columns of Discharge time series, Stage at downstream time series, and +Discharge time series of tributary channel. Select [Second] or [Hour].	
11	Time series of discharge at	Click [Edit] and set the discharge	Select [Second] or [Hour]. The unit

	upstream and water level at downstream	hydrograph and the water level variation	of discharge is "m^3/s."
13	+Discharge time series of tributary channel	Sets the discharge hydrograph of tributary channel when the confluence model is used.	Select [Second] or [Hour]. The unit of discharge is "m^3/s." The same data number and start-end time is required between the main and tributary channel.
14	+Change the supply rate of sediment from the upstream boundary	The user can change the supply rate of sediment from the upstream boundary.	When you select [Periodic boundary condition], you cannot select this option.
15	+The ratio of supplied sediment transport to an equilibrium sediment transport (%)	The ratio of supplied sediment-transport rate to an equilibrium sediment-transport rate can be defined.	The ratio is in %.

Time:



- Output time interval (sec)---- 0.3
- Calculation time step (sec) ---- 0.01
- Start time of output (sec) ---- 0

Descriptions of time-related settings

#	Item	Description	Remarks
1	Output time interval (sec)	Sets the time interval at which calculation results are to be output.	
2	Calculation time step (sec)	Sets the time interval of calculation steps.	This is an important parameter that determines calculation efficiency and stability.
3	Start time of output (sec)	Sets the time to start outputting calculation results.	
4	Start time of bed deformation (sec)	Sets the time to start bed deformation calculation.	If a negative sign is specified, bed deformation calculation is not per-

			formed.
5	Maximum number of iterations of water surface elevation calculation	Sets the number of internal iterations when water surface elevation is calculated.	If water surface elevation calculation is unstable, adjust this setting.
6	Relaxation coefficient for water surface elevation calculation	This is the coefficient that is used when water surface level calculation is performed.	

Many other conditions can be set; however, they do not need to be set for this simulation.

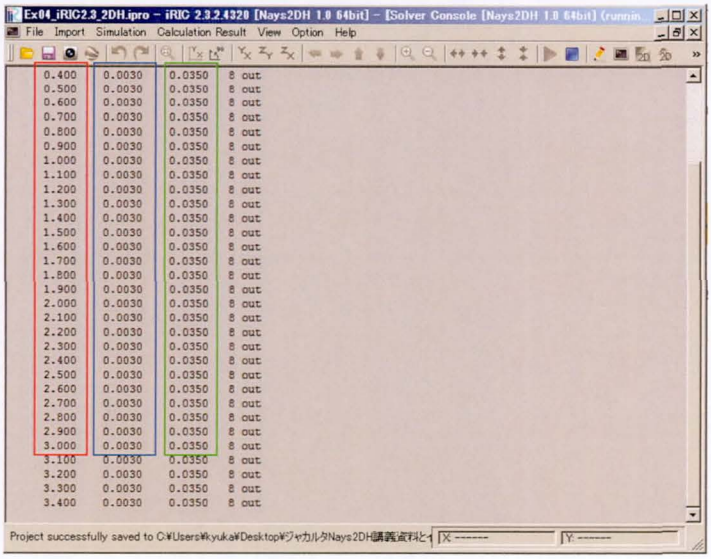
After making the settings above, click on [Save and Close] to close the window.

5. Making a simulation

Before starting the simulation, select [File] - [Select As File (*.ipro)] and save your project. The File extension [*.ipro] means a compressed file. If there is possibility that your project file will be larger than 2G bytes after your calculation finish, select [Save As File….] instead of [Select As File (*.ipro)].

For running a simulation, select [Simulation] -> [Run] in menu bar or select ▶ on the menu tool bar. Then, Solver console window will appear and calculation starts.

Solver Console Window:

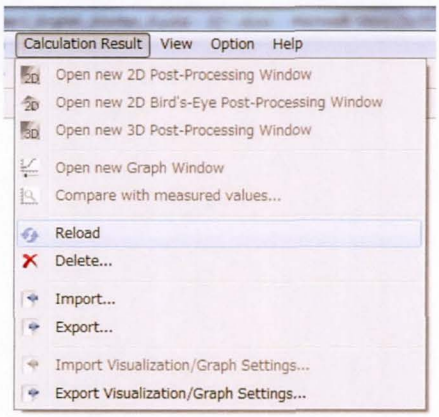


About the output values of the solver console:

- The red box shows the times taken for calculation (sec.),
- The blue box shows the calculated discharges (m^3/s)
- The green box shows the water elevations at the downstream end (m).
- The number of "8" shows iteration time
- Out denotes that the solver is writing the result to the project file.

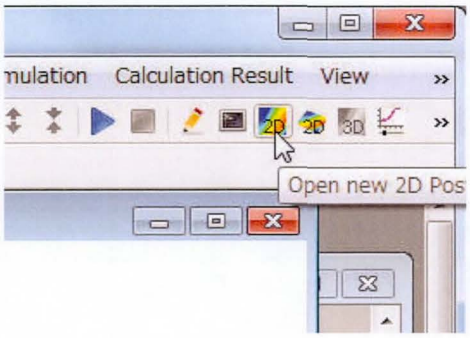
The stop icon for the calculation is [Simulation] - [Stop] on the menu bar.

If you want to visualize your calculation results before the calculation finish, Select [calculation result] – [Reload] on the menu bar.

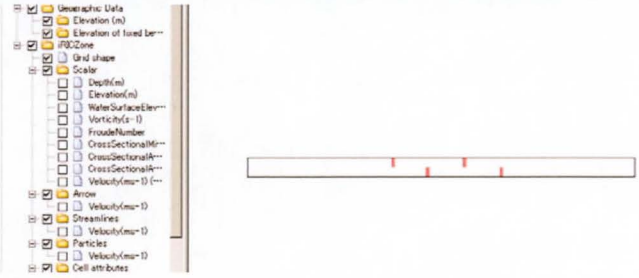


6. Visualization of calculation results

Select [calculation result] -> [Open new 2D post-processing window] on the menu bar, or click on the button [2D] as shown in the following figure. Then [2D Post-processing Window] will open.

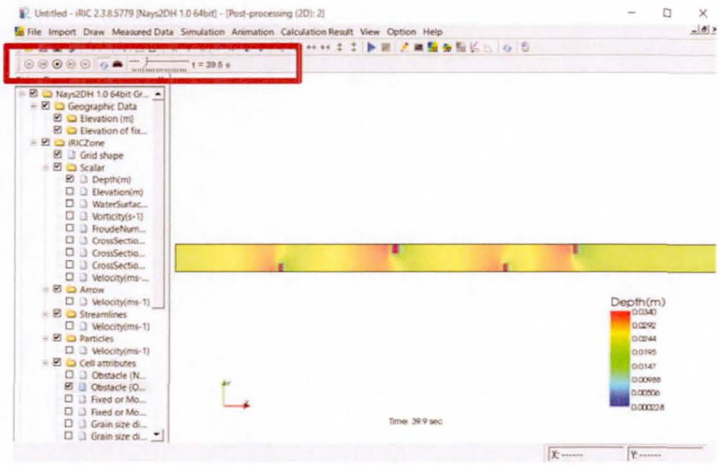


In the Object Browser, select [Nays2DH 1.0 64 (32) bit Grids] - [iRICZone] - [Cell Attributes] - [Obstacle (Obstacle)] and make a check mark ☑ in these boxes. In the Object Browser, select [iRIC Zone] – [Velocity (ms-1)], and then contour map of velocity will appear.



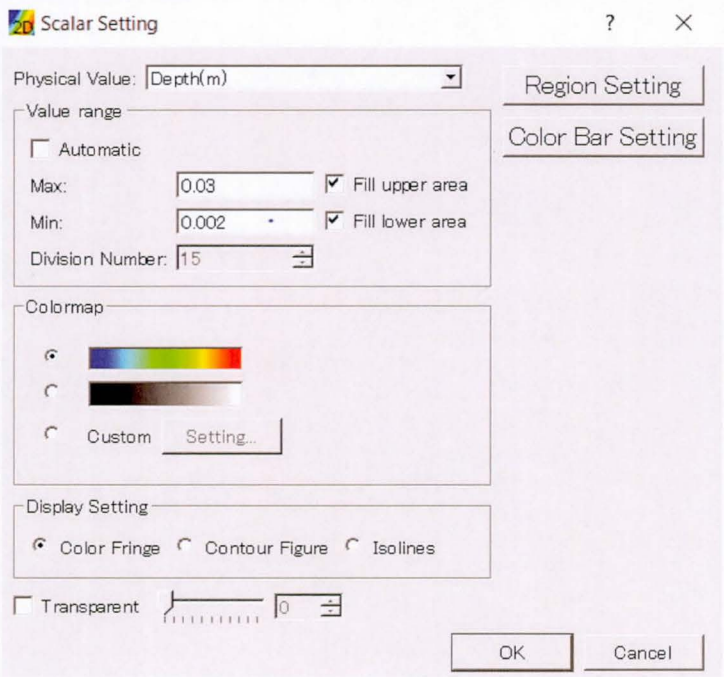
- The obstacles will be indicated by red lines.

In the Object Browser, make a check mark ☑ in [Nays2DH 1.0 xxbit Grids] - [iRICZone] - [Scalar] - [Depth (m)]. Set the time controller arrow icon, which shown in the red box in the figure, set t=0 (t).



- t = 0 (t)
- Click on the start icon of the time controller on the toolbar.
- The animation will start.

To fix the value range of the scalar bar on the depth contour map, right click on [Scalar] and open [Property]. In the property window, make the following settings and click on [OK]. Then, scalar bar will be fixed during the animation.

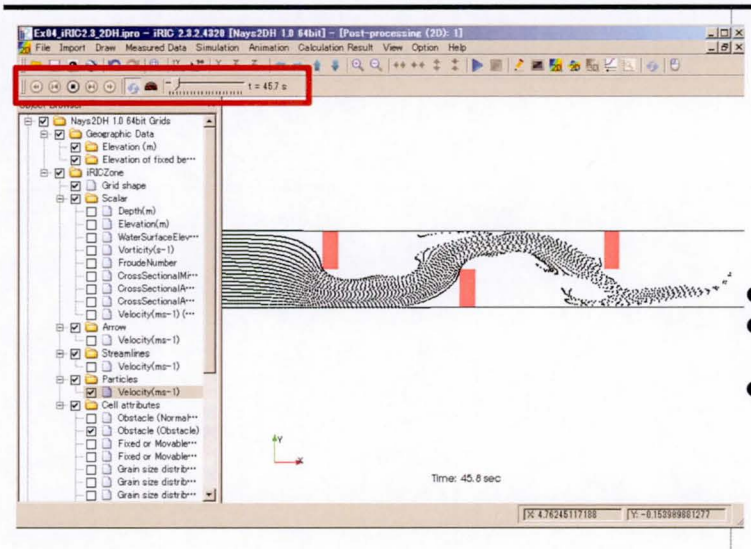


- Remove the check----- Automatic
- Max: ---- no change

  - Min------ 0.03 (m)
  - Max----- 0.002 (m)

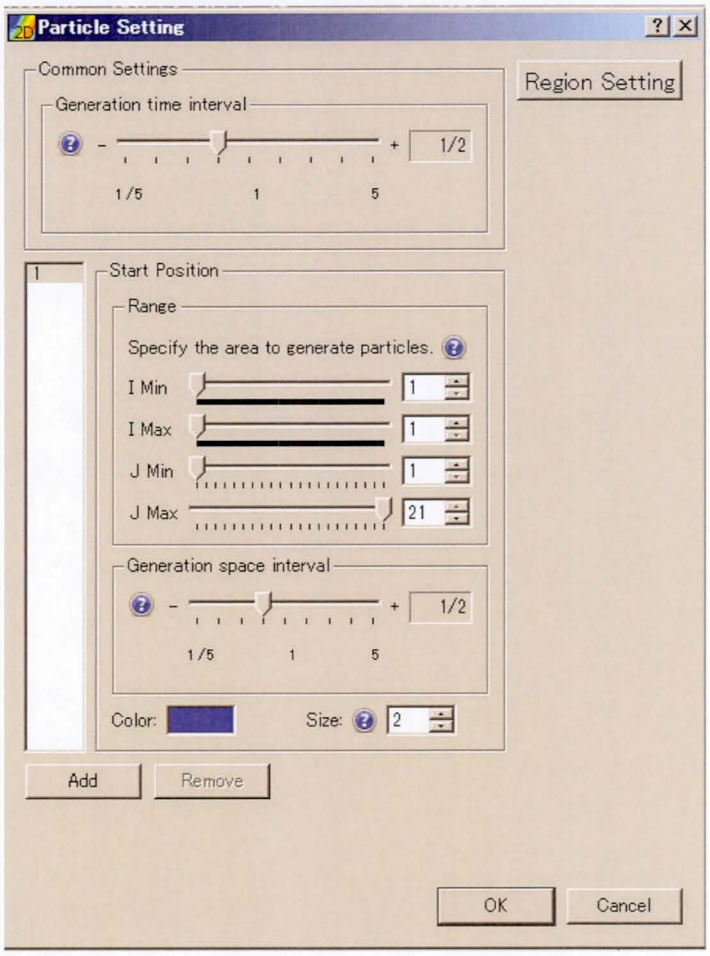
- Generation space interval---- 1/2
- Remove the check----- Transparent

In the Object Browser, select [Nays2DH 1.0 xxbit Grids] - [iRICZone] - [Particles] - [Velocity (ms-1)] and make a check mark ☑ in these boxes. [•] will appear at the upstream end of the simulation.
Set the time controller arrow icon, which is shown in the red box in the figure, set t=0 (t).



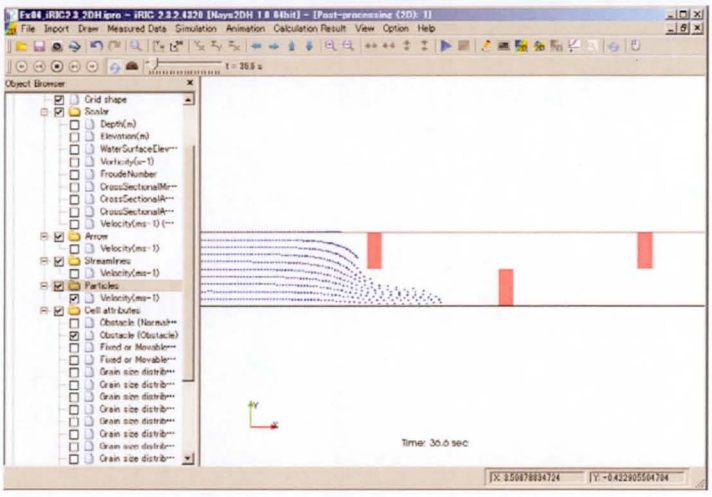
- t = 0 (t)
- Click on the start icon of the time controller on the toolbar.
- The animation will start.

If you want to edit the start position, the generation interval or the size of particles, select [Nays2DH 1.0 xxbit Grids] - [iRICZone] - [Particles] in the Object Browser, and right click on [Particles] to open [Property]. Then, in the [Particle Setting] window, make the following settings and click [OK].

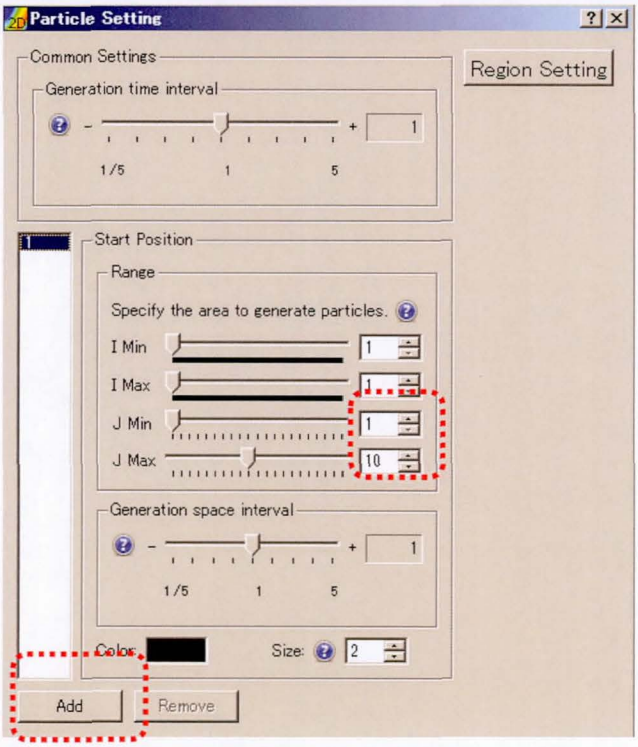


- Generation time interval----- 1/2
- Start Position---- no change
- Generation space interval---- 1/2
- Color---- blue
- Size---- 2

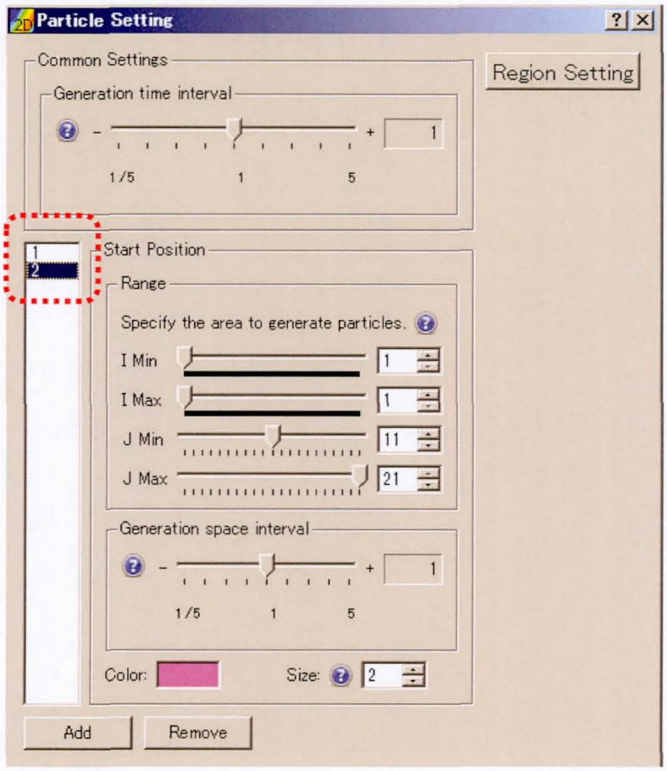
The color of animation is changed.



This is another example. In the [Particle Setting] window, make the following settings and click on [OK].



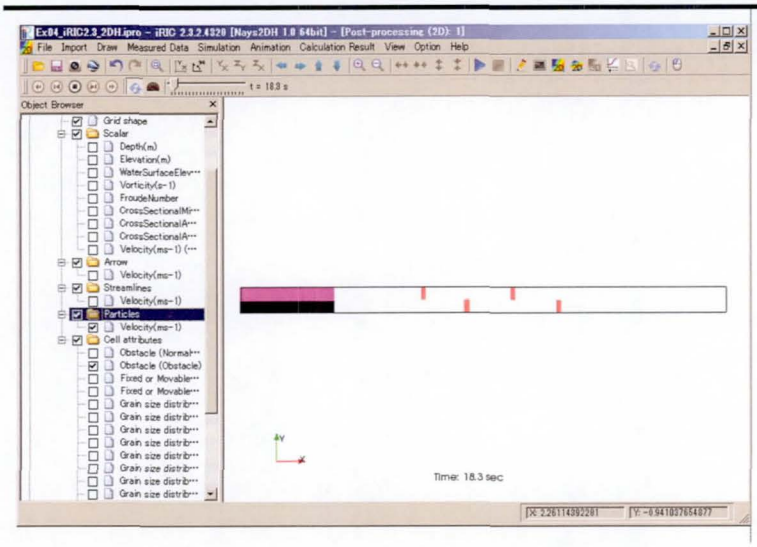
- Generation interval----- 1
- Start Position
  - J Min------ 1
  - J Max----- 10.
- Generation space interval--- 1
- Color----- black
- Click on [Add]



After click on [Add], layer 2 will appear as left figure (red box).

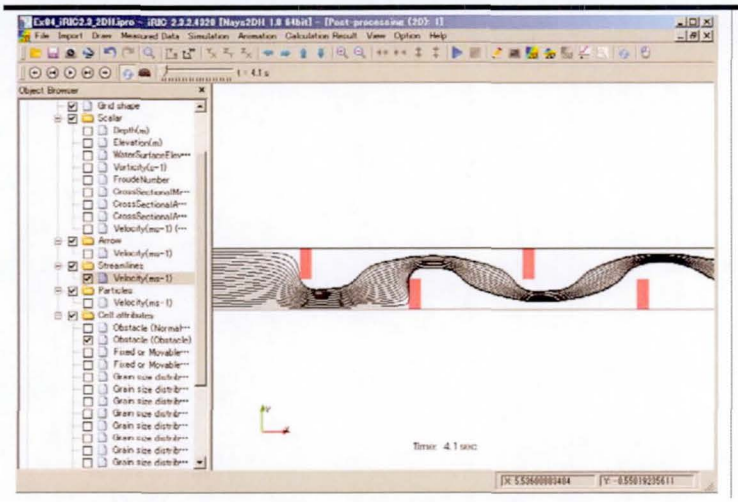
- Start Position
  - J Min----- 11
  - J Max----- 21.
- Color----- pink

The color of animation is changed and the animation is easier to see after it is restarted.



Visualizing with streamline:

In the Object Browser, select [Nays2DH 1.0 xxbit Grids] - [iRICZone] - [Streamline] - [Velocity] and make a check mark ☑ in the box. Then start animation.



End Tutorial 1.

Tutorial 2.

Calculation of flow and morphological change of river bed in a meandering channel

● Purpose of this tutorial

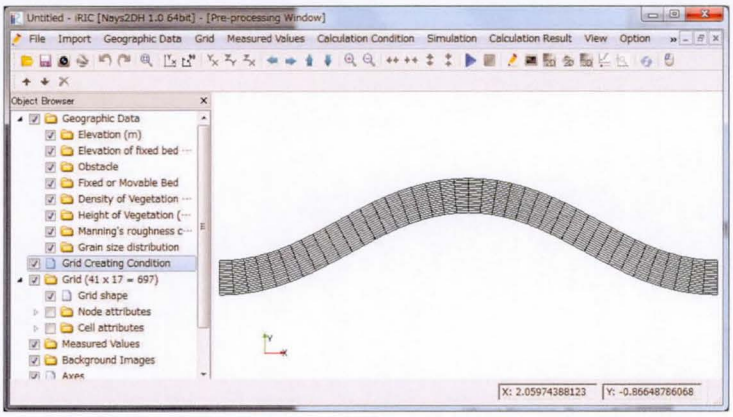
Simulate the flow and bed morphological change in a meandering channel with simple bed geometry to understand the fundamental bed evolution phenomena in a meandering channel and also the basic operation of Nays2DH in iRIC.

1. Generate a computational grid

Generate a computational grid of a meandering channel with a rectangular cross section by using [Multifunction Grid Generator] attached in iRIC. Fist, click Menu bar -> [Grid] -> [Select Algorithm to Create Grid], and select [Multifunction Grid Generator] in the dialog of [select grid creating algorithm]. After the dialog for grid generation is opened, set the various conditions for generating the computational grid as follows;

[Channel Shape]----- Select channel shape: Sine-generated curve
[Cross Sectional Shape Parameters]----- Width: 0.3m, Number of Grid in Lateral Direction: 16
[Channel Shape Parameters]----- Wave Length of Meander: 4.7m, Meander Angle: 28.6 degree,
Number of Grids in One Wave Length: 40
[Bed and Channel Shape]----- Channel Slope: 0.004
The default values are used for the other parameters.

The computational grid will be generated as follows;

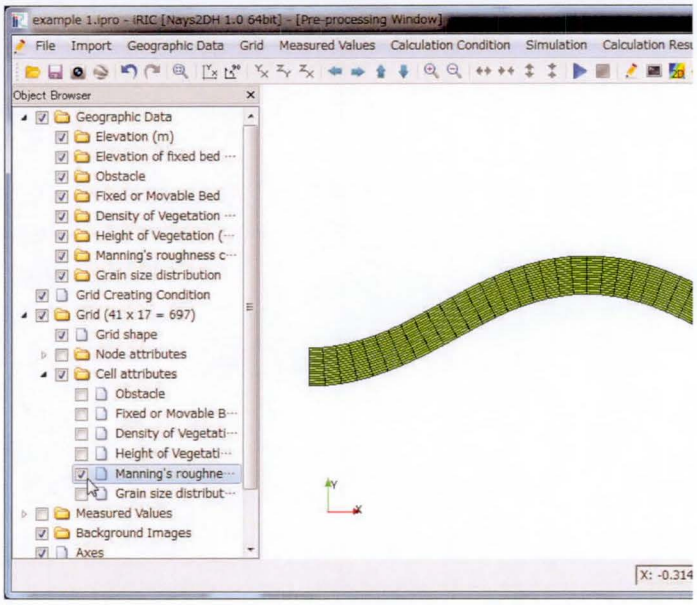


The computational grid will appear on the screen.

2. Set the cell conditions

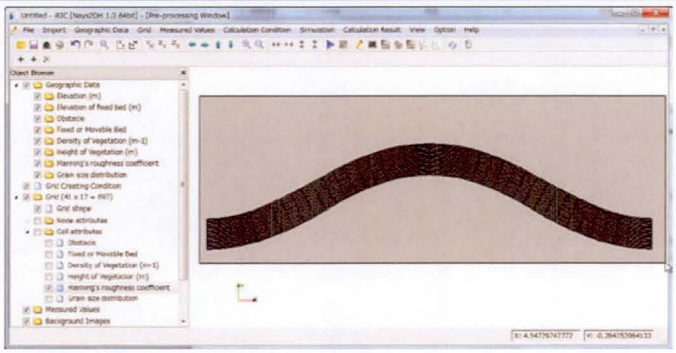
The cell conditions can be set on the generated computational grid. Here, you set the [Manning's roughness coefficient].

Click on [Cell attributes] in the object browser, and change the status of cell attributes to ☑. You can open the item of cell attributes by clicking on ▷ bottom on the left side of [Cell attributes], and then click on [Manning's roughness coefficient].



- The left side of the figure is object browser, and the right side of the figure is the visualization screen.
- When you want to change something such as geographic data, cell attribute, etc. in the visualization screen, you need to click on the related item in the object browser. When you click on that item, the background color of the item changes to light blue. Such status is defined as '**active**' here.
- While you keep the status of [Manning's roughness coefficient] as ☑, the contour on the grid indicates the value of [Manning's roughness coefficient].

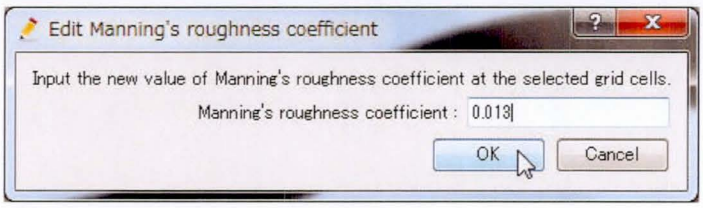
Select all of computational grid by operating your mouse, and then right-click on the selected region. Then select [Edit value] on appeared menu.



- You can directly change the value of cell attributes of each cell.
- Hold Shift key and select some cells. You can select a group of cells, even if they are not directly connected.

On the dialog [Edit Manning's roughness coefficient], you can specify the value of this coefficient in the region you selected.

By this procedure, Manning's roughness coefficient is set to all of cells in the grid.

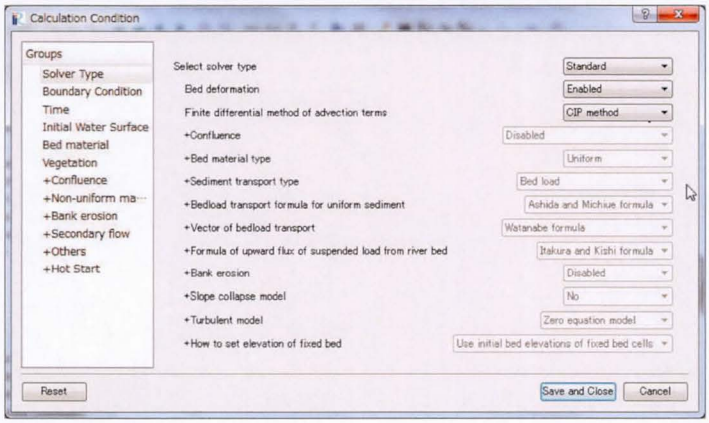


- Manning's roughness coefficient : 0.013

3. Set the computational conditions

Select [Calculation conditions] -> [Setting] on menu bar and set calculation conditions as followings. The default values are used for the other parameters.

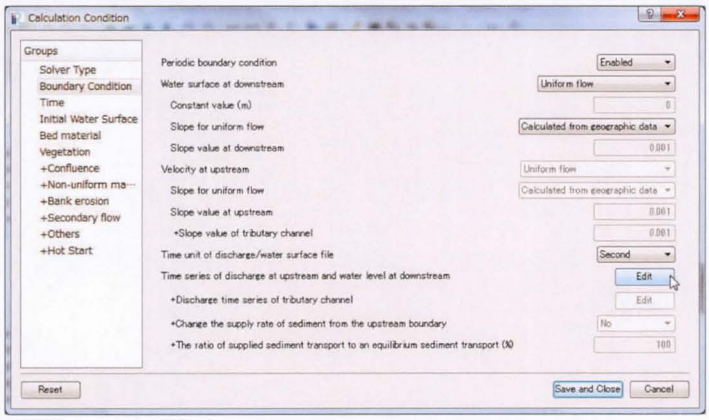
Solver Type:



- Select solver type---- Standard
  - Bed deformation---- Enabled

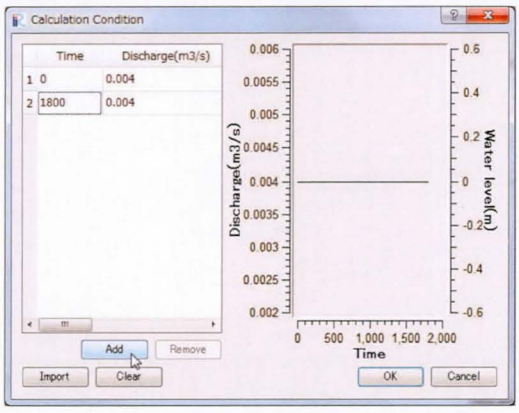
Note) You can select the items with mark [+], in case that you select [Advanced] in [Select Solver Type].

Boundary Condition:



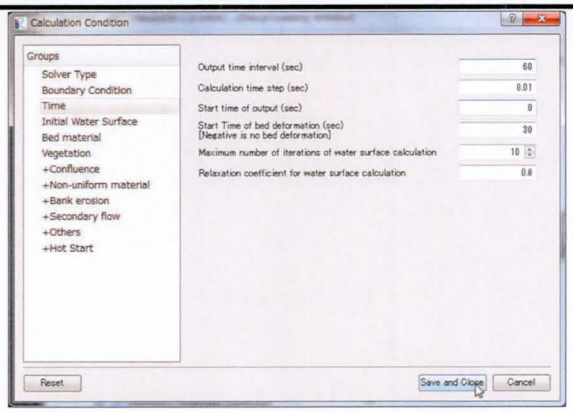
- Boundary conditions

  - Periodic boundary condition : Enabled
  - Time unit of discharge/water surface file : Second



- Set the discharge of upstream end
  - Select [Edit] and input Time and Discharge. Time from 0 to 1800 (sec), and a constant discharge of 0.004 m^3/s, is given in this example.

Time:

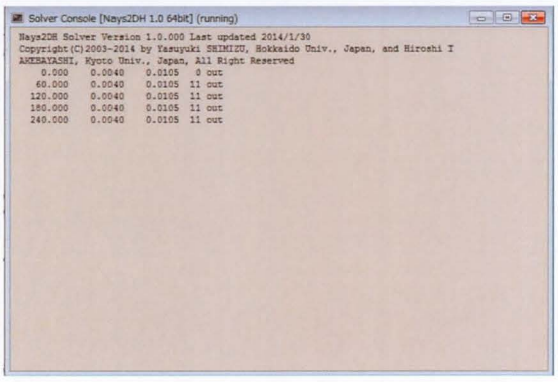


- Time Conditions:
  - Output time interval : 60 sec
  - Calculation time step : 0.01 sec
  - Start time of output : 0 sec

4. Run a simulation

For running a simulation, select [Simulation] -> [Run] in menu bar or select ▶ on menu tool bar. Then, Solver console window will appear and calculation starts.

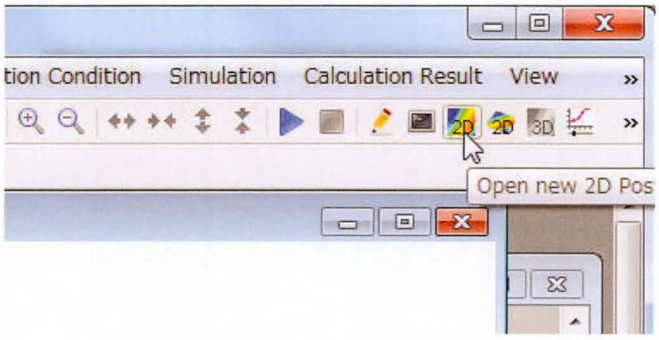
Solver console :



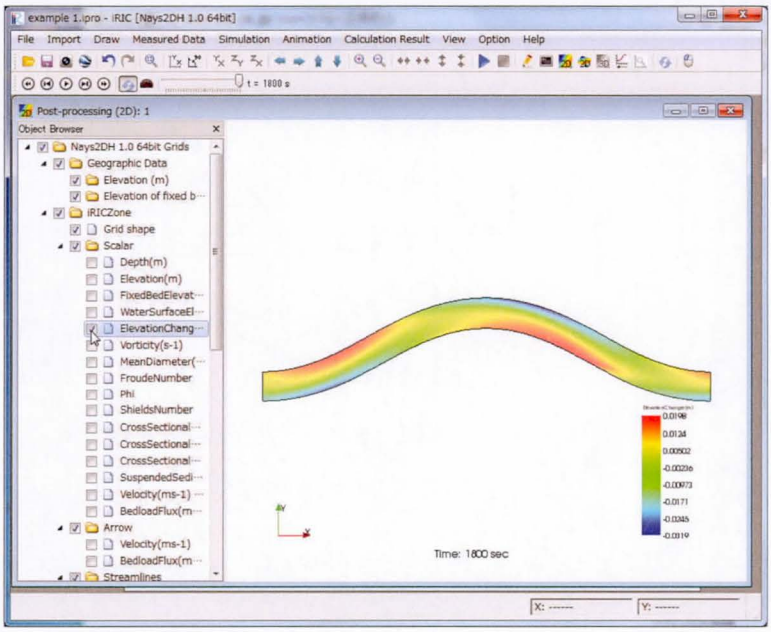
- The numbers shown in solver console are time (sec), discharge (m3/s) and downstream water level (m) (from left to right).
- The [out] shown in right of numbers means that the iRIC output the result to the file to be visualized.

5. Visualize the results

Select [calculation result] -> [Open new 2D post-processing window] in menu bar, or click on the button shown in the following figure;

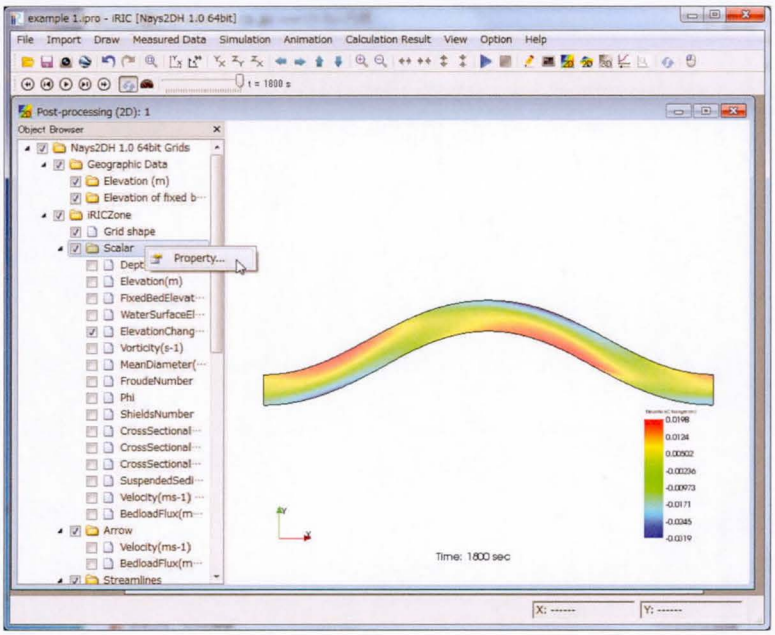


You can visualize the item listed in the object browser to the visualization screen. Firstly, for visualizing the morphological change of bed, check the box of [ElevationChange] under [Scalar].

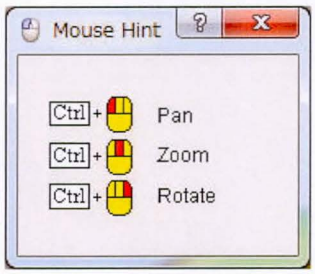


- [Scalar]: show a contour map of selected scalar data.
- [Arrow]: show a vector of selected vector data.
- [Streamlines]: draw streamlines based on the selected vector data.
- [Particles] : visualize particle movement by selected vector data.
- [Cell attributes]: show the cell attributes you set in the cell conditions.
- [Measured values]: You can read the measured data such as experimental data.

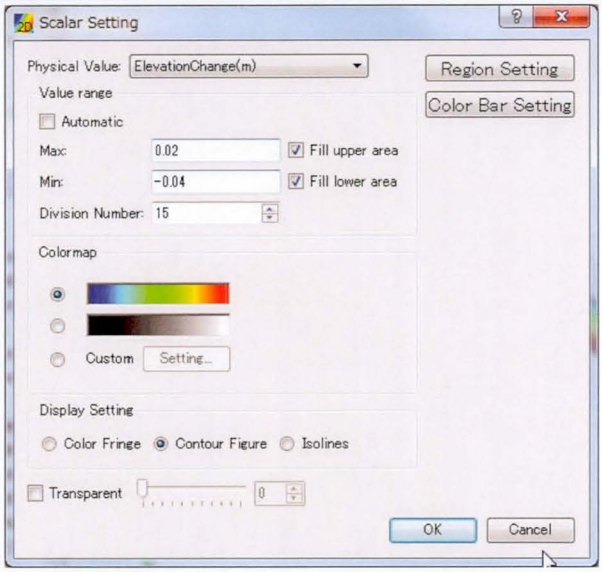
For changing the range of color legend, right-click on [Scalar] and select [Property].

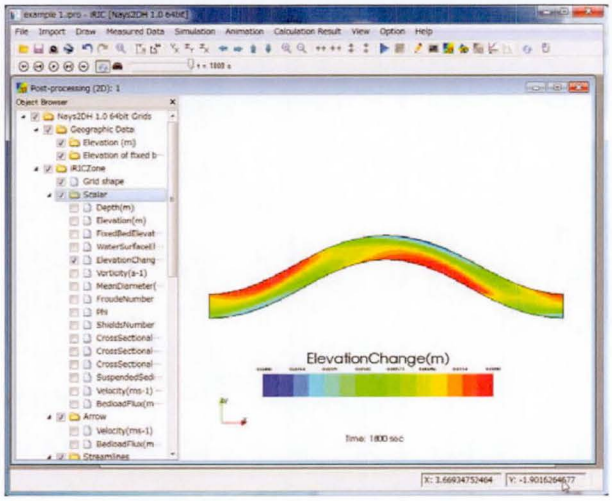


- You can change the location of the color legend by dragging it. (You need to click [Scalar] to make the status of [scalar] 'active'.)
- You can pan, zoom and rotate the visualization screen by mouse operations shown in following figure:



In this dialog, you can set the maximum/minimum value for the legend colors range, the number of labels for the color legend and etc.



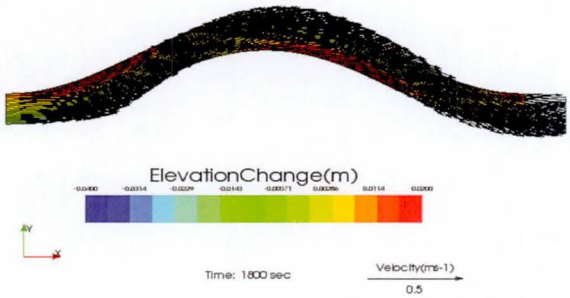


- Deselect [Automatic] box
- Max : 0.02 (m)
- Min : -0.04 (m)
- Display setting : Contour Figure
- Deselect [Transparent] box

After set the conditions above, the contour map will be clear and point bar is clearly observed a on the inner bend with the erosion in the outer bend.

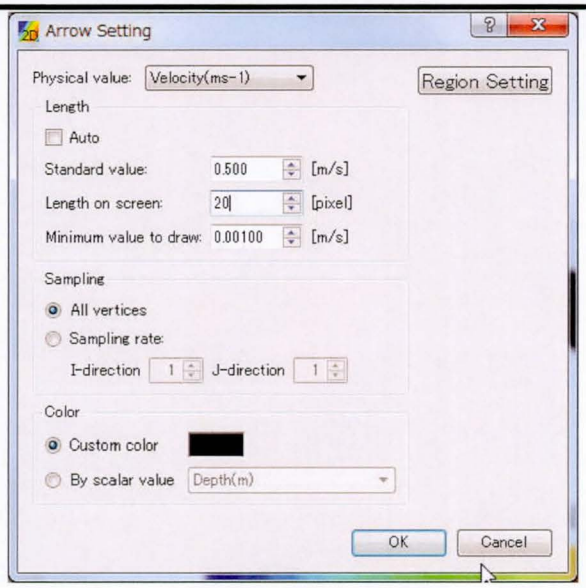
Note) The default setting of display setting is [color Fringe]. But, a color contour will be much clear by selecting [Contour Figure]. In this case, the visualization will have higher quality but the load for visualization will be larger.

In the next step, the flow field is visualized by vectors. Click on the box of [Velocity] under [Arrow] in the object browser.

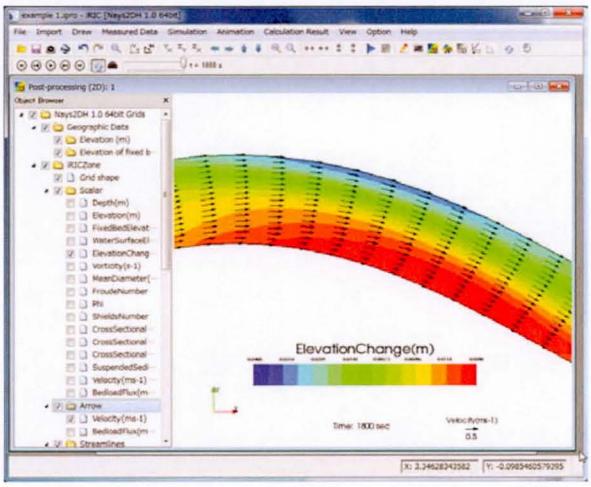


The flow in the computation is visualized by vectors. However, the length of vectors is not suitable for the visualization.

For setting the properties of vectors, right-click on [Arrow], and select [Property] on the appeared pop-up menu. The dialog for setting the vector will open.

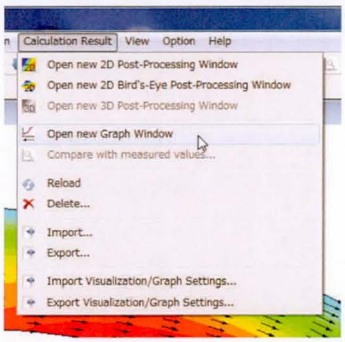


- deselect [Auto] check box
- Length on screen : 20 pixel

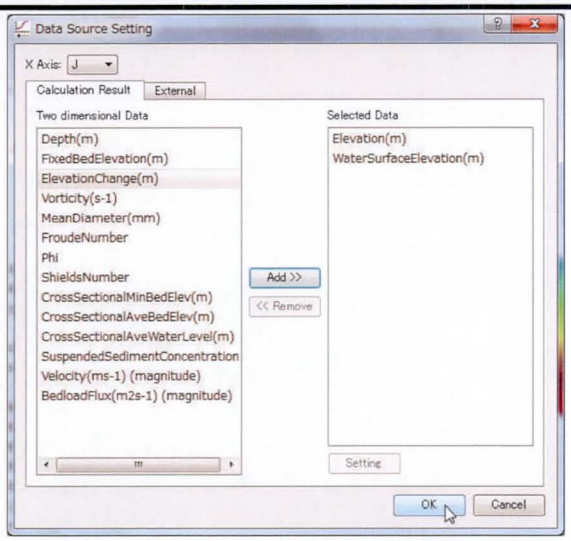


By adjusting the length of vectors, you now can see the flow field in the meandering channel.

One dimensional graphic function is used for visualizing the water level and bed elevation in a cross sections. Select [Calculation Result] -> [Open new graphic window].

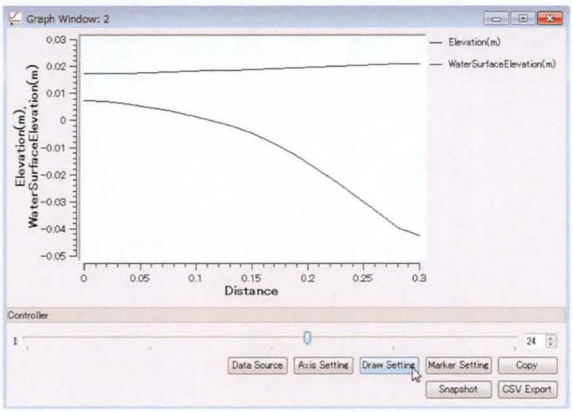


The dialog for setting the data of 1D graph will open. Select X Axis as [J], and add [Elevation] and [WaterSurfaceElevation] from [Two dimensional data] to [Selected data].



- X Axis : J
- Two dimensional Data : Select [Elevation] , [WaterSurfaceElevation] and click on [Add].

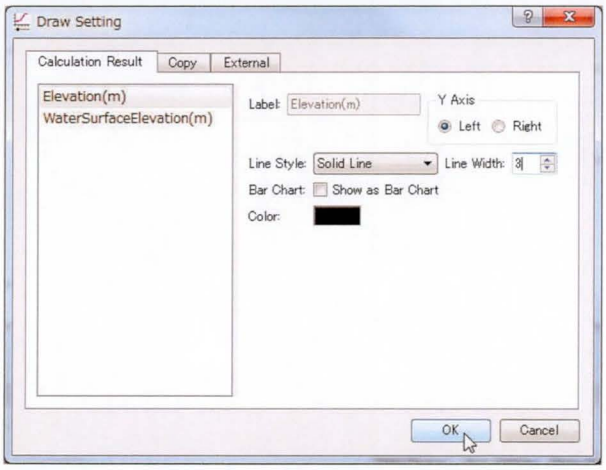
Then, The water surface and bed in a cross section are visualized. Click on [Draw Setting] for changing the visualization properties of this graph.



By moving the location of I on Controller, you can see the 1D graph in a different cross sections.

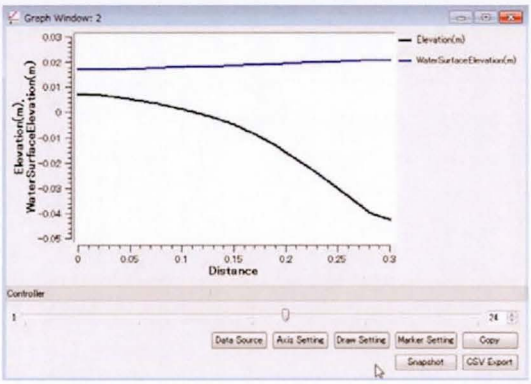
Note) I and J mean the index of longitudinal and transverse direction in the grid.

You can change the style width and color of lines.

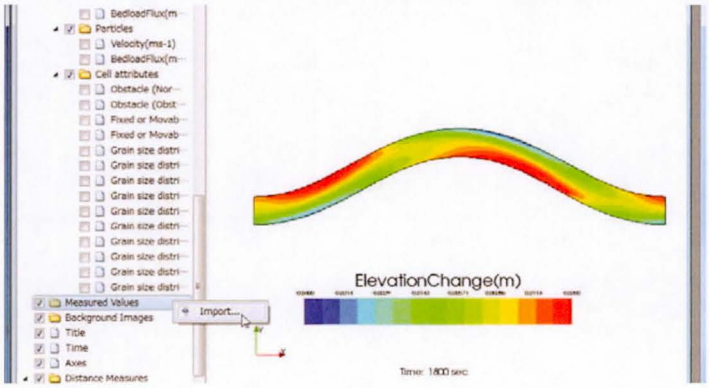


- Setting for Elevation(m)
  - Line style : Solid Line
  - Line Width : 3
  - Color : Black
- Setting for WaterSurfaceElevation(m)
  - Line style : Solid Line
  - Line Width : 3
  - Color : Blue

You can visually understand the line of water surface and river bed.

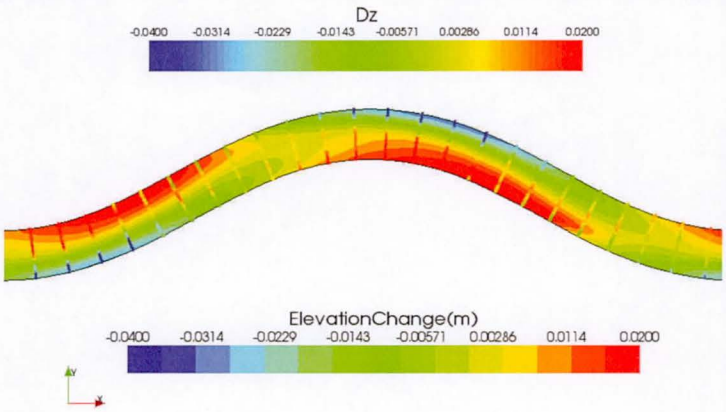


As the last step, simulated bed elevation change is compared with the experimental results. Right-click on [Measured Values], and select [Import] on the pop-up menu.



- "measured.txt" is specified as a [Measured values].

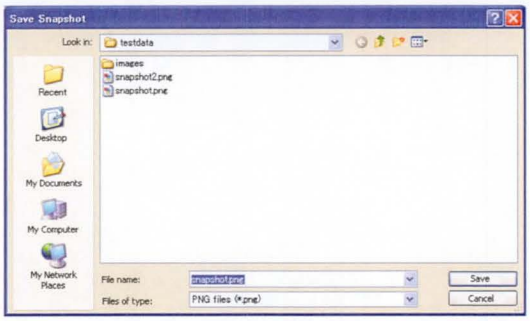
The measured data will be imported. The range of color legend in the simulated result and measured value is set to be same. You can change the range of color legend by right-clicking [Scalar] under [Measured Values] and selecting [property] on appeared pop-up menu.



- Nays2DH can reproduce the trend of morphological change of bed in the experiment.

Saving the image file of the results:

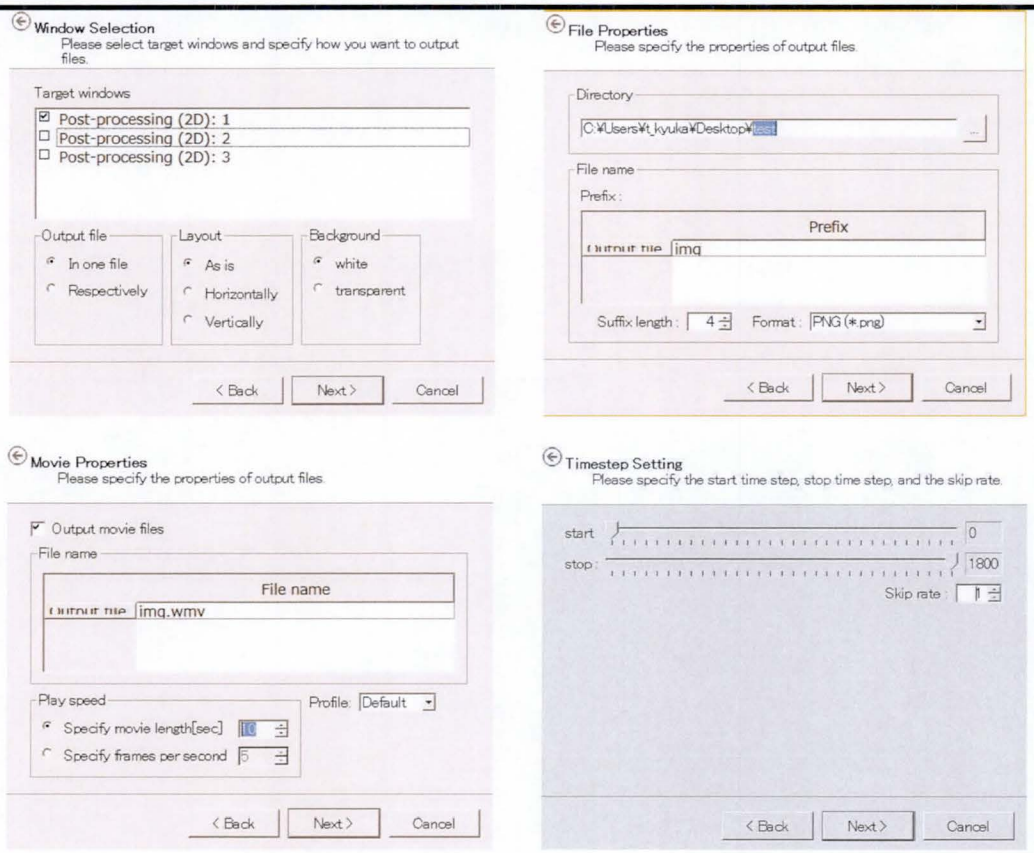
Select [File] -> [Save Snapshot] on Toolbar, then [Save Snapshot] dialog will open. Input the file name and click on [Save].



Saving the animation file of the results:

Select [File] -> [Continuous snap shot/ Movie/ Google earth export...] on Toolbar. ① Select target window which you want to export as an animation file. ② Select the file folder from the dialog. ③ Check output movie files and set specify movie length [sec]. ④ Set start and stop time of the movie, and set skip rate

.



End Tutorial 2.

Tutorial 3.

Flow and Morphodynamics Simulation for an Actual River

- Purpose of this tutorial

Simulate flow and morphology changes in an Actual River by using Nays2DH, and verify the simulation results.

- Outline:

Creating the calculation grid

Using cross-sectional river survey data of an actual river, create calculation grids: 11 division points in the transverse direction, and 216 division points in the longitudinal direction.

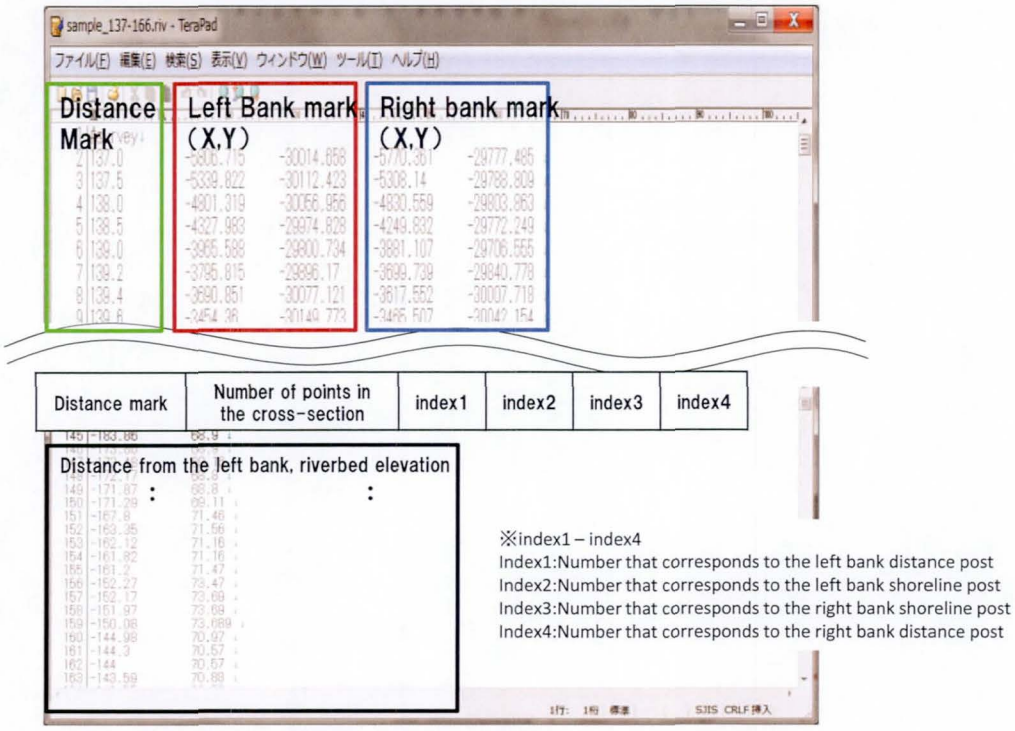
Setting the calculation conditions

Set simulation conditions for a steady flow discharge of 300 m3/s for 1 hour or 3,600 sec. Set various other conditions necessary for simulation.

Making a simulations

Visualizing the calculation results

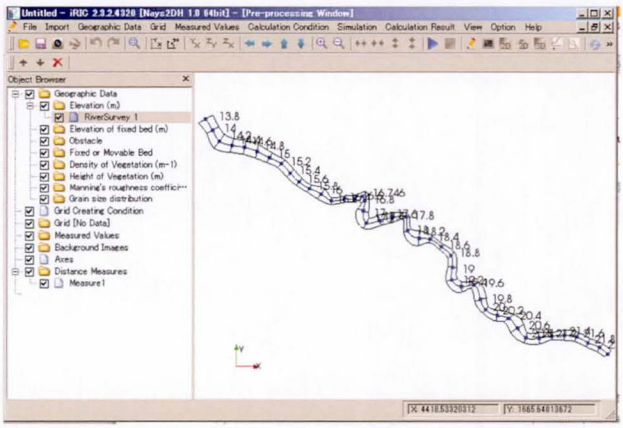
- Format of the river survey data:



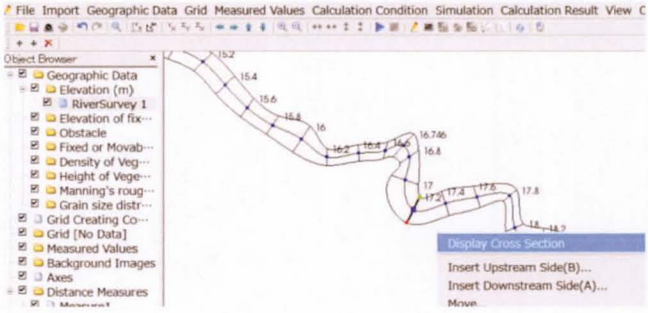
1. Generate a computational grid

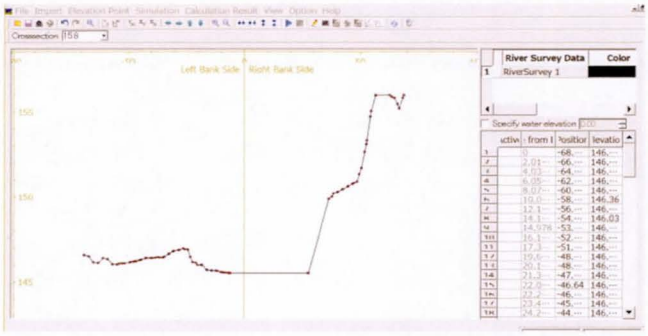
Import the elevation data (cross-section river survey data) of the target river, and then generate a computational grid by using one of the grid generators called [Create Grids from River survey data] attached in iRIC.

On the menu bar, select [Import] -> [Geographical Data] -> [Elevation], and then open the sample data named [ori.riv]. After the data is imported successfully, the shape of the river as below will appear on the pre-processor window of iRIC. The number of each cross-section show the distance from the river mouse.



To confirm the topography data, select [Geographic Data] -> [Elevation] -> [River Survey 1] in the Object Browser, and right click on a cross sectional line where you want to confirm the bed topography. Then, select [Display Cross Section] as below.



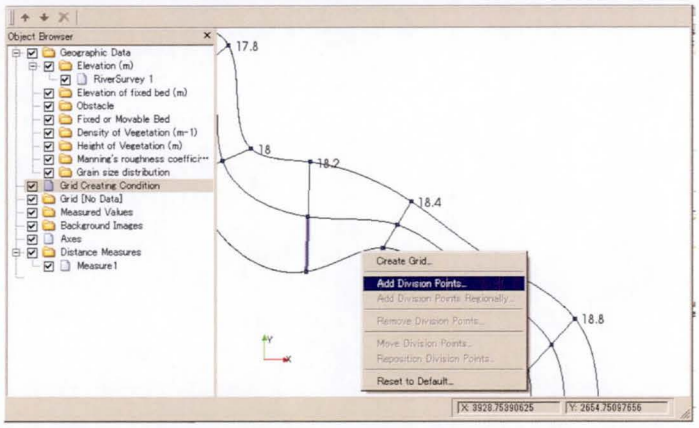


Cross Sectional data

To create calculation grid, select Menu bar -> [Grid] -> [Select Algorithm to Create Grid], and select [Create Grids from River survey data] in the dialog of [select grid creating algorithm]. The procedure to create calculation grid is as follows.

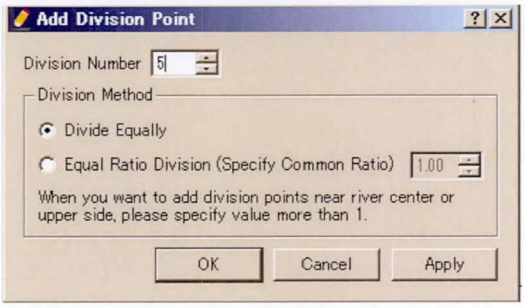
① Setting the division points in the transverse direction:

Each river "transverse line" is divided into two sections. Click on a transverse line anywhere, so change the color and width of the selected line. Then, right click on the selected line and select [Add Division Points] from the right click menu.



By using the mouse, you can move the grid location, and also zoom in and zoom out the grid.

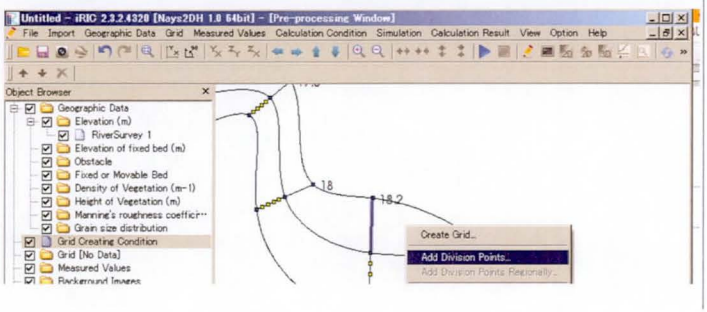
After click on [Add Division Points] from the right click menu, [Add Division Points] window will appear. In this window, input the [Division Number] and [Division Method] as follows.



- Division Number (example) ---- 5
- Division Method----- Divide Equally

Division points (yellow circles) will appear as figure below.

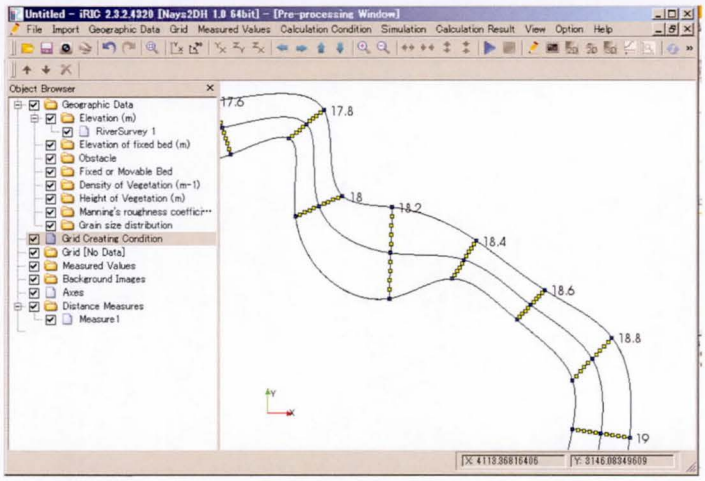
Then, click on the other section of the transverse line and make division points in the same way.



Set same division number for the other section :

- Division Number (example) ---- 5
- Division Method----- Divide Equally

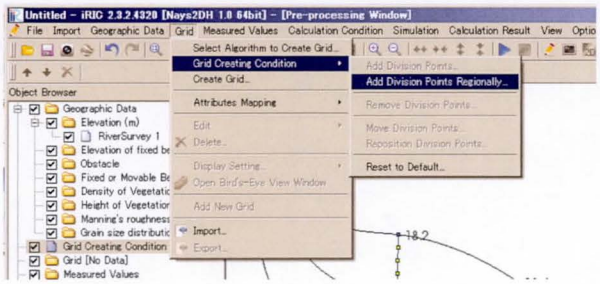
Division points are displayed as follows.

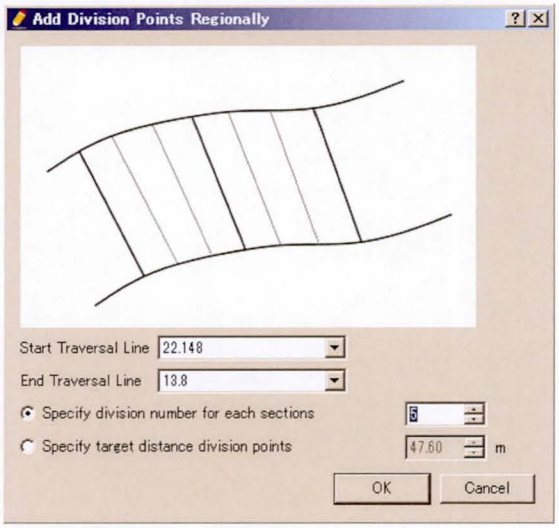


The number of division points in the transverse direction is 11 (i.e.,10 transverse sections).

② Setting the division points in the longitudinal direction:

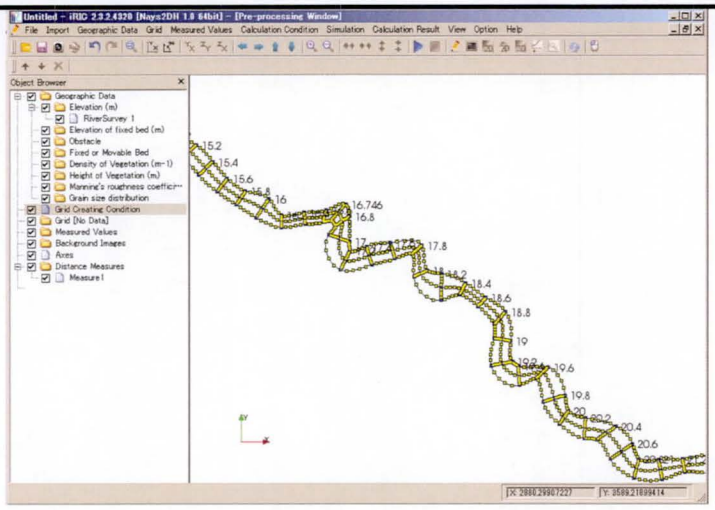
On the menu bar, select [Grid] - [Grid Creating Condition] - [Add Division Points Regionally]. Then, [Add Division Points Regionally] window will appear. In the window, make the following settings.





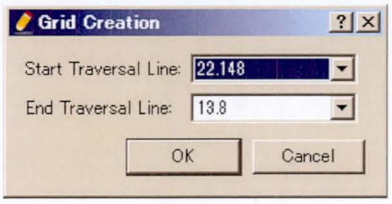
The river section is divided by transverse lines into 5 longitudinal reaches.

Division points are displayed as follows.



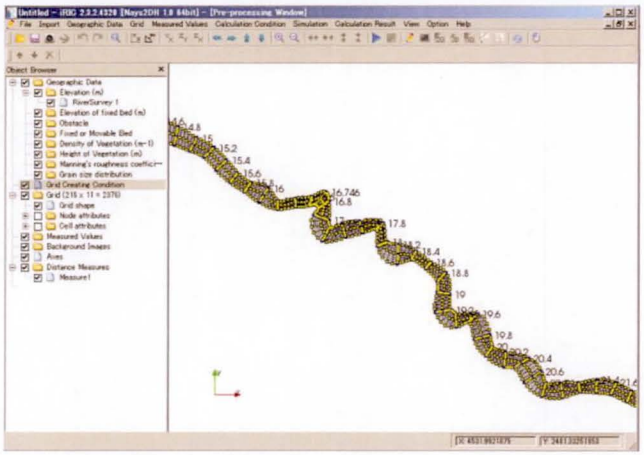
③ Create grid:

On the menu bar, select [Grid] - [Create Grid], then [Grid Creation] window will appear as below. Click on [OK] here.



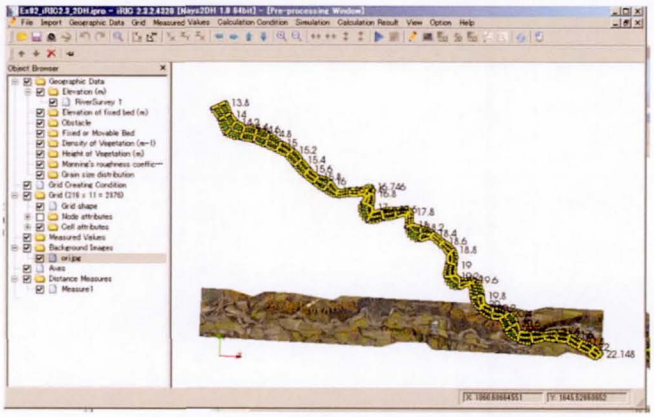
The elevations of cross-sectional river survey data are converted into an elevation for each grid point.

When division points and grids for calculation are displayed as follows, the creation of grids for calculation is completed. Check [Node attributes] -> [Elevation] in Object Browser, so you can confirm the elevation contour map of the calculation grid.

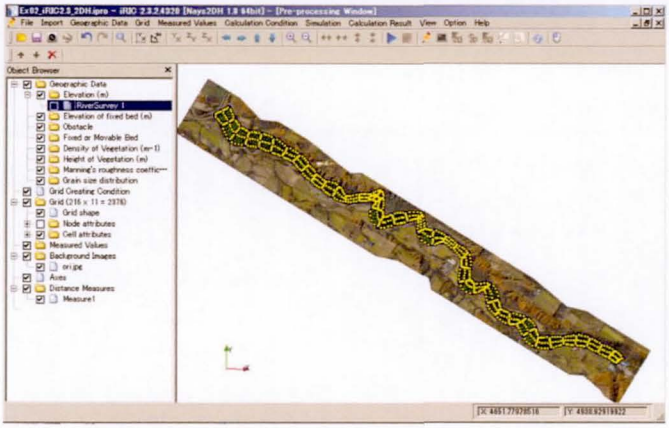


2. Background Image File:

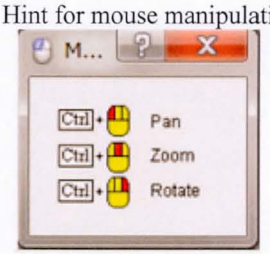
To import background image file, select [Image] - [Background image] on the menu bar and open [ori.jpg] file. Background image will be imported as below.



Moving, zooming and rotating manipulations are possible using the mouse (see the following hint) to match the location between calculation grid and back ground image file.



Hint for mouse manipulations.



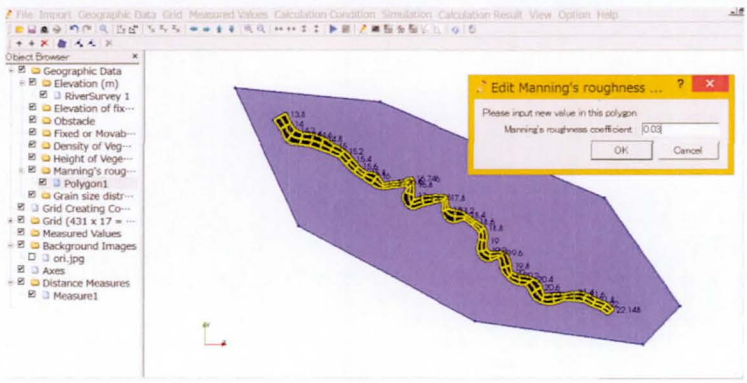
Alt+ Scaling

3. Setting the calculation conditions

Set Manning Manning's roughness coefficient and calculation conditions.

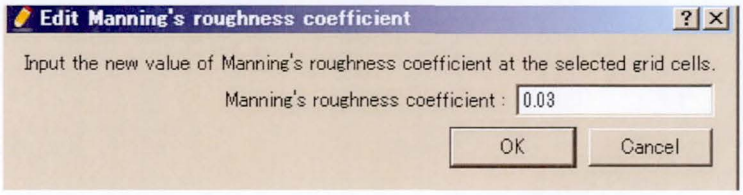
① Manning's roughness coefficient:

Right click on [Geographical Data] -> [Manning's roughness coefficient] in the Object Browser and select add [Polygon]. Make a polygon where you want to set Manning's Roughness coefficient by your PC mouse.



This example selects the entire area of the computational grid.

In the window, input Manning's roughness coefficient.



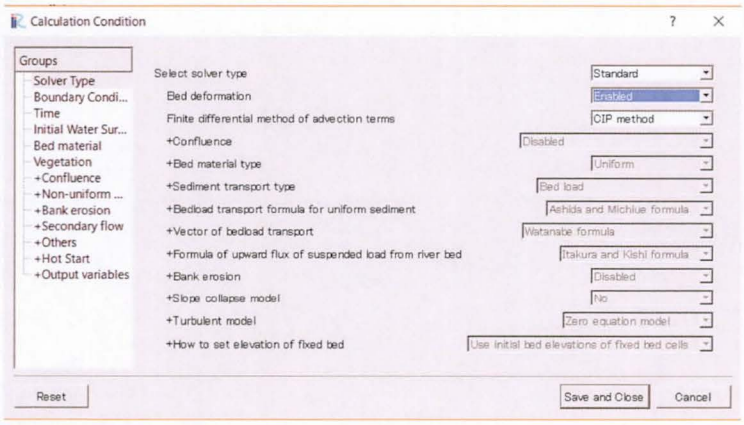
- Manning's roughness coefficient ---- 0.03

The value has to be set greater than zero.

② Calculation conditions:

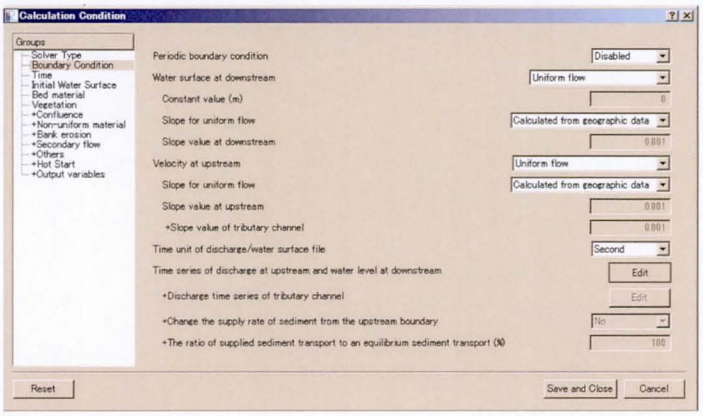
Select [Calculation Conditions] - [Setting] on the menu bar, then set various conditions as followings

Solver Type:

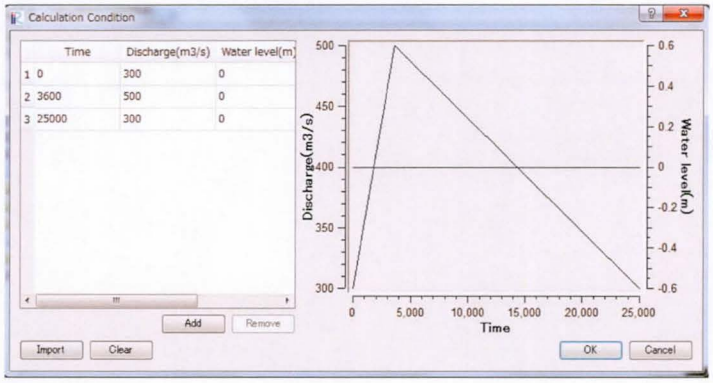


- Select solver type---- Standard
- Bed deformation---- Enabled
- Finite difference method of advection terms----- CIP method

Boundary Condition:



- Periodic boundary condition ---- Disabled
- Water surface at downstream ---- Uniform flow
- Velocity at upstream ---- Uniform flow
- Time--- Second
- Click on [Edit] to set discharge

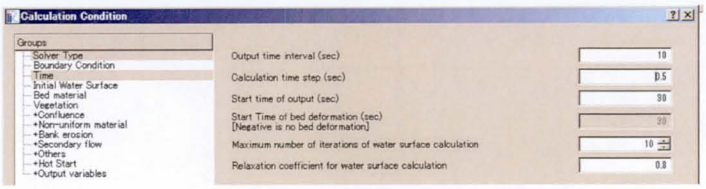


Input the discharge

- Time---- 0 sec, 3600 sec, 18000 sec
- Discharge---- 300, 500, 300 (m3/s)

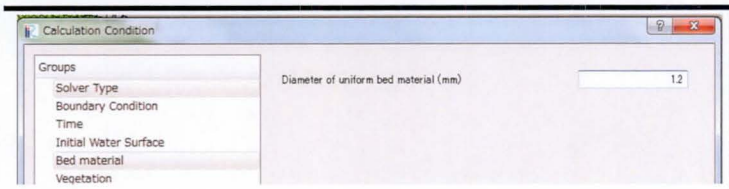
Note: no need to input the water level in this dialog, because this case uses [Uniform flow] for the boundary condition at downstream end.

Time:



- Output time interval (sec)----60
- Calculation time step (sec) ---- 0.2
- Start time of output (sec) ---- 0

Bed material:



- Diameter of uniform bed material (mm) ---- 1.2 mm

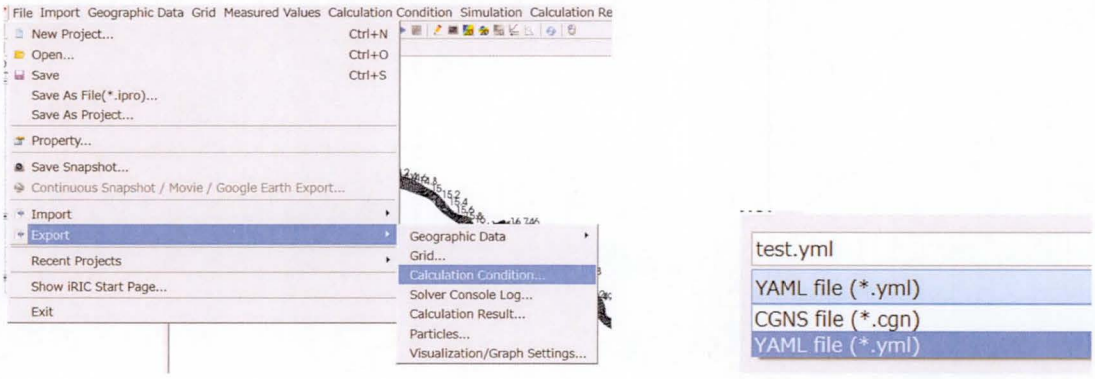
Description of the uniform bed material setting

Item	Description	Remarks
1	Diameter of uniform bed material (mm)	Enter a bed material grain size to be used for bed deformation calculation.

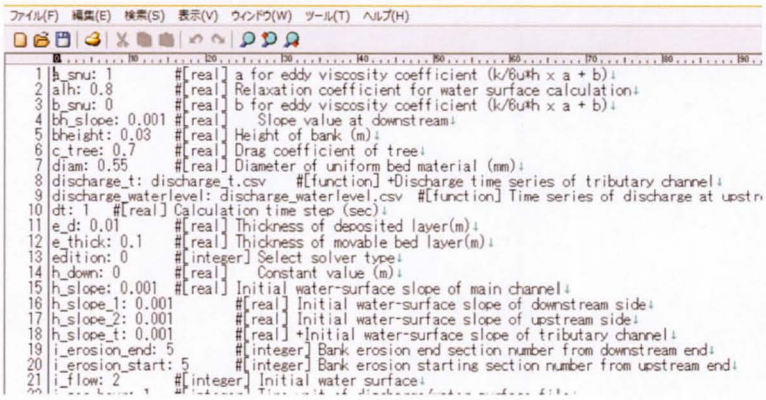
Many other conditions can be set; however, they do not need to be set for this simulation.
After making the settings above, click on [Save and Close] to close the window.

③ Export the setting calculation conditions as a text format:

If you use iRIC version after iRIC 2.3.9 6024, you can export your setting calculation conditions as a text format. In toolbar, select [File] -> [Export] -> [Calculation conditions], and save your file as "**.yml" extension as below.



Change the file extension from "yml" to "txt", and then open the "**.txt" file. Calculation conditions can be confirmed as below. In addition, if you change the file extension from "txt" to "yml" again, the setting calculation conditions can be imported into Nays2DH in iRIC.



```
a_snu: 1          #[real] a for eddy viscosity coefficient (k/6uth x a + b)
alh: 0.8          #[real] Relaxation coefficient for water surface calculation
b_snu: 0          #[real] b for eddy viscosity coefficient (k/6uth x a + b)
bh_slope: 0.001   #[real]    Slope value at downstream
bheight: 0.03     #[real] Height of bank (m)
c_tree: 0.7       #[real] Drag coefficient of tree
diam: 0.55        #[real] Diameter of uniform bed material (mm)
discharge_t: discharge_t.csv    #[function] +Discharge time series of tributary channel
discharge_waterlevel: discharge_waterlevel.csv  #[function] Time series of discharge at upstr
dt: 1   #[real] Calculation time step (sec)
e_d: 0.01         #[real] Thickness of deposited layer(m)
e_thick: 0.1      #[real] Thickness of movable bed layer(m)
edition: 0        #[integer] Select solver type
h_down: 0         #[real]    Constant value (m)
h_slope: 0.001    #[real] Initial water-surface slope of main channel
h_slope_1: 0.001          #[real] Initial water-surface slope of downstream side
h_slope_2: 0.001          #[real] Initial water-surface slope of upstream side
h_slope_t: 0.001          #[real] +Initial water-surface slope of tributary channel
i_erosion_end: 5          #[integer] Bank erosion end section number from downstream end
i_erosion_start: 5        #[integer] Bank erosion starting section number from upstream end
i_flow: 2         #[integer] Initial water surface
```

4. Making a simulation

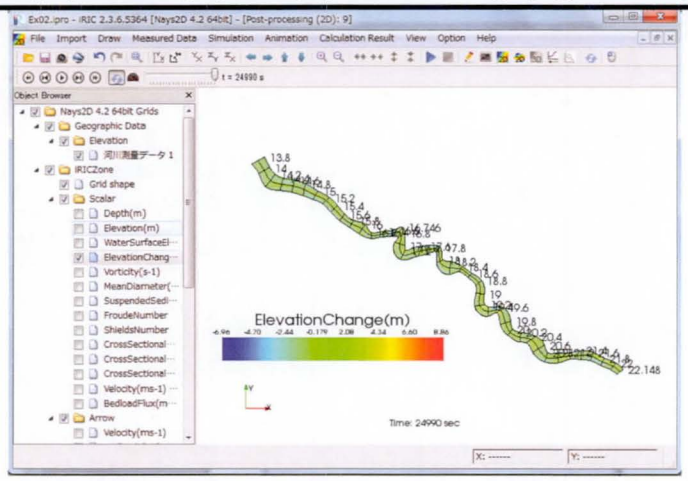
Before stating the simulation, select [File] - [Select As File (*.ipro)] and save your project.

For running a simulation, select [Simulation] -> [Run] in menu bar or select ▶ on menu tool bar. Then, Solver console window will appear and calculation starts. If you want to visualize your calculation results before the calculation finish, Select [calculation result] – [Reload] on the menu bar.

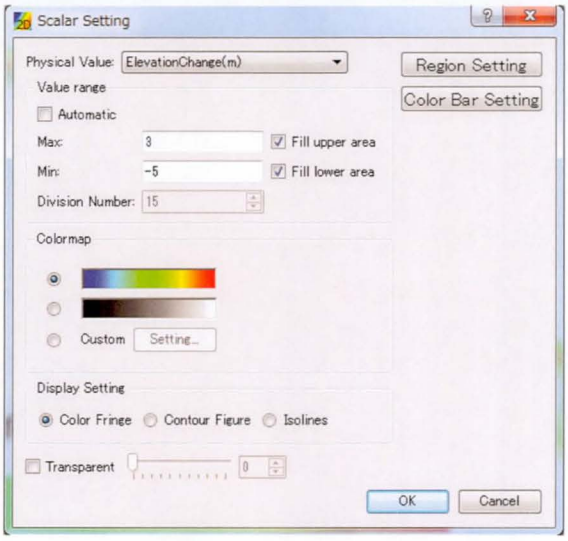
5. Visualization of calculation results

Select [calculation result] -> [Open new 2D post-processing window] on the menu bar, or click on the button [2D], named [Open 2D post-processing window].

Select [Nays2DHGrids] - [iRICZone] - [Scalar] - [Elevation change (m)] in the Object Browser.



To fix the scale bar, select [Nays2DHGrids] - [iRICZone] - [Scalar] in the Object Browser and right click on [Contour] to select [Property]. In the [Property window] as shown in figure below, set the value range as followings.

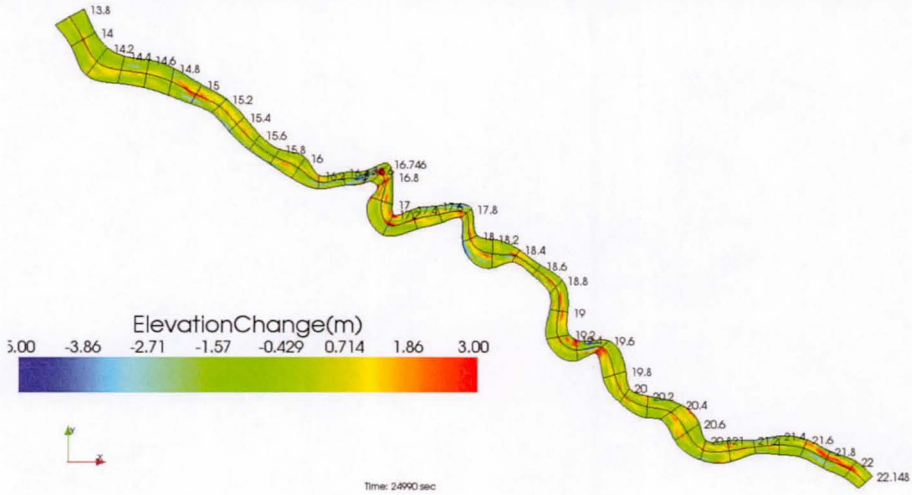


- Value Range
  ------ Remove the check of Automatic

  - Max: 3
  - Min: -5

- Transparent
  ------ Remove the check

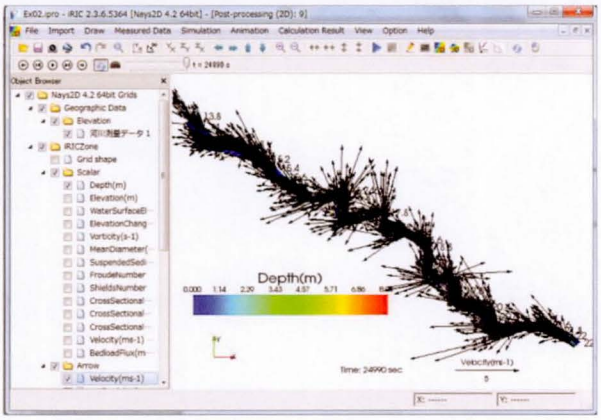
The contour map is easier to see after it is restarted.

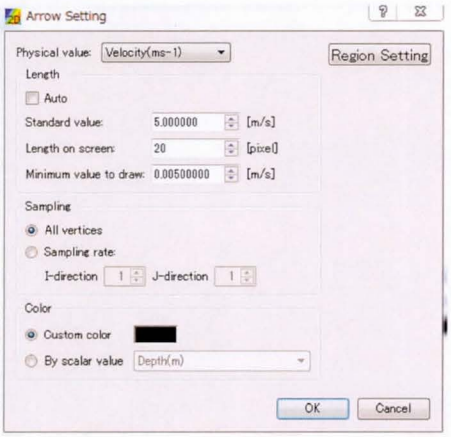


Visualizing the flow velocity:

In the Object Browser, select [Nays2DHGrids] - [iRICZone] - [Arrow] - [Velocity].

The generated vector map is difficult to understand in default. To fix the scale of vector arrow, right click on [Arrow] and select [Property].





Contour map : Depth (m)

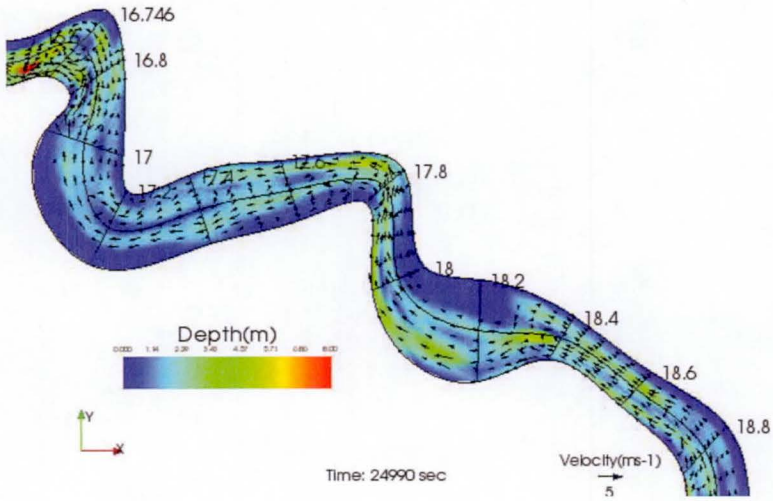
Vector map : Velocity (ms-1)

[Property] Window:

- Length
  ------ Remove the check of Automatic

  - Length on screen: 20

The contour map is easier to see after it is restarted.

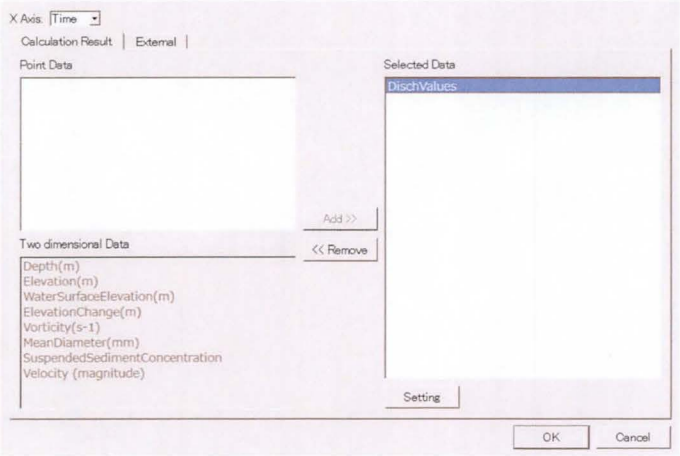


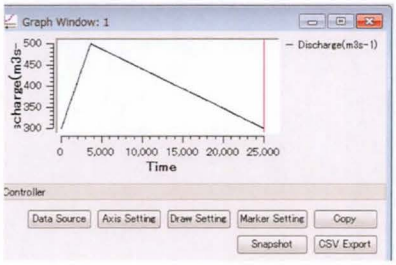
1D Graph Window:

Either of the following actions opens a new graph window.

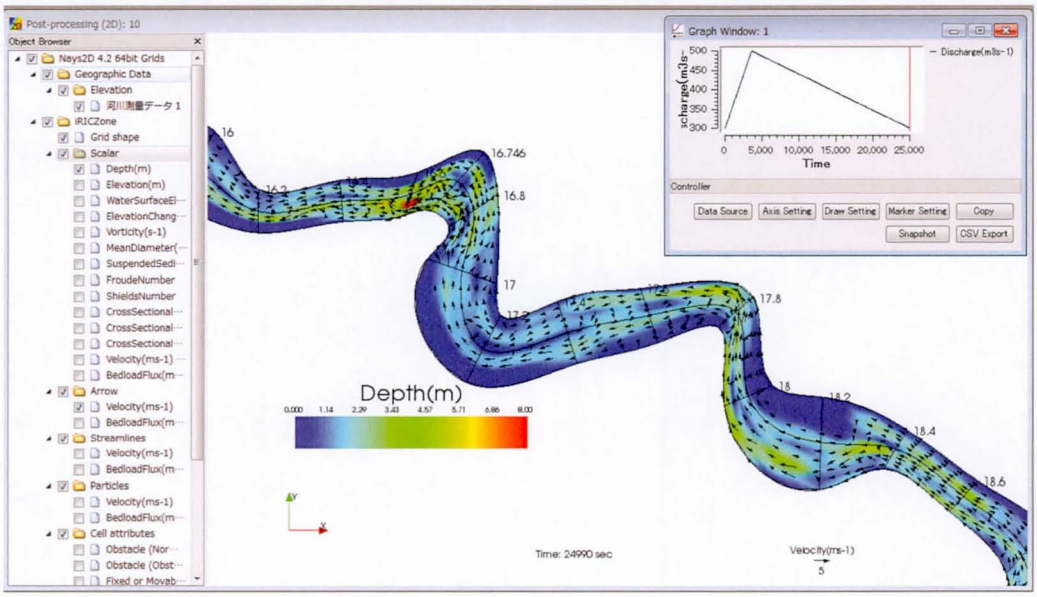
Menu bar: [Calculation Results] (R) ► [Open New Graph Window]

Operation Toolbar:



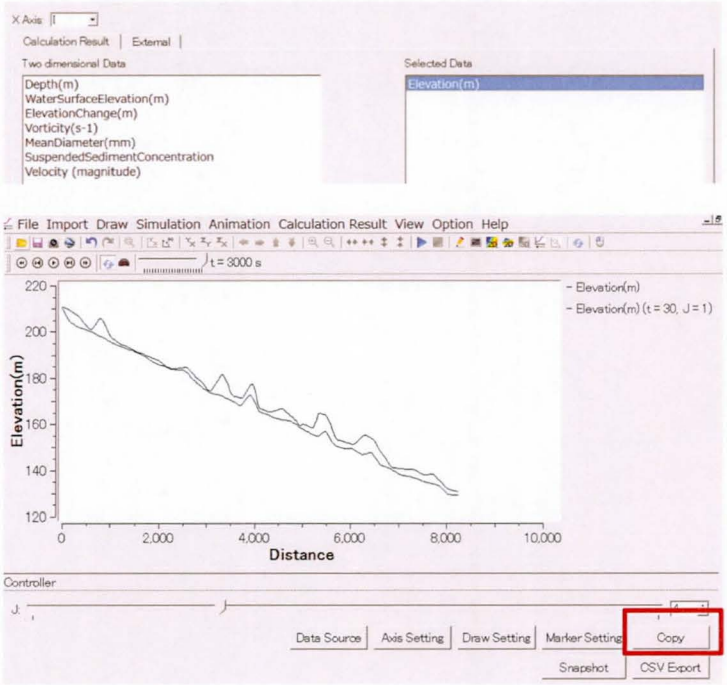


Visualize contour and vector map with discharge graph.



1D Graph Window:

Elevation change in each cross section can be confirmed with 1D Graph window. Open 1D Graph window ( ) and select X axis and Y axis as below. Using copy function, elevation can be compared.



Export Animation: Select [File] -> [Continuous Snapshot / Movie / Google Earth Export….].

End Tutorial 3.

iRIC Software

Changing River Science

Nays2D Flood Examples of Asian Rivers

Copyright 2015 iRIC Project All Rights Reserved.

Nays2D Flood basic operating procedures

The following are the basic procedures for operating Nays2D Flood with iRIC:

Launching Nays2D Flood
Prepare to use Nays2D Flood on iRIC.

↓

Creating the calculation grids
Create a grid for calculation using Digital Elevation Model (DEM) data or the like.

↓

Setting the calculation conditions
Set simulation discharge, boundary conditions, roughness and other items.

↓

Making simulations
Run Nays2D Flood for the simulation.

↓

Visualizing the calculation results
Visualize the simulation results, such as flow velocity, water depth and riverbed elevation, by means of a contour map/vector map to see whether the simulation has successfully run.

Simulation conditions of Nays2D Flood

This manual teaches how Nays2D Flood is used to simulate river flow and riverbed deformation.

Hence, there are some omissions in the explanations of the physical and numerical aspects of the simulation conditions that are to be set. Nays2D Flood has functions (setting conditions) additional to those explained by this manual. For details, please refer to the Nays2D Flood Solver Manual.

Chapter 1
An example of flood calculation for an actual river section

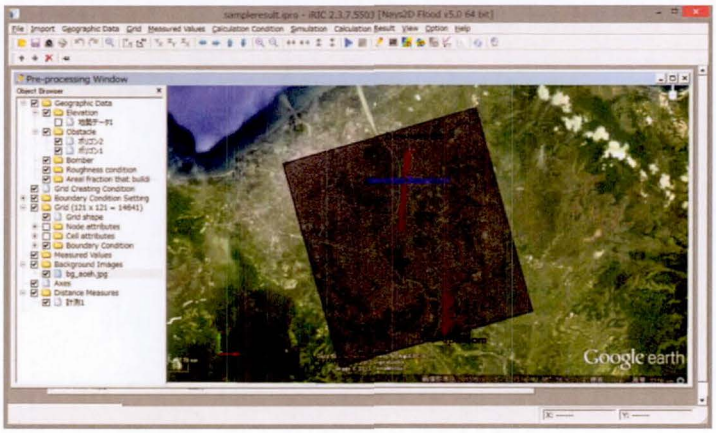
Objectives

Simulate the flow regime (water depth and flow velocity) by using Nays2D Flood for an actual river section with a flood discharge, and see whether the simulation is successfully run.

Outline

1 Creating the calculation grid

Using elevation data of an actual river section, create a calculation grid: 31 divisions in the transverse direction, and 64 divisions in the longitudinal direction.



2 Setting the calculation conditions

Set flood discharge for unsteady flow. Set various other conditions necessary for simulation.

3 Making a simulation

4 Visualizing the calculation results

Here, we introduce how to display a water depth contour map and a flow velocity vector map.

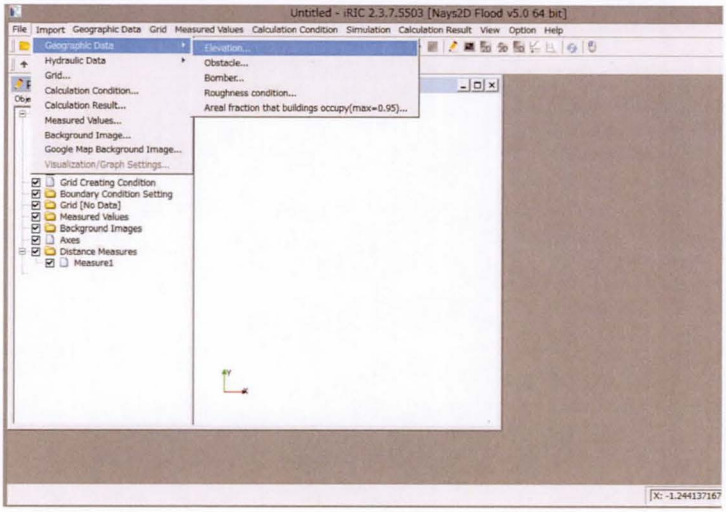
Creating the calculation grid

1. Importing cross-sectional river survey data

① Importing geographic data

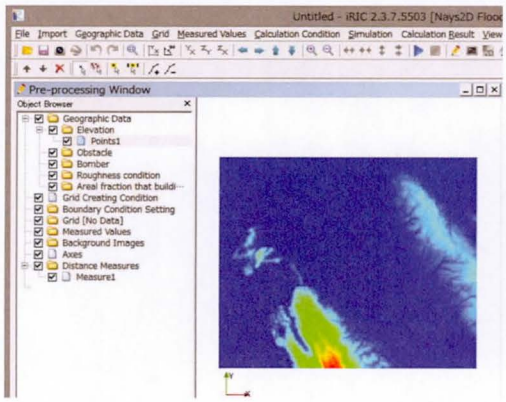
On the menu bar, select [Import] - [Geographic Data] - [Elevation].

Open [chapter1]-[1_inputdata], select [acehcity.tpo] and click on [Open].



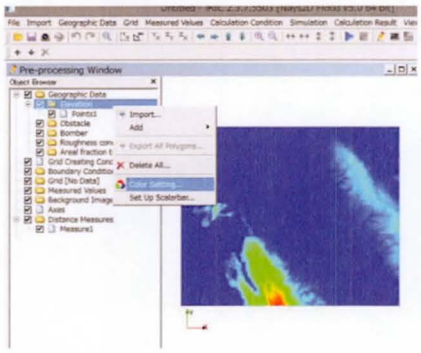
➢ On the dialog [Filtering Setting], input [1] for [Filter] value and select [OK].

Set a larger number for filter value and filter the dataset if the operation is slow because of the number of data.

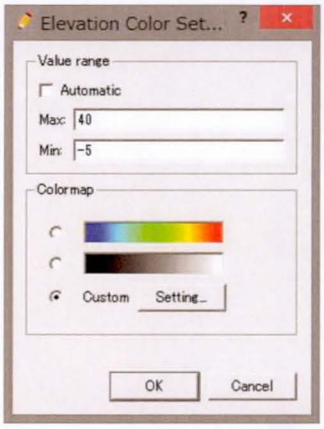


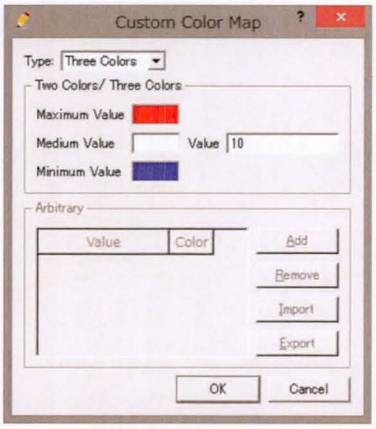
If the [Pre Processing] window shows the shape of the river section that you are simulating, the data have been successfully imported.

In the object browser, right click the "Elevation" of Geographic Data folder and click "Color Setting"



Elevation color setting window will open. Uncheck the "Automatic" and set the value range; 40 for Maximum and -5 for Minimum.
Then click Custom Setting in Colormap section. Select "Three Colors" for the legend type, choose white for the intermediate color and set the value as 10. Click OK.





Value range:
---- uncheck: Automatic

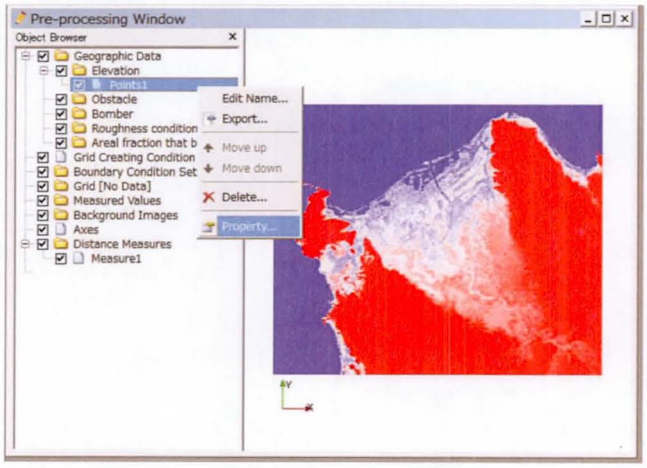
---- Max.: 40
---- Min: −5

Colormap: Custom
------- Setting --------
Type: Three Colors

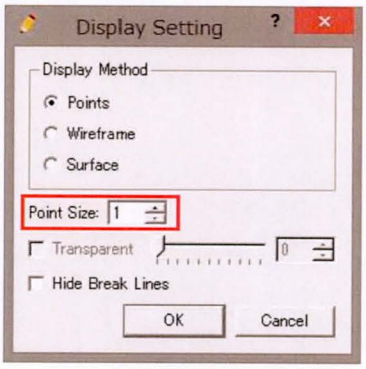
Medium Value:
White (Recommended)

Value: 10

In the object browser, right click the "Points1" of Elevation folder and click "Property".



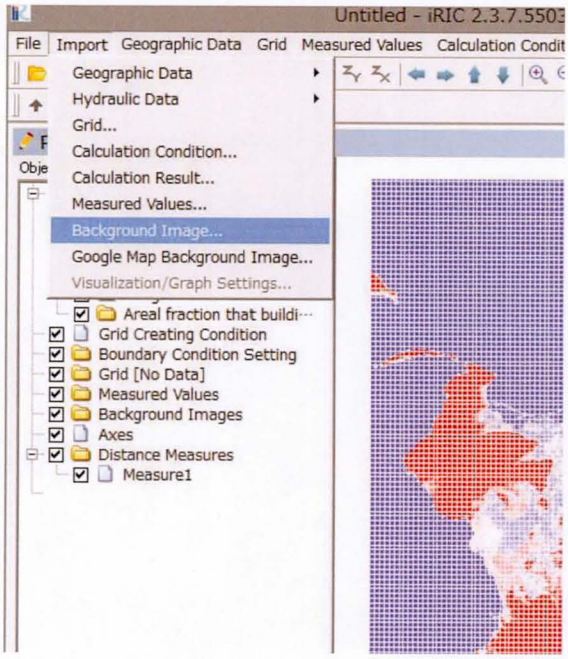
In the Display Setting popup window, set 1 for the point size.



Point size: 1

② Importing a background image

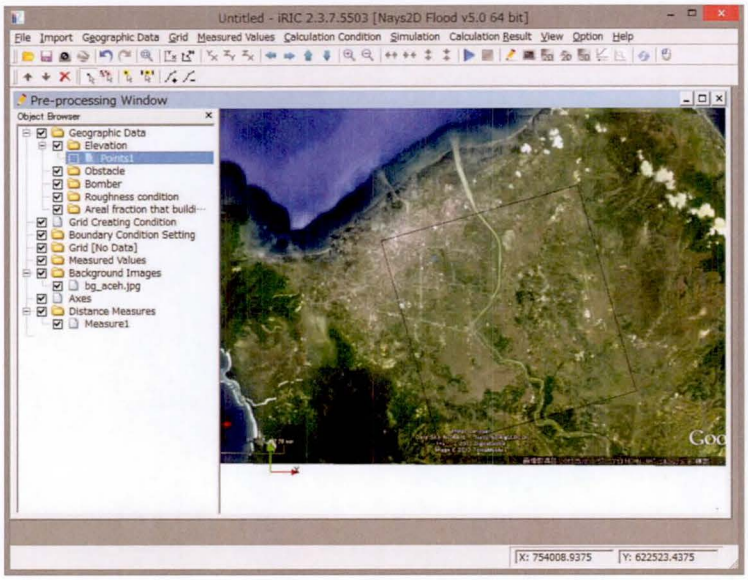
➢ On the menu bar, select [Import] - [Background Image] and import [chapter1]-[1_inputdata] and select [bg_aceh.jpg].



Background image:

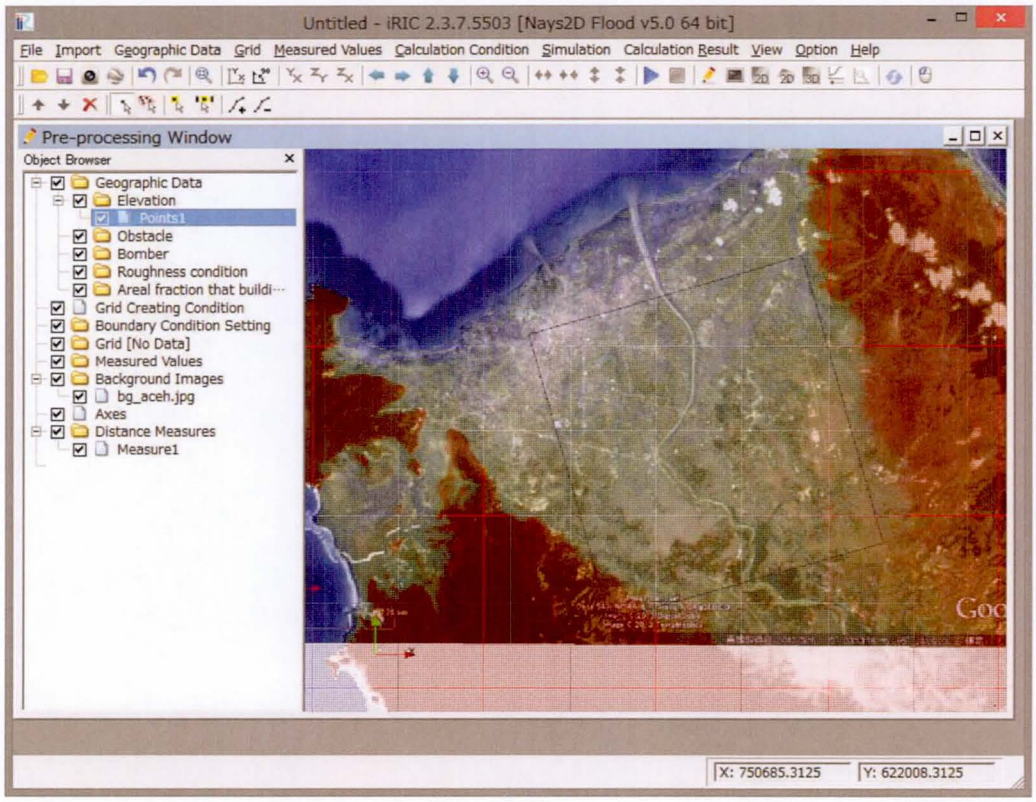
When creating grids for calculation, importing background images such as maps and aerial photos makes it possible to create grids that incorporate riverbanks and land use. Obstacle cells and roughness cells mentioned below can be set in reference to the background image.

➢ Using the [Move], [Rotate] and [Zoom] functions, match the background image with the elevation data.



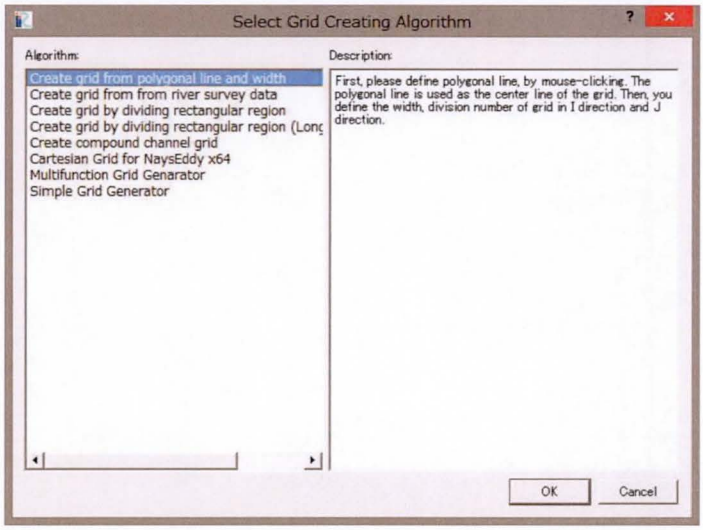
Hint:
It is convenient to find the geographic features and mark on the background image when you match the background image with elevation data. In this case, the flame line is drawn and you can use this flame line to match.

➢ Referring to the hint above, match the background image with the evaluation data.



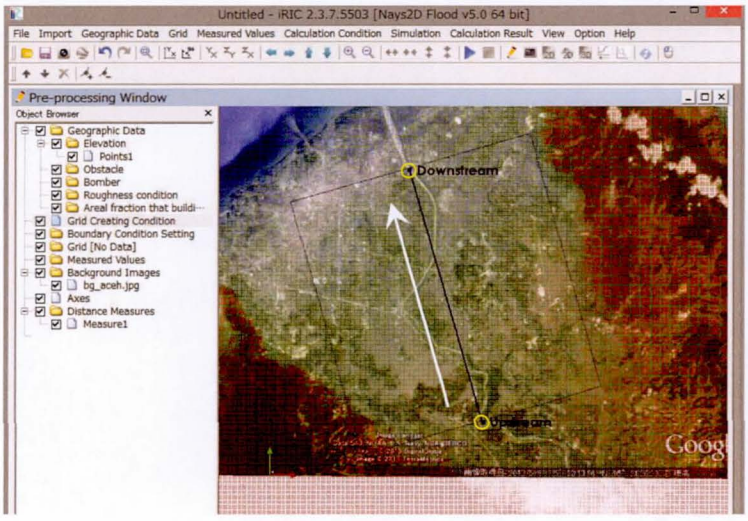
2. Selecting an algorithm for creating a grid

On the menu bar, select [Grid] - [Select algorithm to create grid] and select [Create grid from polygonal line and width] from the list below the [Select Grid Creating Algorithm] window, and click on [OK]. Instruction popup will appear. Nays2D Flood primarily creates a grid from polygonal lines and grid widths.



3. Creating a grid

➢ Click on the upstream and downstream points through which the grid centerline passes, and press the "Enter" key.



2. Downstream

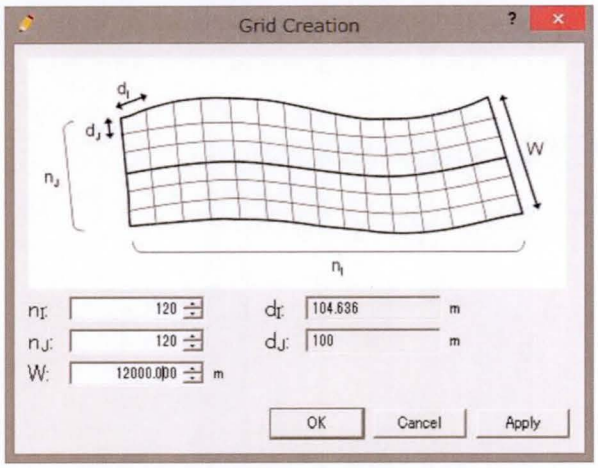
Setting the grid centerline:

Set the grid centerline from the upstream end, where flood flow enters, to the downstream end, where the flood flow exits. To finish, double-click on the end of the centerline, or hit the "Enter" key.

➢ In the [Grid Creation] window, make the following settings and click on [OK].

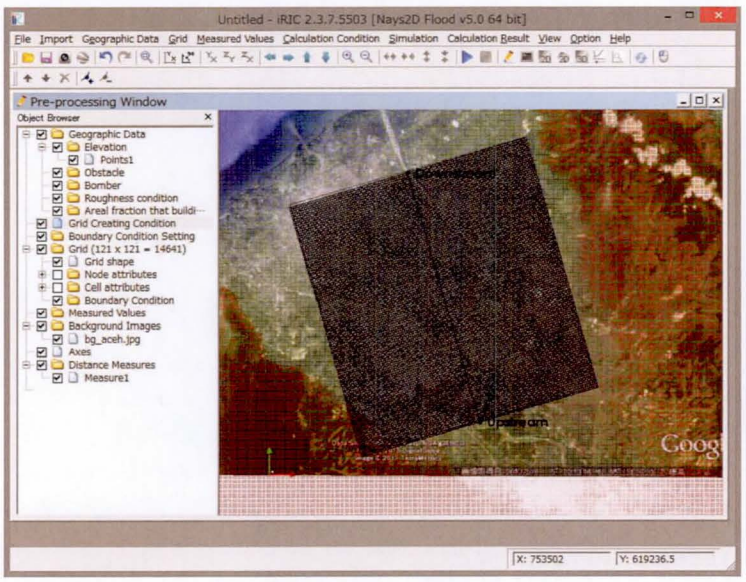
The resulting mesh size would be d_I = 100 m (flow direction) and d_J = 100 m (cross sectional direction)

➢ Click on [Yes] in the [Confirmation] window.



- Number of divisions in the longitudinal direction: 120
- Number of divisions in the transverse direction: 120
- Grid width in the transverse direction: 12000 m

Calculation grid will be created.



Adjusting the calculation grid:

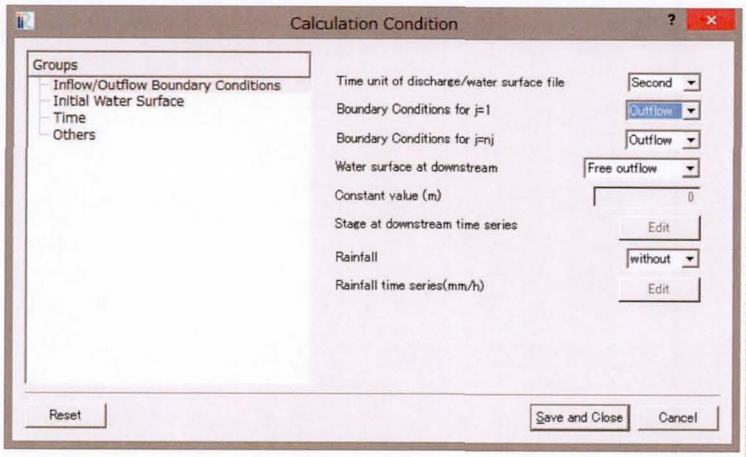
It is possible to move, add or remove centerlines under [Grid Creating Conditions] even after the grid is created.

Setting the calculation conditions

4. Open "Calculation Conditions"

On the menu bar, select [Calculation Conditions] - [Setting]. The [Calculation Conditions] window will open.

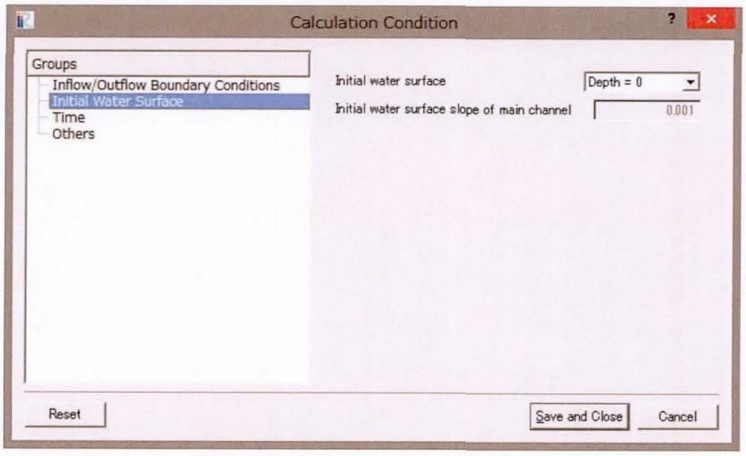
Setting the inflow boundary conditions : click on [Inflow/ Outflow Boundary Conditions] from the [Group] list to make the following settings:



※Inflow conditions are decided at [6. Setting the inflow river conditions]
※Use constant value or read from file when the stage at downstream is effected by sea level or overflow level at outflow.

- Time unit of discharge/water surface elevation files: second
- Boundary Conditions for j=1: Inflow
- Boundary Condition for j=nj: Outflow
- Water surface at downstream: Free Outflow

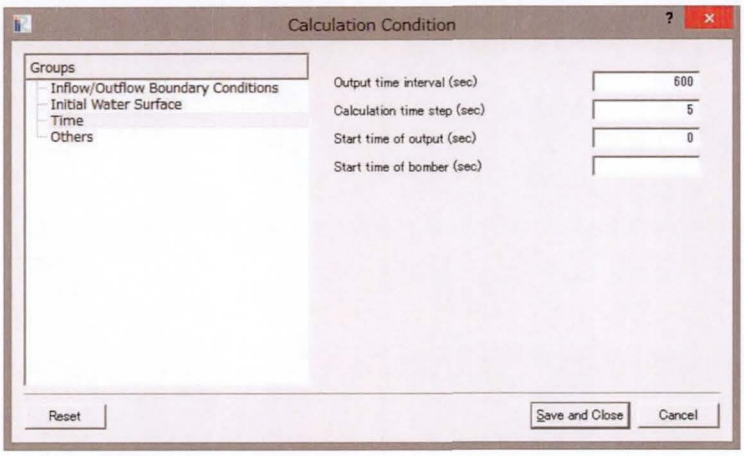
Setting the initial water surface profile: click on [Initial Water Surface Elevation] from the [Group] list to make the following settings:



- "Initial water surface": Depth = 0

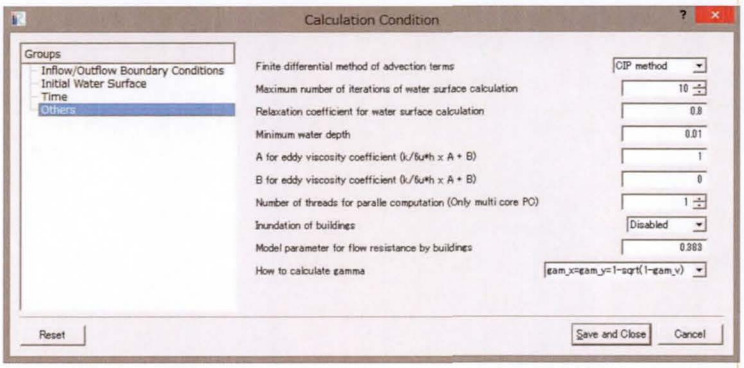
Note: When the water surface elevation of the downstream end may be affected by the sea level or the downstream flood level, use a constant value (a line).

Setting the time: click on [Time] from the [Group] list to make the following settings:



- Output time interval (sec): 600
- Calculation time step (sec): 5
- Start time of output (sec): 0
- Start time of bomber (sec): BLANK

Other settings: Click on [Other] from the [Group] list to make the following settings:



- Finite difference method of advection terms: CIP method
- Maximum number of iterations of water surface calculation: 10
- Relaxation coefficient for water surface calculation: 0.8
- Minimum water depth: 0.01

Many other conditions can be set; however, they do not need to be set for this simulation. Details of other parameters are described in Nays2D Flood Solver Manual.
After making the settings above, click on [Save and Close] to close the window.

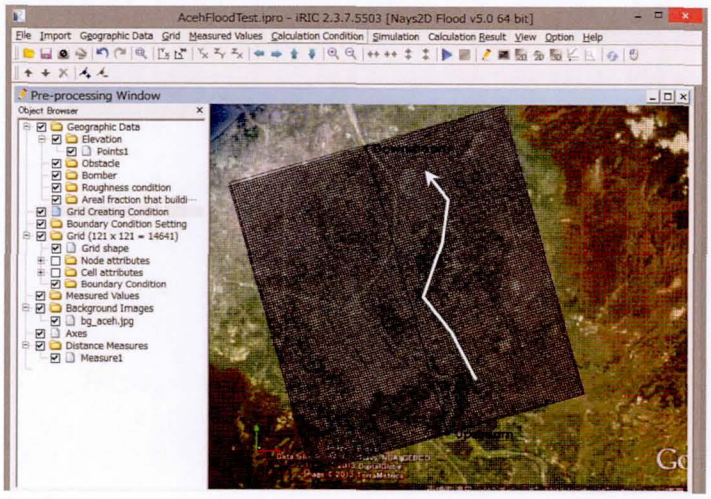
5. Saving the project

Before proceeding, saving the project is important. In the menu bar, click File and select "Save As File (*.ipro)" This option enables you to save the data for this pre-process as a single file (Compressed file). On the other hand, "Save As Project" means saving the project in an arbitrary folder.

6. Setting inflow condition

① Inflow river

Set the inflow river condition at the upstream boundary.

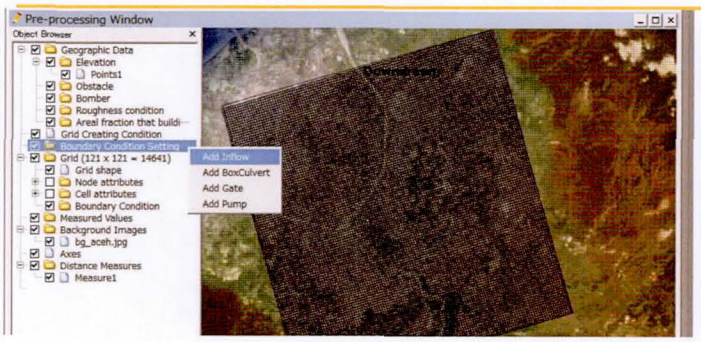


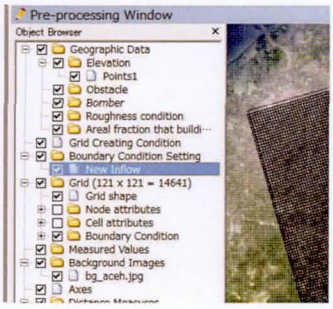
- In this river section, one river (Aceh River) flows across the calculation domain, as indicated by the white line.

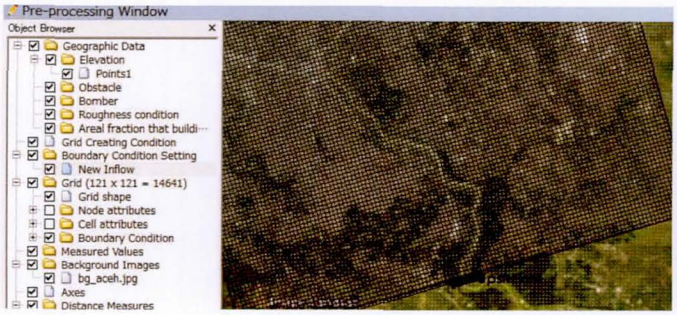
- No river comes in the area from j=1 and j=nj and these boundary conditions are outflow; see the boundary condition

② Setting of inflow condition

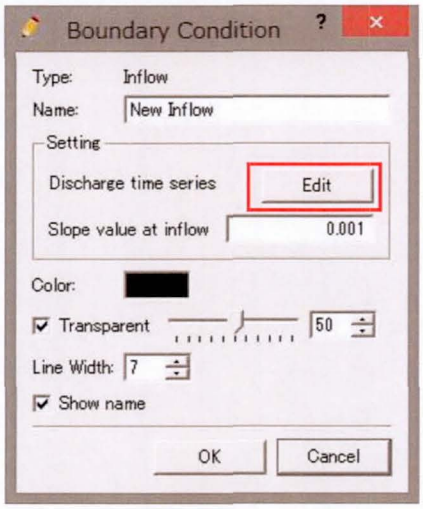
In the object browser, right-click the [Boundary Condition Setting] folder. Select [Add inflow], then "New Inflow" setting will be added. Selecting "New Inflow", enclose the sides of inflow location with a polygon. Click "Enter" and the location is confirmed.







The Boundary Condition setting dialog opens.

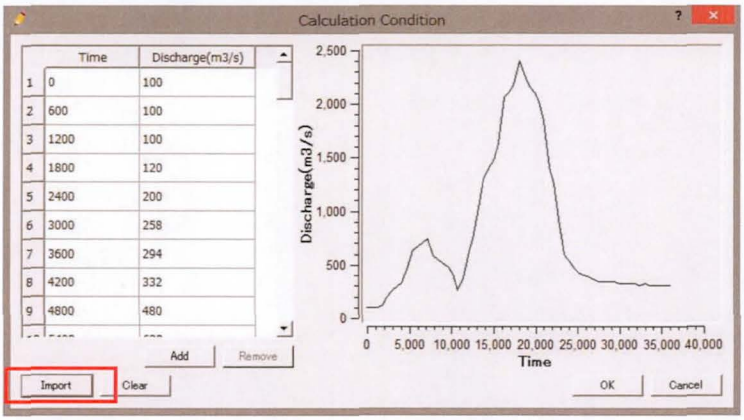


Name: arbitrary name

--------- Setting ---------
Discharge time series:
Select [Edit] and input / import discharge data as the figure below

Slope value at inflow:
0.001

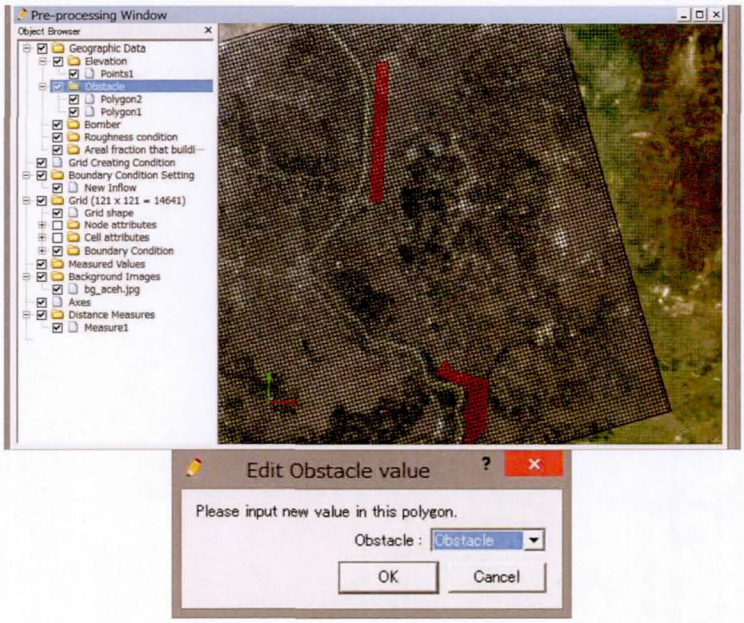
After selected [Edit], input/import the discharge from the upstream boundary. In this tutorial, select [Import] discharge and select [chapter1]-[2_hydraulicdata] and open "Qin_Aceh.csv"



Hydrograph will be imported

7. Setting obstacle cells

In the Object Browser, select [Geographic Data] - [Obstacle] - [Add] - [Polygon]. Enclose the grids of "Obstacle" locations with polygons. Set the obstacle from "Edit Obstacle value" popup.

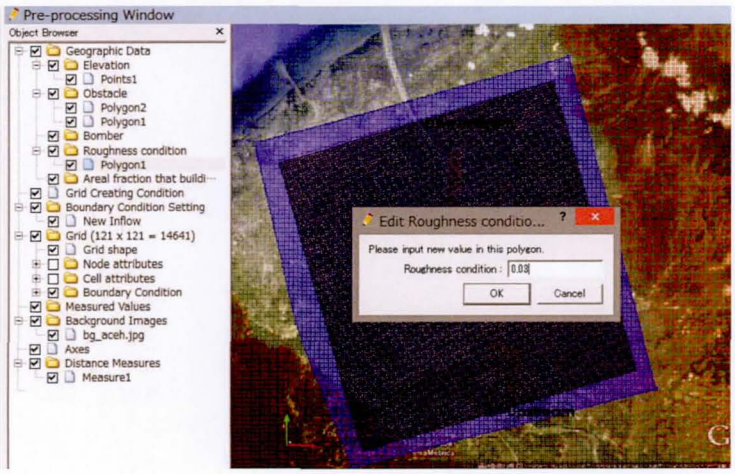


•Set obstacles such as roads, banks and embankments.

Note: Unless the calculation grid cells are small enough to depict roads, banks and embankments as a part of the topography, set roads, banks and embankments as obstacles.

8. Setting the roughness

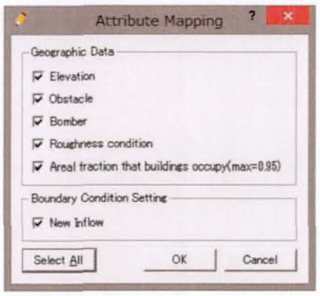
In the Object Browser, select [Roughness condition] - [Add] - [Polygon]. Enclose the all grids with a polygon. Input Manning's roughness coefficient.



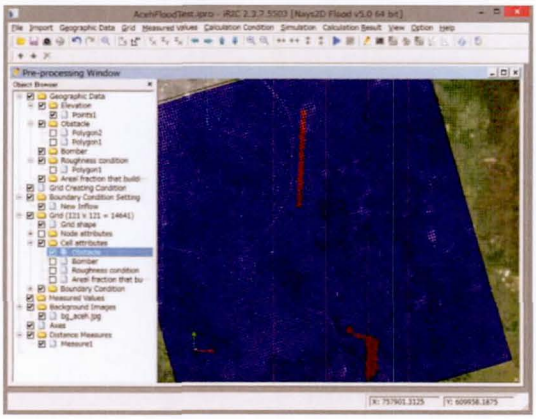
• Roughness coefficient: 0.03

Note: Set Manning's roughness coefficient by taking comprehensive consideration of calculation model, land use and past flooding data.

On the menu bar, select [Grid] - [Attributes Mapping] - [Execute]. Attribute Mapping confirmation will appear and click OK. Information popup will appear and check [Obstacle] and [Roughness condition], and then click OK.



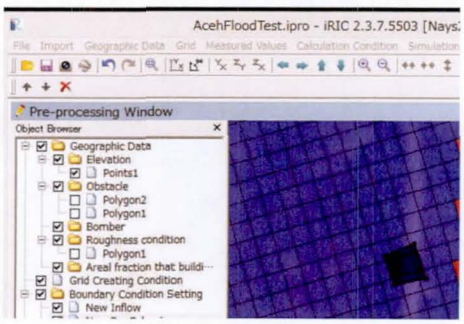
In the Object Browser, uncheck [Geographic Data] – [Obstacle] and [Roughness condition]. In the Object Browser, check [Grid] – [Cell attributes] - [Obstacle]



Confirm the cells which was given obstacle attributes

9. Setting for the box culvert

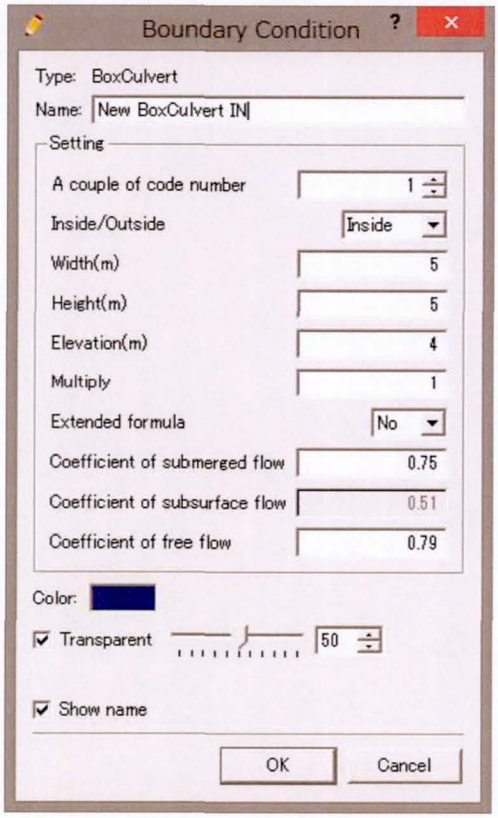
In the Object Browser, select [Boundary condition setting] - [Add Box Culvert] to set a Box culvert cell (inflow side). Enclose the grids for box culvert location (inflow side) with polygons.



Box culvert can be specified in one cell units.

Set an "inflow side" of a Box culvert using polygon.

When the location is confirmed, the [Boundary Condition] setting dialog opens. Set the conditions of box culvert from in the dialog and click OK.



Name:
New BoxCulvert IN

A couple of code number: 1

Inside/Outside: In

Width: 5m

Height: 5m

Elevation: 4m

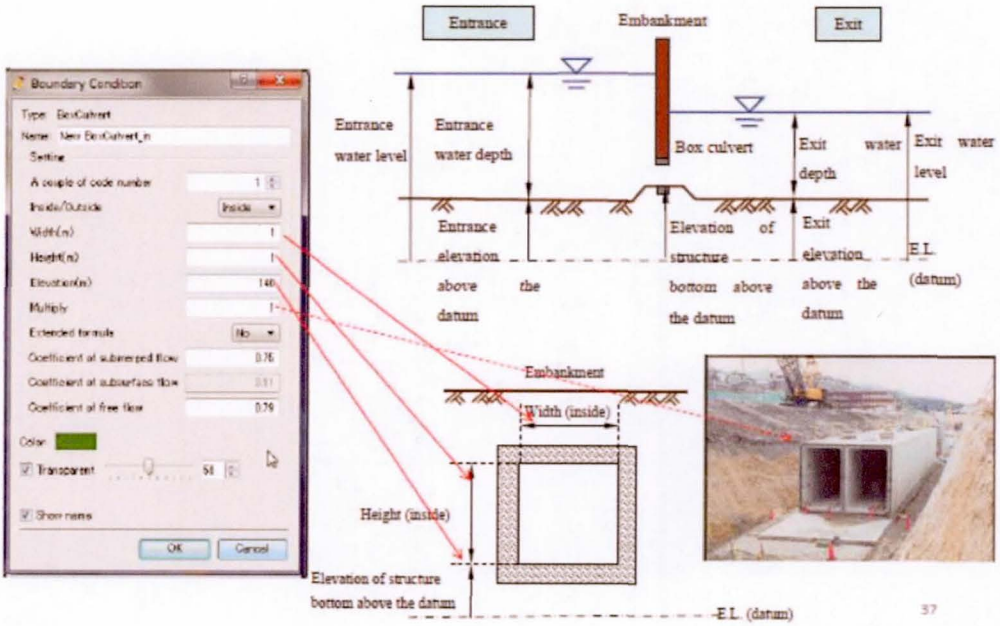
Multiply: 1

Extended formula: No

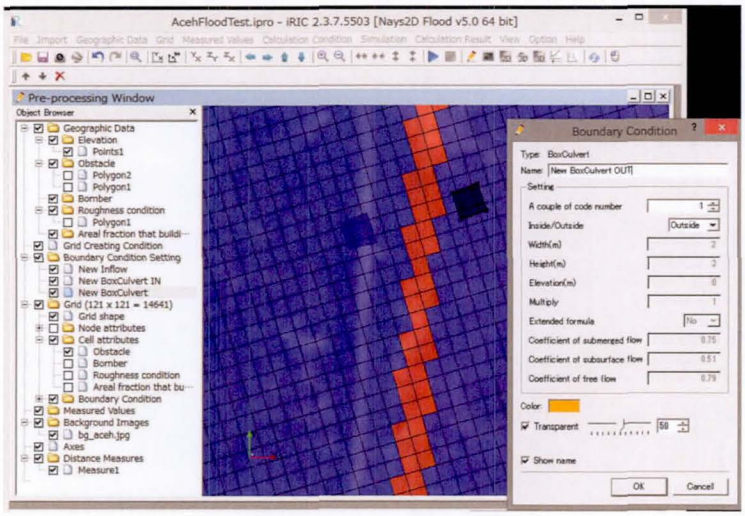
Coefficient of submerged flow: 0.75

Coefficient of free flow: 0.79

Box culvert

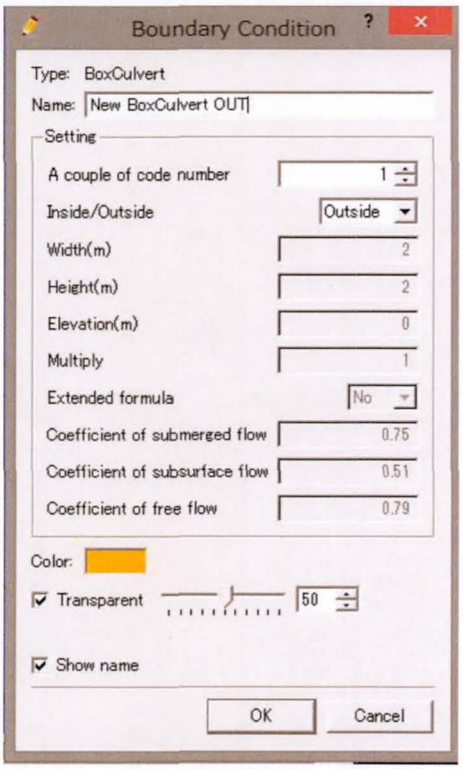


In the Object Browser, select [Boundary condition setting] - [Add Box Culvert] to set a Box culvert cell (outflow side). Enclose the grids for box culvert location (outflow side) with polygons.



Set an "out flow side" of a Box culvert using polygon.

When the location is confirmed, the [Boundary Condition] setting dialog opens. Set the conditions of box culvert from in the dialog and click OK.

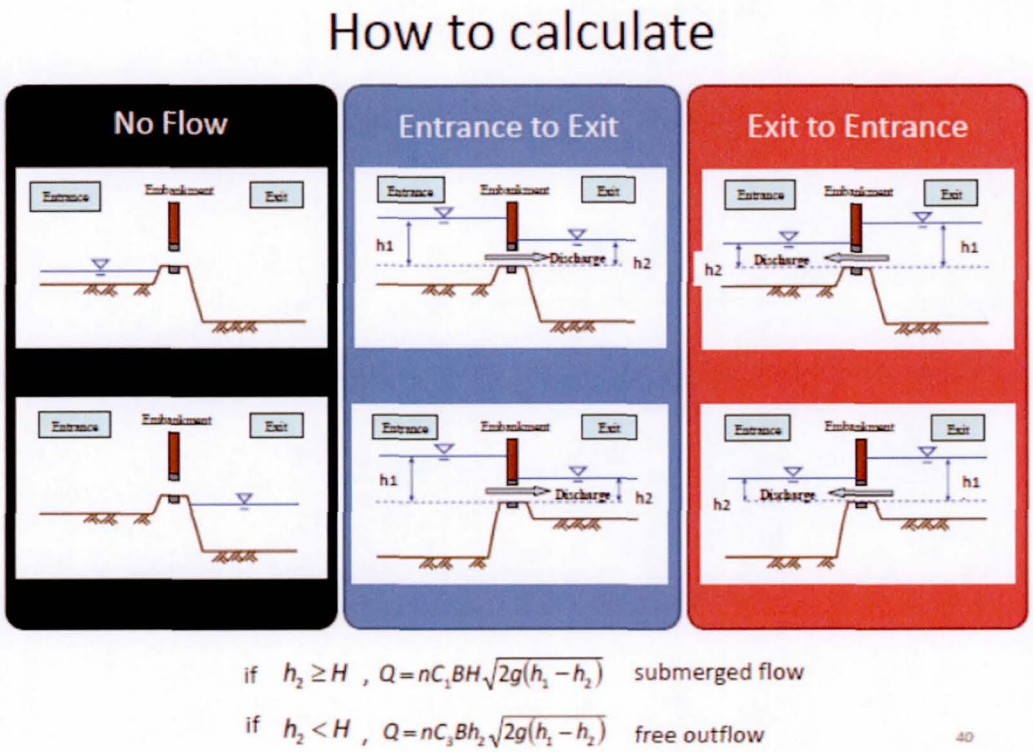


Name:
New Box Culvert

Inside/Outside: OUT

A couple of code number : 1

Outside



On the menu bar, select [Grid] - [Attributes Mapping] - [Execute] and select New Box Culvert IN and OUT, and then Click OK.

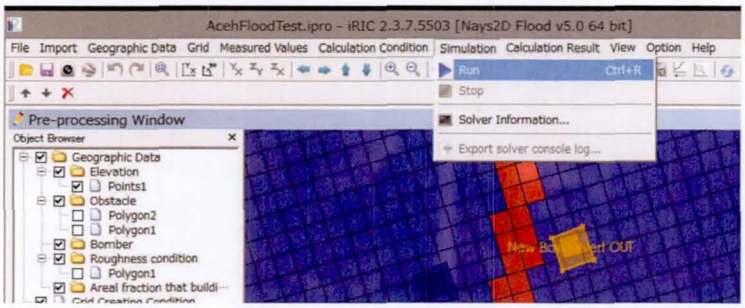
10. Saving the project

Before start calculation, select Save in the File menu of the menu bar.

Making a simulation

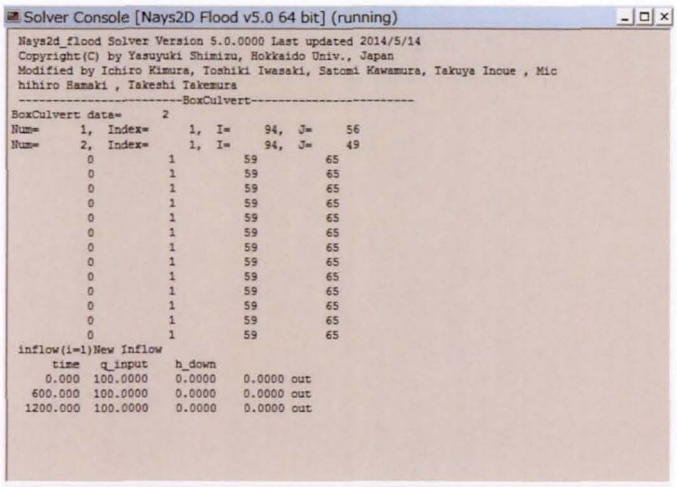
1. Start your calculation project

On the menu bar, select [Simulation] - [Run].



Click Yes for the information popup.

The [Solver Console [Nays2D Flood] (running)] window will open to start the simulation.

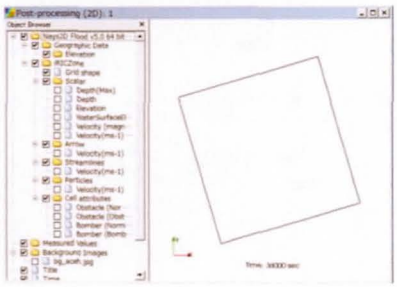


Visualizing the calculation results

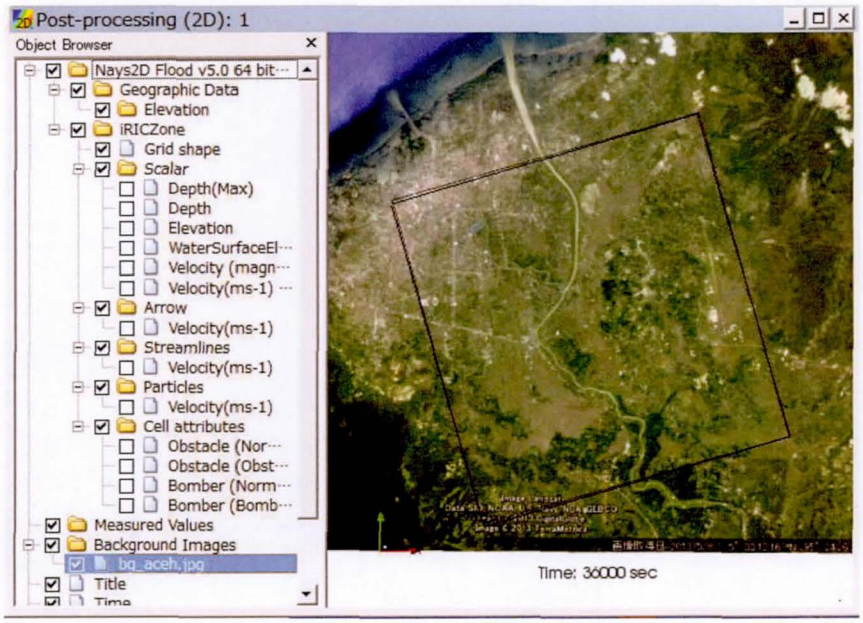
2. Open the "2D Post-Processing" window

On the menu bar, select [Calculation Result] - [Open new 2D Post-Processing ...].

The "Post-Processing (2D)" window will open.



In the object browser, select "bg_aceh.jpg" in the "Background Images" folder.

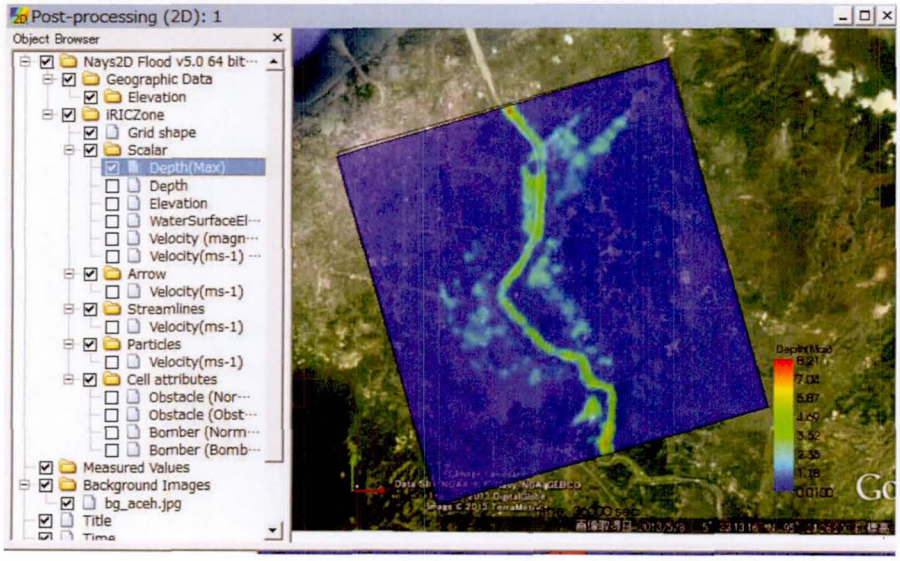


3. Visualizable quantities

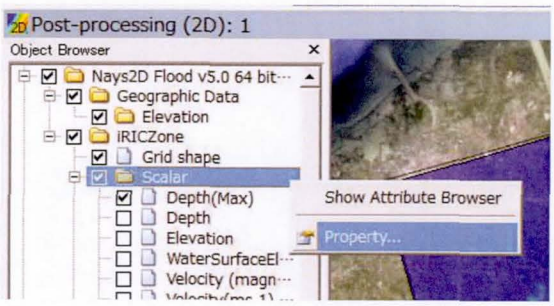
Descriptions in the Object Browser	Description of each quantity
• Contour	
Depth(Max)	The max. water depth by the time of visualization (m).
Depth	The water depth at the time of visualization (m)
Elevation	The ground height of calculation grid (m)
WaterSurfaceElevation	The water surface elevation at the time of visualization (m)
Velocity (magnitude Max)	The max. flow velocity by the time of visualization (m/s)
Velocity (magnitude)	Flow velocity at the time of visualization (m/s)
• Vector	
Velocity	Vector of flow velocity (m/s) at the time of visualization
• Streamline	
Velocity	Displays a streamline.
• Particles	
Velocity	Displays particles.
• Cell attributes	
Normal cell (a normal cell)	Displays a normal cell.
Obstacle cell (a cell with an obstacle)	Displays an obstacle cell.

4. Visualizing the maximum water depth

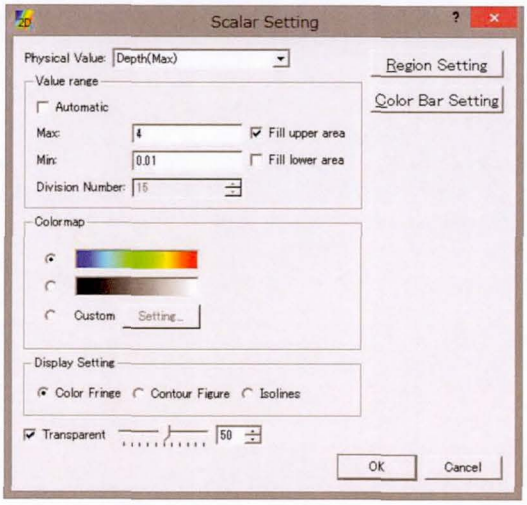
In the Object Browser, select [Nays2D Flood Grids] - [iRICZone] - [Scalar] - [Depth (Max)]. A contour map of water depth will open.



In the Object Browser, select [Nays2D Flood Grids] - [iRIC Zone] - [Scalar]. Right-click on [Scalar] to select [Property]. The [Contour Setting] window will open.

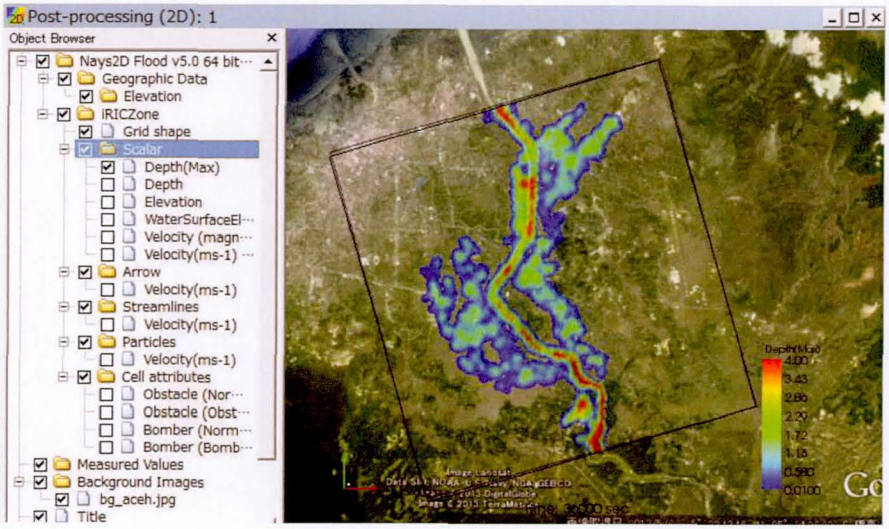


In the [Scalar Setting] window, make the following settings and click on [OK].



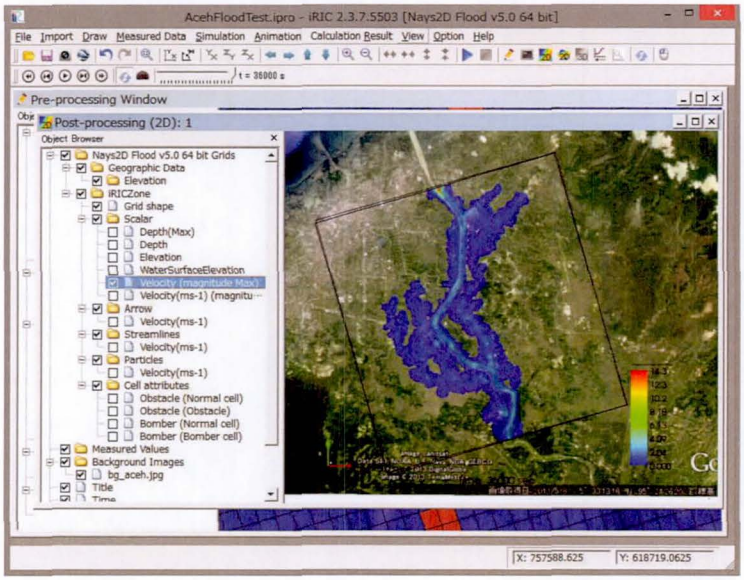
- Value range:
  Remove ☑ from Automatic.
  - Max: 4
  - Min: 0.01
  - Remove ☑ from Fill lower area.
- Display Setting:
  Contour Fringe

The contour map is easier to see now.



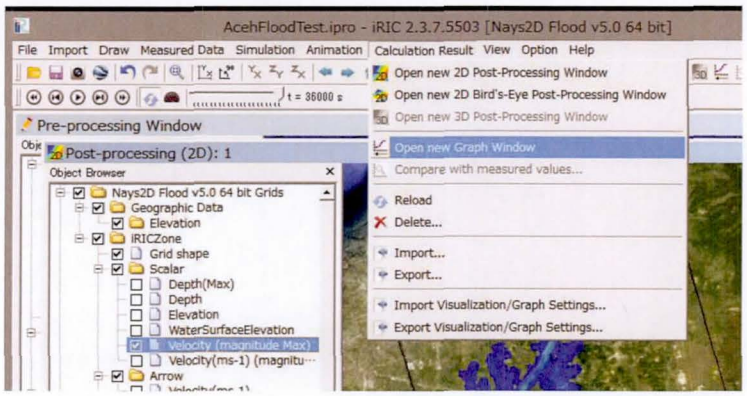
5. Visualizing the max. flow velocity

In the Object Browser, select [Nays2D Flood Grids] - [iRIC Zone] - [Vector] - [Velocity (magnitude Max)]. Open the [Property] window of [Scalar] and fix the value range of the scalar bar of velocity contour map.



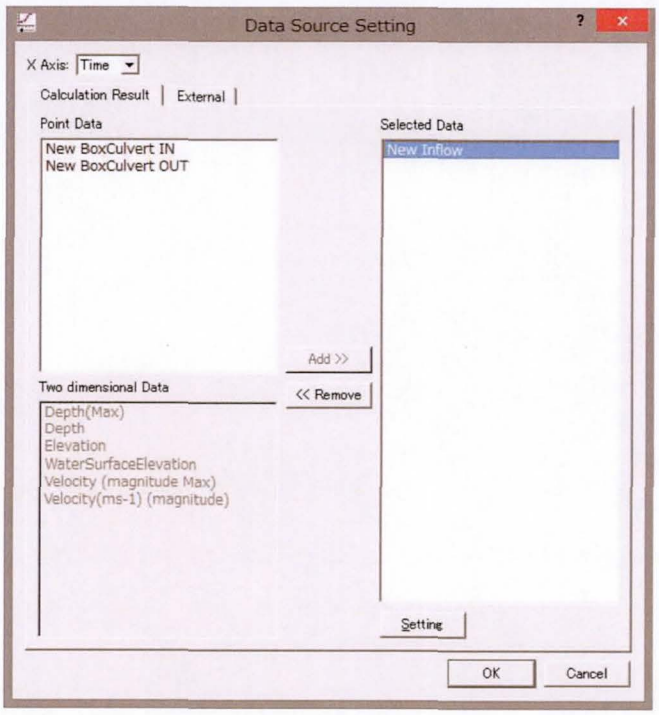
6. Visualizing the inflow discharge hydrograph

Open new Graph Window from the calculation results menu.



Open the Graph window

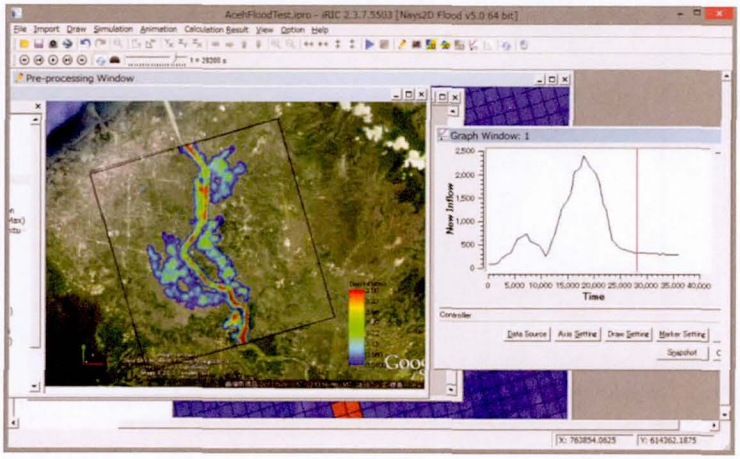
Select [Calculation Result] - [Point data] - [New Inflow]. Click [Add] button at the center.



X Axis: Time

Selected Data: Inflow

The hydrograph can be mapped with a contour map.



The hydrograph can be mapped with a contour map.

iRIC Software

Chapter 3
Examples of Tsunami Runup Analysis Using SRTM data

◆ Objectives

Simulate tsunami-runup-induced flooding with Nays2D Flood.

◆ Outline

1. Creating the calculation grid

Using elevation data of an actual river basin, create calculation grids: 101 division points in the transverse direction, and 131 division points in the longitudinal direction.

2. Setting the calculation conditions

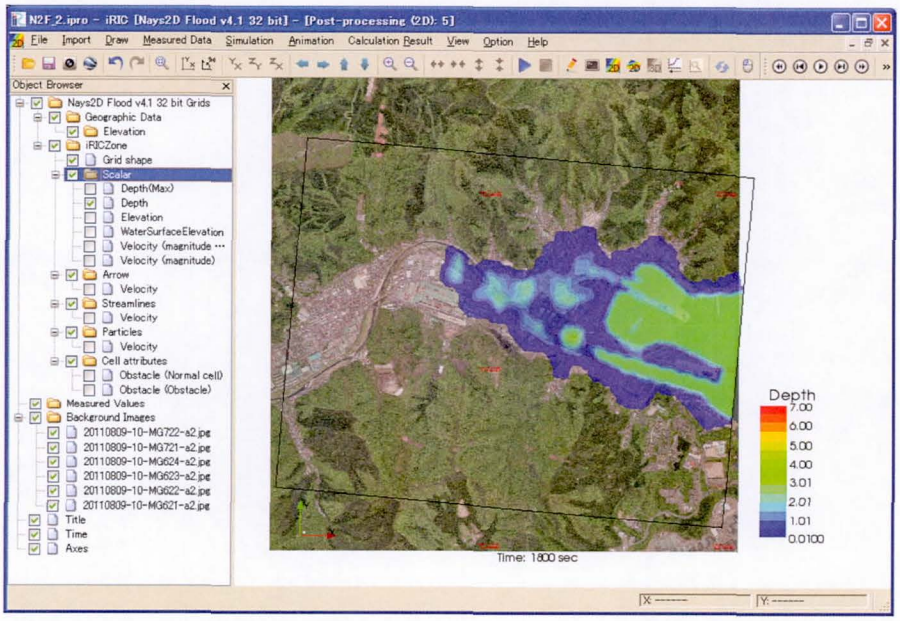
Set a value for tsunami wave height assuming that a large-scale earthquake has occurred. Set various other conditions necessary for simulation.

3. Making a simulation

4. Visualizing the calculation results

Here, we introduce how to display a water depth contour map and a flow velocity vector map.

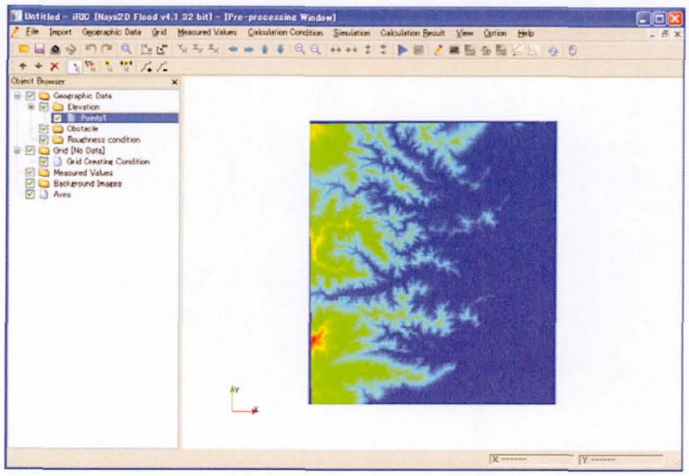
Exercise2. Tsunami flooding analysis



Creating the calculation grid #1

1. **Importing geographic data**

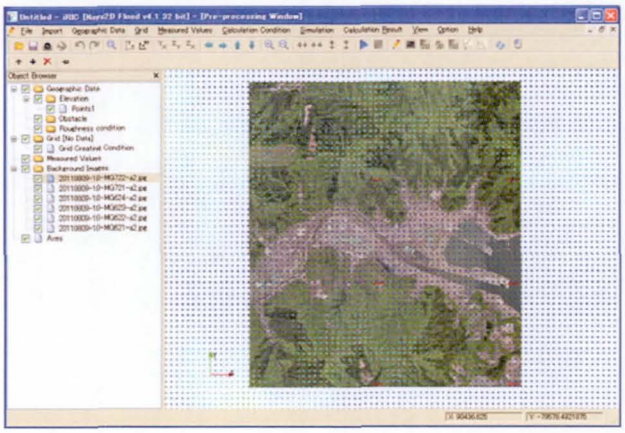
- On the menu bar, select [Import] - [Geographic Data] - [Elevation].
- Open the [¥SampleData¥N2F2] folder to select [tsunami.tpo] and click on [Open] to import the data.
- Select [1] for the value of [Filter] in the [Filtering Setting] window.



Creating the calculation grid #2

2. Importing a background image

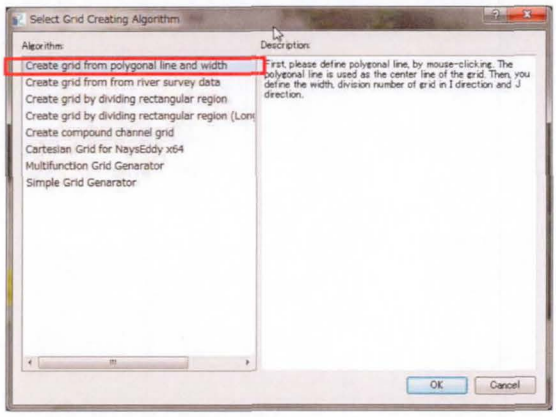
- Open all jpg files in the [¥SampleData¥N2F_2] folder, the positions of background images are automatically corrected.



Creating the calculation grid #3

3. Selecting an algorithm for creating a grid

- On the menu bar, select [Grid] - [Select algorithm to create grid].
- Select [Create grid from polygonal line and width] from the list below the [Select Grid Creating Algorithm] window, and click on [OK].

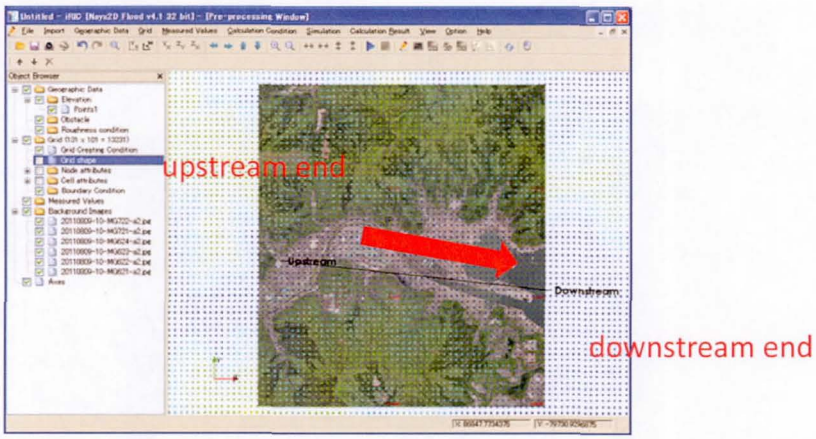


✓ Nays2D Flood primarily creates a grid from polygonal lines and grid widths.

Creating the calculation grid #4

4. Creating a grid

- Right-click on several points through which the grid centerline passes, and press the "Enter" key.



- In the [Grid Creation] window, make the following settings and click on [OK].

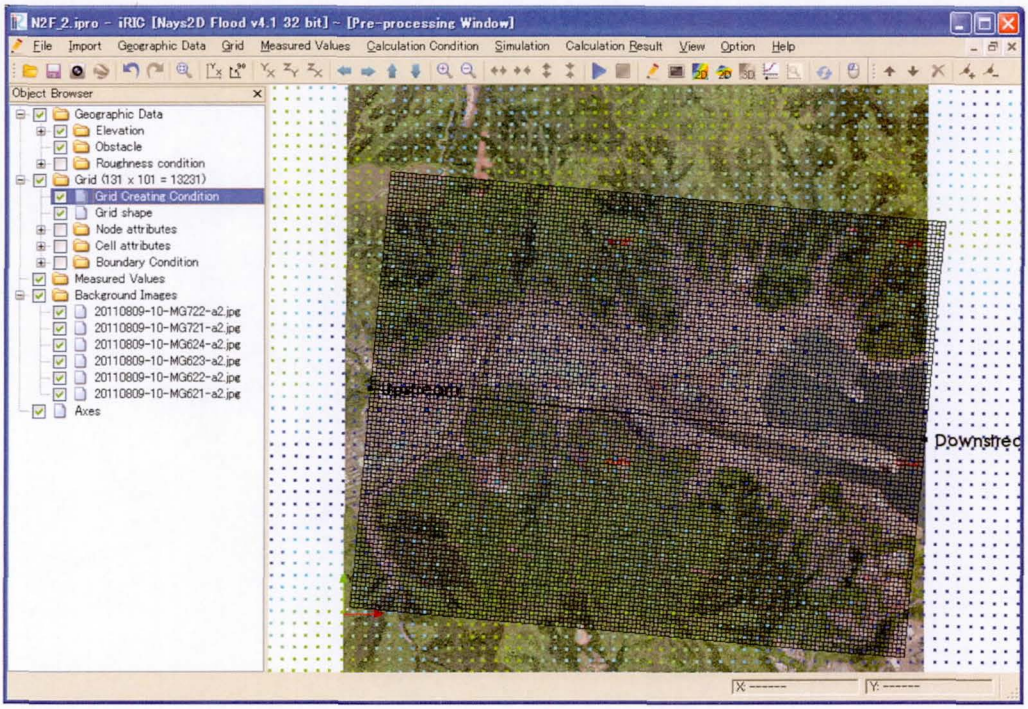
  - [nj] (Number of divisions in the longitudinal direction): 100
  - [ni] (Number of divisions in the transverse direction): 130
  - [W] (Grid width): 3000m

  Note: In this example, grid cell intervals di, dj are set at around 30 m.

- Click on [Yes] in the [Confirmation] window.

Creating the calculation grid #5

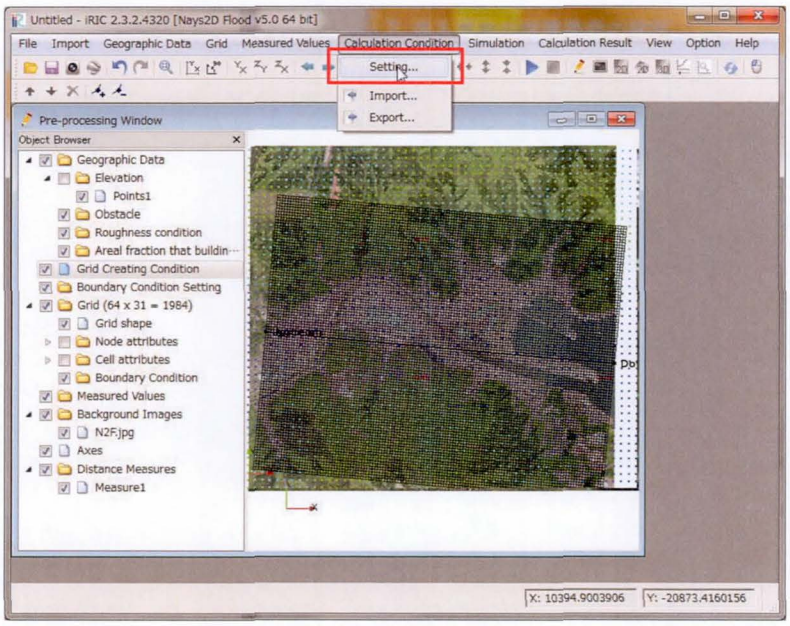
4. Creating a grid



Setting the calculation conditions #1

1. "Calculation Conditions"

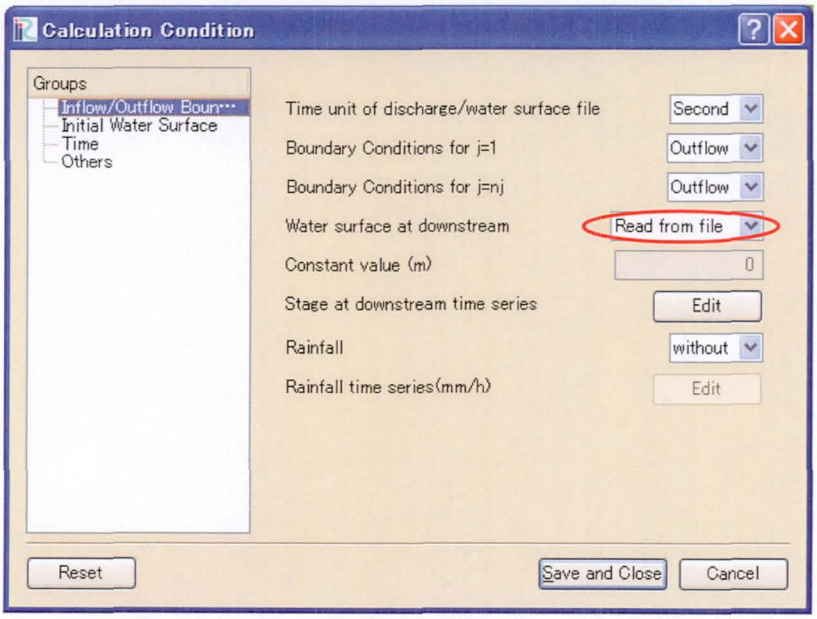
➢ On the menu bar, select [Calculation Conditions] - [Setting].



Setting the calculation conditions #2

1. "Calculation Conditions"

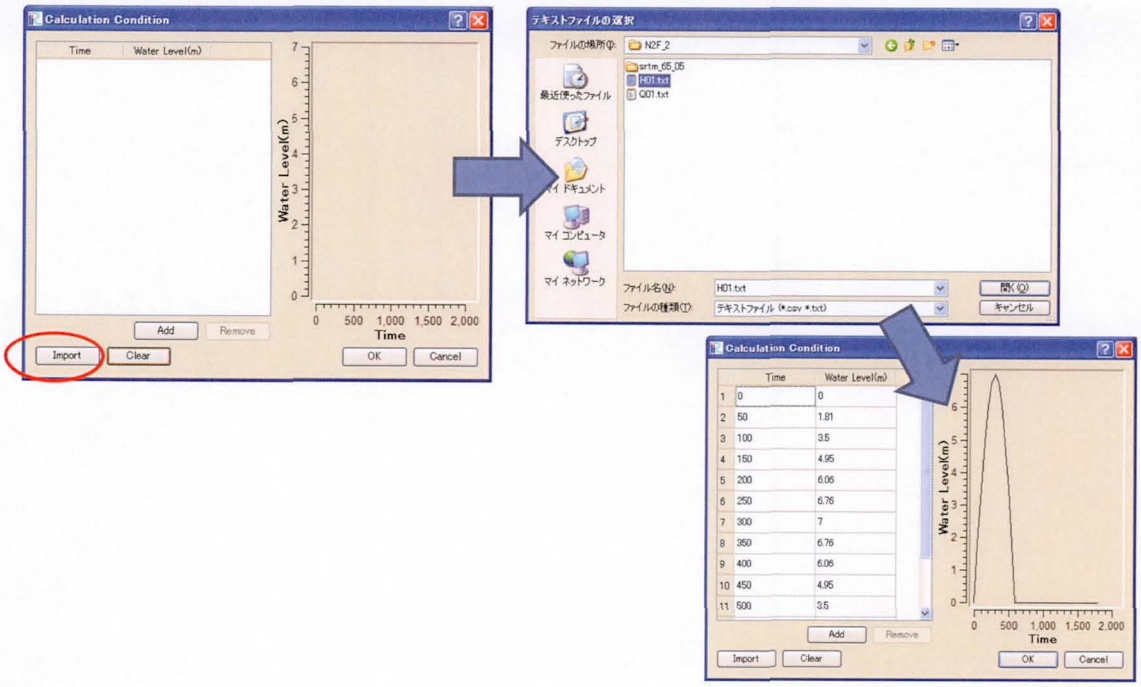
➢ Click on [Inflow/ Outflow Boundary Conditions] from the [Group] list to make the following settings:



Setting the calculation conditions #3

1. "Calculation Conditions"

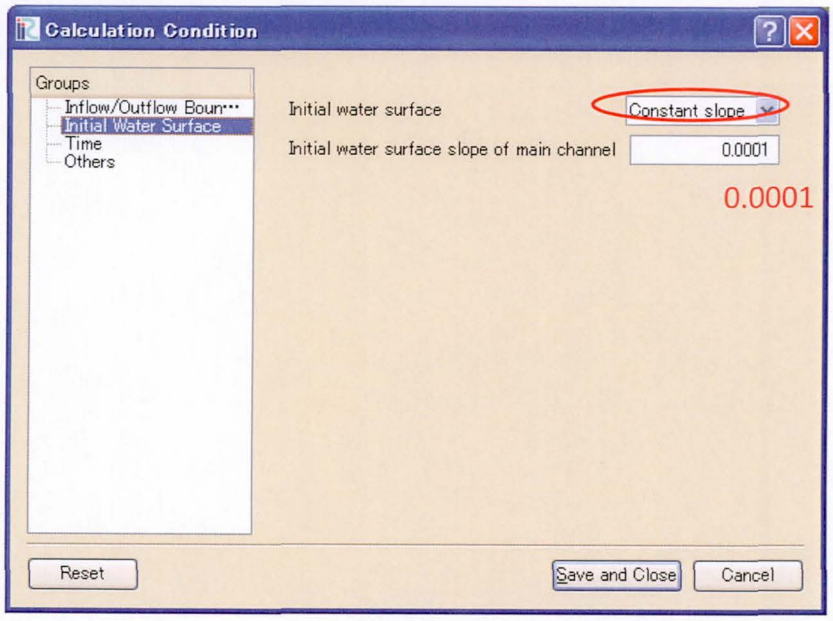
- Click on [Stage at downstream time series] to edit.
- Click on [Import] on the [Calculation condition] window.
- Select [¥SampleData¥N2F_2] - [H01.txt] and click on [Open].



Setting the calculation conditions #4

1. "Calculation Conditions"

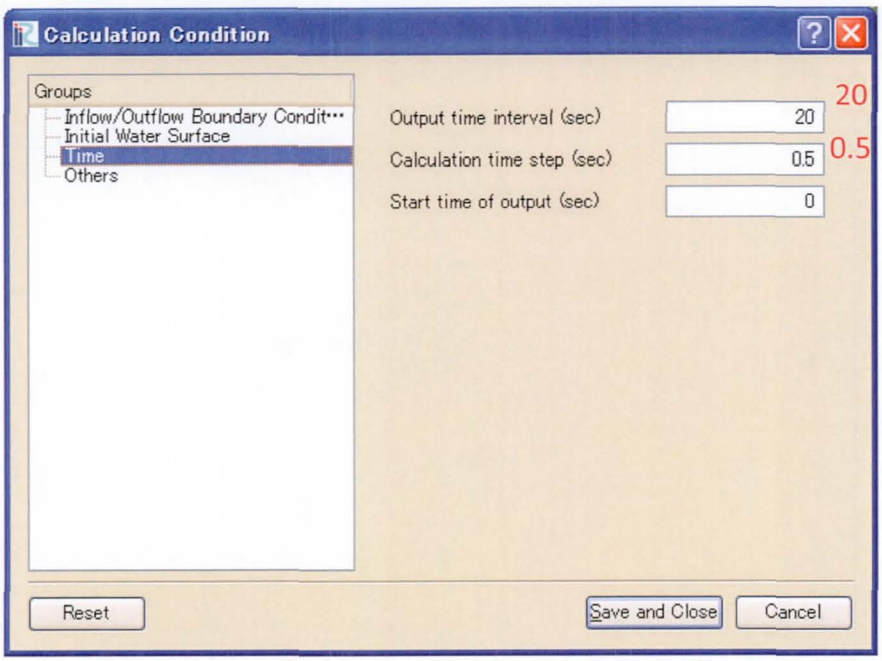
- Click on [Initial Water Surface] from the [Group] list to make the following settings:



Setting the calculation conditions #5

1. **"Calculation Conditions"**

- Click on [Time] from the [Group] list to make the following settings:
- Conditions in [Others] are not changed.



Setting the calculation conditions #6

2. **Setting inflow rivers**

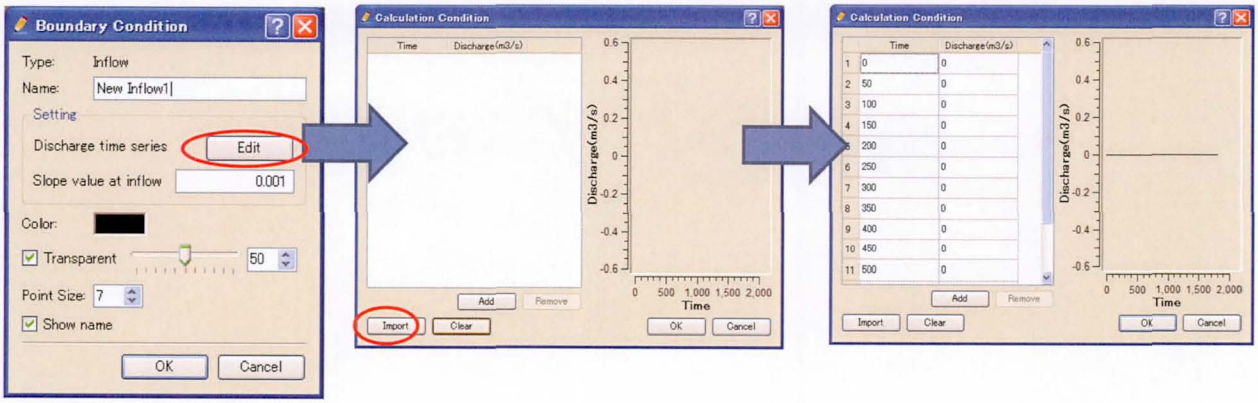
Additional setting of inflow condition

- In the Object Browser, select [Boundary condition setting]-[Add inflow]
- While selecting the [Inflow], enclose the sides of "Inflow" locations with polygons.
- When the location is confirmed, the [Boundary condition] setting dialog opens.

Setting the calculation conditions #7

2. Setting inflow rivers

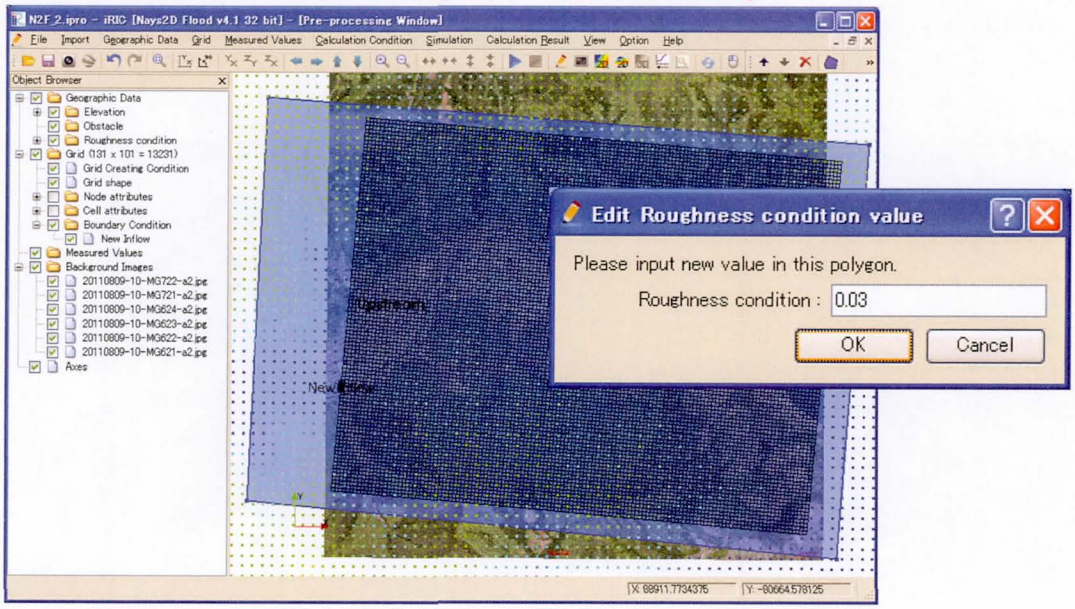
- On the [Boundary Condition] window, input "New Inflow 1" to the [Name] box, and select [Edit] for the [Discharge time series] box under [Setting].
- Click on [Import] on the [Calculation Condition] window.
- Open [¥SampleData¥N2F], select [Q01.txt] and click on [Open].



Setting the calculation conditions #8

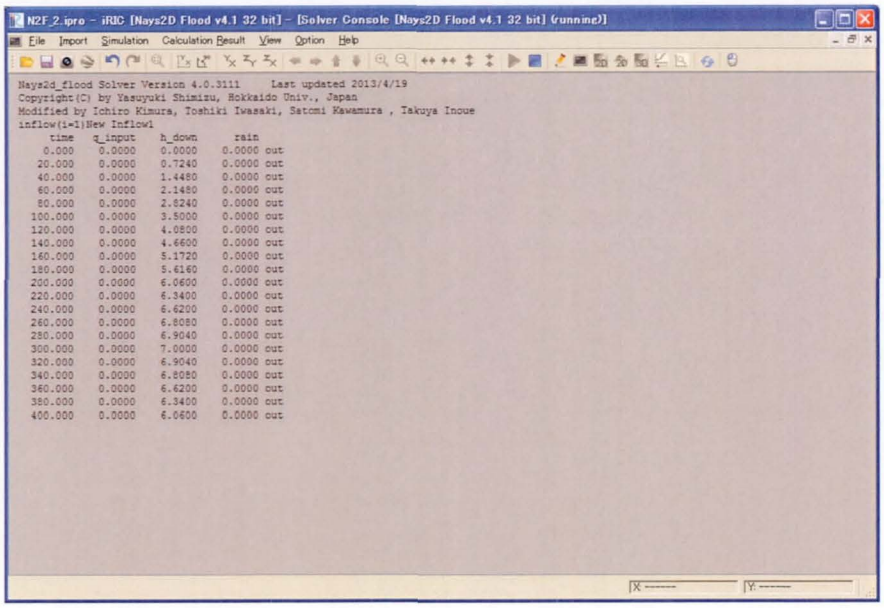
3. Setting the roughness

- In the Object Browser, select [Geographic Data] - [Roughness condition].- [Add] – [Polygon]
- Enclose the all grids with polygons
- Set the obstacle from [Edit Roughness Condition Value].



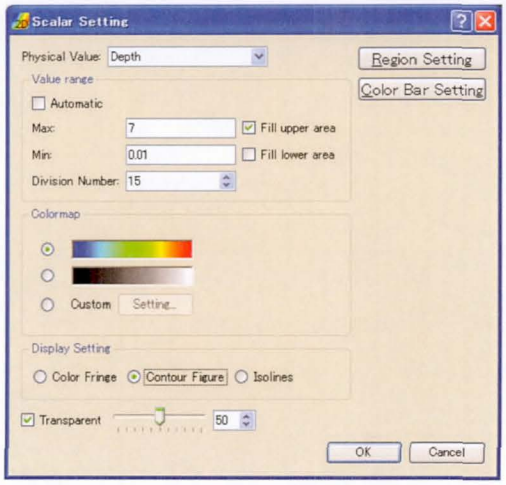
Making a simulation

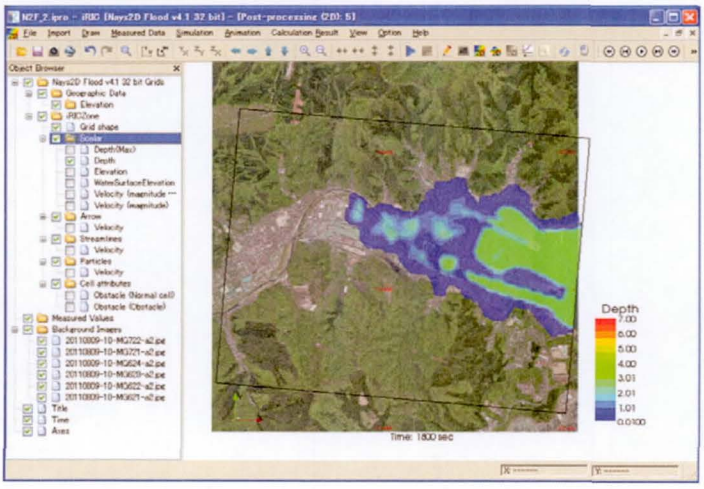
- On the menu bar, select [Simulation] - [Run].
- Save the project.
- The [Solver Console [Nays2D Flood] (running)] window will open to start the simulation.



Visualizing the calculation results

- On the menu bar, select [calculation results] - [Open New 2D Post-Processing Window].
- In the Object Browser, select [Nays2D Flood Grids] - [iRICZone] - [Scalar] - [Depth] by making a check mark in each box.
- In the Object Browser, select [Nays2D Flood Grids] - [iRICZone] - [Contour]. Right click on [Scalar] to select [Property].
- On the [Scalar Setting] window, make the following settings and click on [OK]:





End.

iRIC Software

TUTORIAL 2: Green River, UT

This tutorial shows how to prepare the data necessary to run the SToRM model within the iRIC 2.3 graphical modeling system (henceforth called iRIC only, for simplicity). Data collected in the Green River, Utah, is used to illustrate the application. An aerial view of the region of interest is shown below.

Contents:

1 ..... Getting Started
2 ..... Preparing Bathymetric Data
3 ..... Creating the Computational Mesh
4 ..... Defining Inflow and Outflow Boundaries
5 ..... Roughness and Initial Conditions
6 ..... Running the Model
7 ..... Visualization of Results
8 ..... Refining the Solution
9 ..... Thinning Areas of Dry Nodes
10 ..... Model Calibration

1. Getting Started

Using iRIC and SToRM in a productive work environment usually results in an iterative cycle in which successive model runs hone into a progressively more accurate—perhaps the term refined is more appropriate here—solution, each benefitting from feedback from the previous ones. The workflow of an iRIC/SToRM project and consists in the following schematic steps:

1. Import bathymetric data into iRIC.
2. At this point ancillary data can also be imported into iRIC. Ancillary data may consist of items such as field measurements and orthorectified aerial imagery.
3. Analyze and edit the data.
4. Prepare a computational grid for use by SToRM.
5. Define initial and boundary conditions, including roughness factors and other driving parameters.
6. Specify model run parameters, such as choice of numerical scheme(s) and duration of the run.
7. Run SToRM. May take minutes (go get a cup of coffee), hours (nap time), or days (don't less the boss know the computer is doing all the work).
8. Analyze the model results, compare with measurements, all while exercising critical engineering judgment.
9. Are model results grid-independent? If yes, proceed to the next step; if not, backtrack to steps 4.
10. Is the model properly calibrated? If yes, proceed to the next step; if not, backtrack to steps 5 or 6.
11. Are model results satisfactory? Do they pass critical scrutiny? If yes, proceed to the next step; if not, backtrack to steps 4, 5, or 6.
12. Production run completed.

This set of tutorials, of which this is tutorial number two, aims at teaching how to use the technical tools comprised in iRIC 2.3 to accomplish the tasks described above.

2. Preparing Bathymetric Data

The first step consists in importing the topographic data into iRIC. The data used in this tutorial is located in file GR.tpo. GR.tpo is an ASCII editable file that contains the coordinates and elevation of each data point. To load this data into iRIC:

- Use pull-down command **Import | Geographic Data | Elevation** from the main menu bar.
- Find the data file in the folder where it is located and click OK in the pop-up window.

The final result is shown in Fig. 2.1. The individual survey points are plotted, colorized by bed elevation. (Note about notation used in this tutorial: bold faced words separated by a vertical bar denote a cascade of menus and submenus in iRIC, or a hierarchy of objects in the **Object Browser**.)

Note the objects in the **Object Browser** window (Fig. 2.2). Among other things (which will be seen later), this window can be used to turn specific aspects of the data on or off in the main display canvas. For example, click on the checkbox to the left of **Points1** to uncheck it and the data will disappear. Click on it again to check it and the data will reappear. This is true for all the graphic entities created during the course of this tutorial. The color range is calculated automatically from the data, but it can be changed from an appropriate control window (Fig. 2.3, left), which is brought up by highlighting **Points1** in the **Object Browser** window (highlighting of any item in the **Object Browser** window is accomplished by left-clicking the mouse on the

desired object) and then right-clicking the mouse on **Geographic Data | Elevation** and selecting **Color Setting** from the pop-up menu. The same pop-up menu has other functionality, such as controlling the display a color legend. The use of this menu is simple and straightforward.

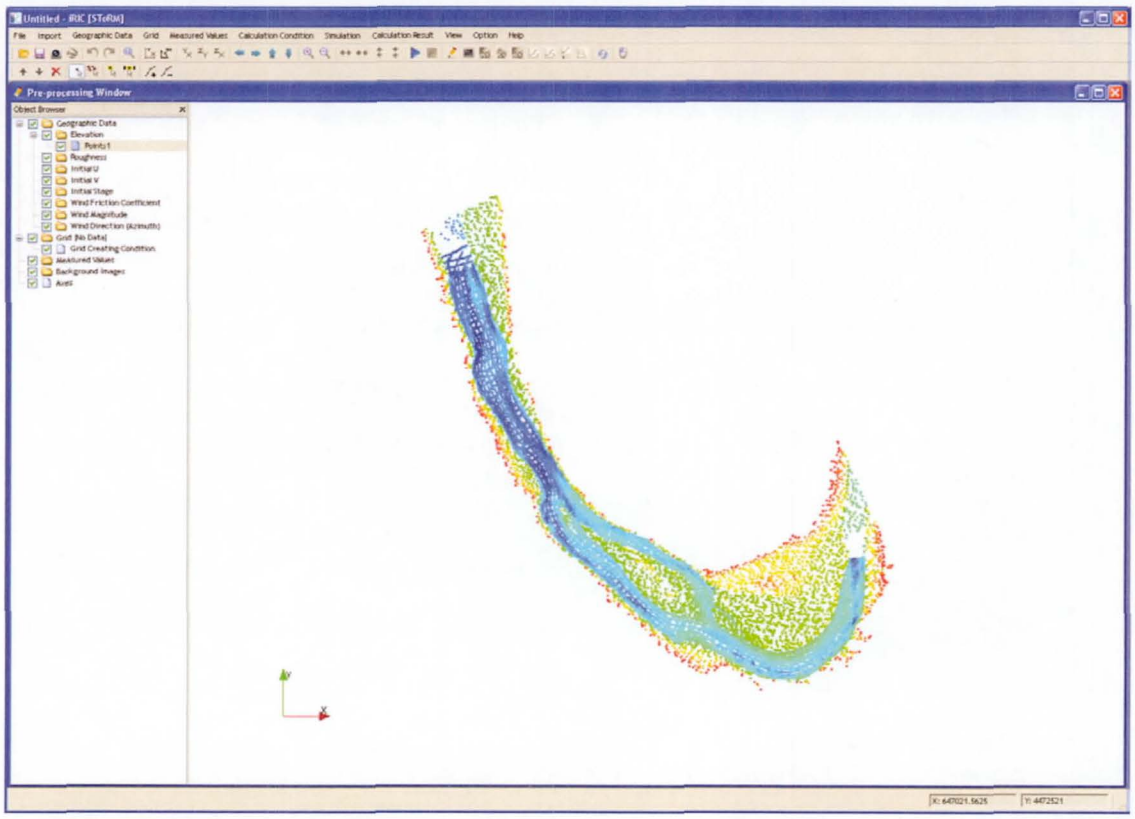


Figure 2.1

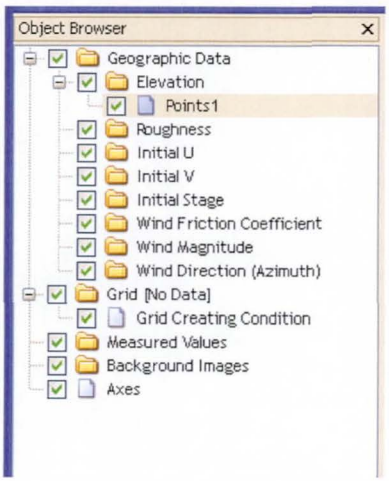


Figure 2.2

Right-clicking the mouse on **Points1** directly brings up another pop-up menu with different functionality. For example, it is possible to change the name of the dataset (**Points1** is the name

assigned to the data by default) and the points' appearance. To change how the topographic data is plotted:

- Right-click the mouse on **Geographic Data | Elevation | Points1** and select **Property**. The window in Fig. 2.3 (right) will appear. Click OK when finished.

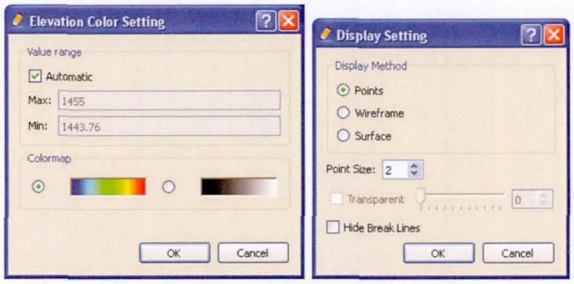


Figure 2.3

It is possible to use geo-referenced images and plot them under the topographic data. This provides a much more interesting visual feedback when working with the data. To import a background image:

- Select **File | Import | Background Image** from the main menu. Alternatively, **Import | Background Image** can also be used.
- In the **Open Image** dialog window, select GR with WS.jpg.

The image is now displayed in the main drawing canvas under the topographic data, as shown in Fig. 2.4. Note that displaying this image is optional: its display can be turned off by unchecking **Background Images | GR with WS.jpg** in the **Object Browser** window. Turning the topographic data set off (by unchecking **Geographic Data | Elevation | Points1**) shows the background image without the data points. The blue circles indicate the locations where measurements of the water surface elevation were made; these measured data will be used later in this tutorial.

At this point it may be convenient to save the project. To save the project:

- From the main menu, chose **File | Save as Project**. Never choose command **Save as File (*.ipro)** when using SToRM.
- A pop-up window (Fig. 2.4) lets you select the location where the data will be saved. Select the desired location and click on **Make New Folder**. Assign it a name and click OK.
- This operation creates a new folder containing all the data used by iRIC and by SToRM. The data used later by SToRM is saved in a CGNS-formatted file with extension .cgn. CGNS (CFD General Notation System) is a data format used widely in the computational fluid dynamics community.

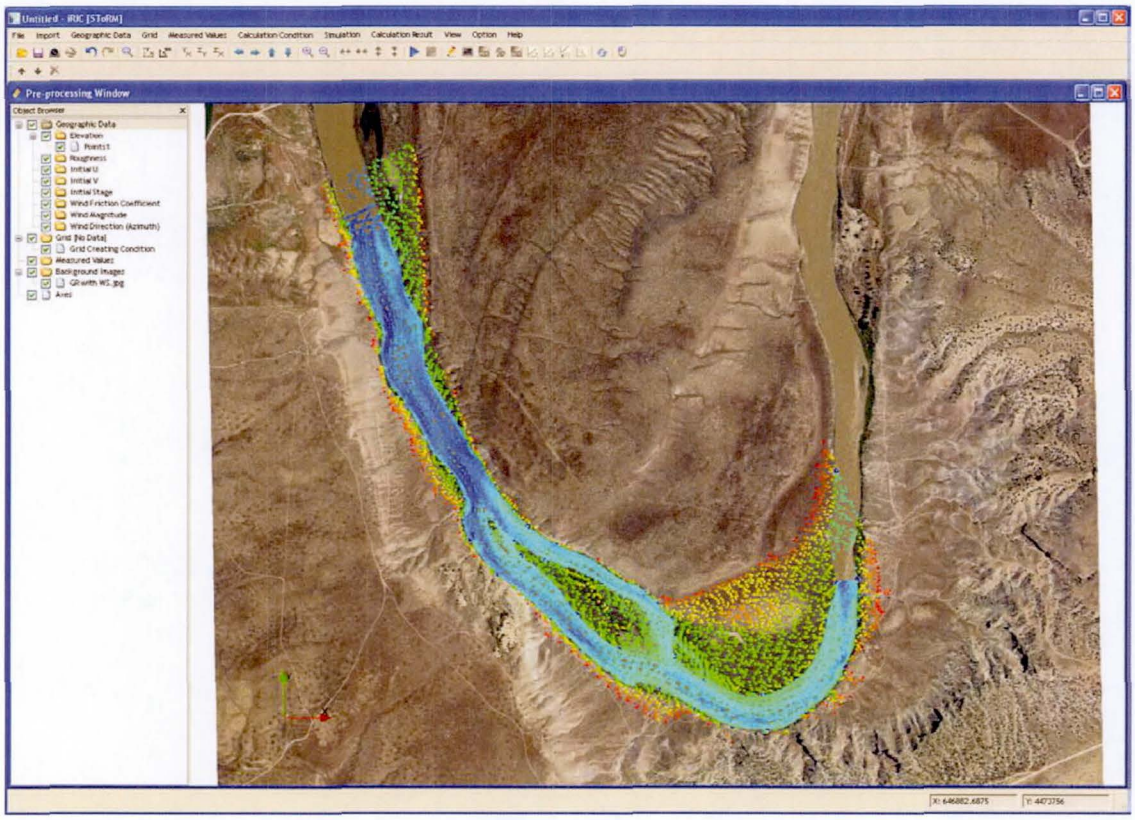


Figure 2.4

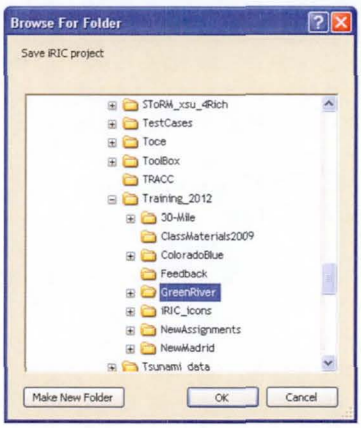


Figure 2.5

3. Creating the Computational Mesh

In this exercise, the area of interest is comprised of the elbow and the island located West of it. Zoom and pan to that area, which can be accomplished by using buttons in the main toolbar. Zooming can also be done by rotating the mouse wheel and panning by right-clicking the mouse and moving it on the display canvas while pressing the <Ctrl> key. After zooming and panning to the desired area, iRIC will look like what is displayed in Fig. 3.1.

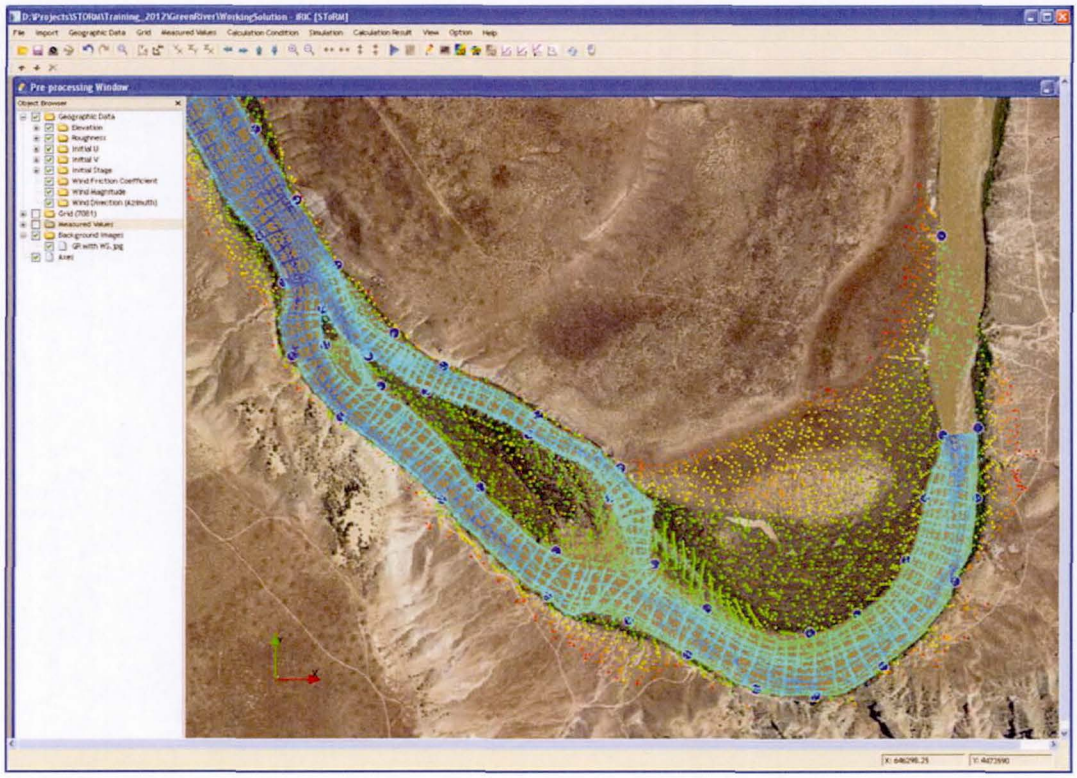


Figure 3.1

In iRIC, the computational grid is created automatically by a triangulation program using a few parameters defined by the user. These parameters are: a polygon encompassing the area of interest, maximum area of the triangles, and their minimum allowed internal angle. The steps needed to create the grid are:

- From the main menu bar, use command **Grid | Select Algorithm to Create Grid...** and select **Create grid from polygonal shape** by clicking on it.
- The computational domain is defined by a polygon that encompasses the area of interest. Define it by left-clicking the mouse at the desired polygon vertices, in the proper sequence. iRIC provides guiding visual feedback during this process. Double-click the last point or press <Enter> to finish the polygon. Use Fig. 3.2 to guide you on the shape and location of the polygon.
- Right-click on **Grid[No Data]** in the **Object Browser**. This starts the grid generator.
- The **Grid Creation** window is used to define the basic parameters that control grid quality. Enter the values shown in Fig. 3.3 (maximum angle for cell vertices: 32 degrees; maximum area for cells: 73 m^2).
- When prompted to map geographic data to grid, click **Yes**. This operation transfers the topographic data to the computational grid.

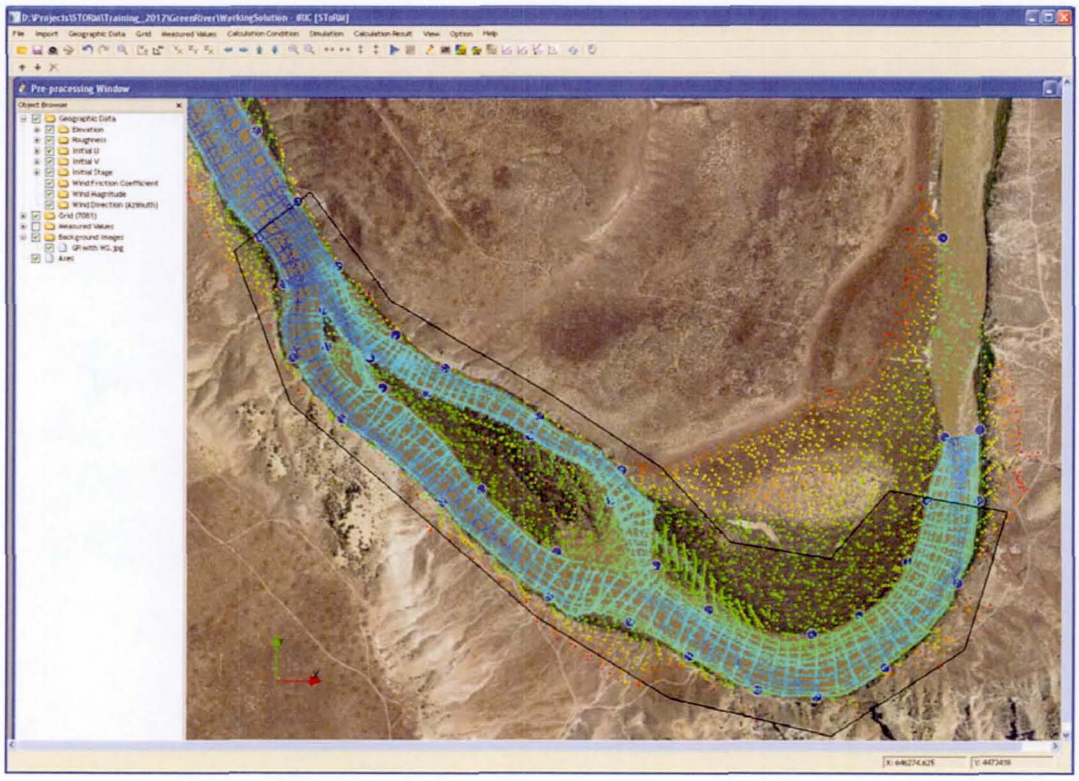


Figure 3.2

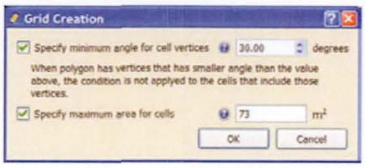


Figure 3.3

The result of these steps is shown below in Fig. 3.4, where the background figure display was turned off to help visualizing the grid. The computational mesh was created within the shaded polygon. The flow is from East to West. Build the polygon to extend the reach of interest. Narrow sections away from bends are good locations to use as inflow and outflow boundaries because they have stronger and somewhat straighter velocity profiles than wider areas. It is a bad idea to place these boundaries near zones of recirculation.

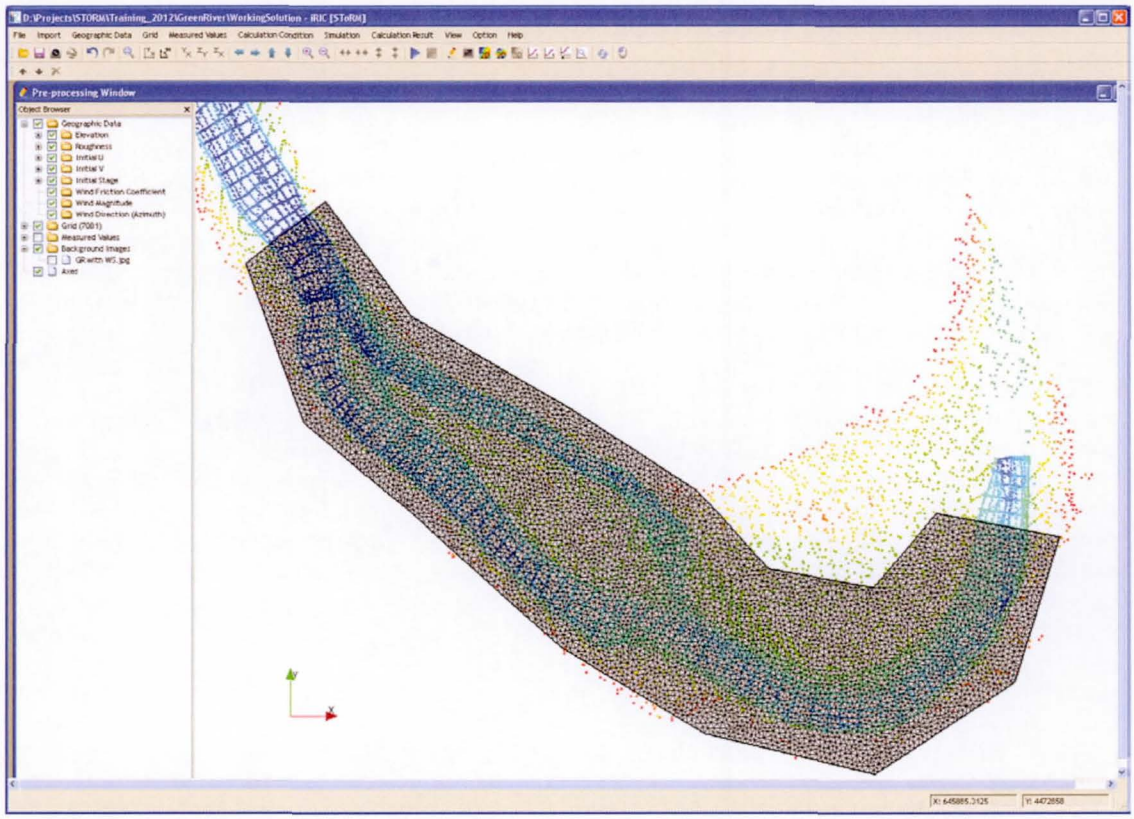


Figure 3.4

The objective of the internal angle constraint is to avoid creating triangles that are long and narrow, which have poor interpolating qualities and can impact negatively the accuracy of the model's numerical solutions. Ideally, the value chosen should be close to 60 degrees, which are the internal angles of an equilateral triangle. Unfortunately, the grid generator may become unstable for an angle greater than 30 degrees, therefore the use of this value is recommended.

The maximum triangle area constraint was determined from considerations about the required resolution. Querying the topographic data indicated that the river width at the section entering the computational grid is approximately 130 m. A good rule-of-thumb is to have a starting mesh with at least 10 grid segments across any area of interest (in this case, across the river width). Calculations on finer grids are more accurate than on coarser ones, but take longer computer run times. A good compromise is sought, where there are enough grid points for accurate solutions without the computational burden of a finer discretization. This determines that a good size for the edges of the triangles may be 13 m (= 130/10). The formula for the area of an equilateral triangle whose edges are Δx in length is

$$A = \frac{\sqrt{3}}{4}\Delta x^2$$

Using this expression with Δx = 13, an approximate triangle area of 73 m^2 is found.

4. Defining Inflow and Outflow Boundaries

The computational domain defined by the grid is the only part of the river "seen" by SToRM during the calculations. In other words, the model does not have any sort of information about the region outside the computational grid. To be able to carry out the backwater calculations, it is necessary to know something about what is upstream and downstream from the computational region. Basic hydraulics principles require that, for subcritical river flow, the discharge entering the grid and the water surface elevation at the downstream-most region of the grid must be known. These regions, where the stage and discharge must be specified, are usually called the hydraulic boundaries of the domain. The quantities prescribed there (discharge and stage) are called the boundary conditions. This section illustrates how to define the boundary conditions in iRIC.

First, we are going to define the inflow boundary. This can be thought of as being the upstream-most cross section in the computational domain, and the flow discharge must be specified there. It is convenient to define this boundary as a straight line perpendicularly to the centerline of the river channel. This is done when defining the main boundary polygon for the mesh, but the appropriate nodes need to be assigned to the inflow boundary. Each inflow (and outflow) boundary is composed of a string of contiguous nodes located on the boundary of the computational grid. The steps below show how to mark the inflow nodes and assign an inflow boundary condition to them.

Zoom to area of interest, as shown in Fig. 4.1, and:

- Right-click on **Boundary Condition Setting** item in the **Object Browser** and select **Add Inflow** in the pop-up window.
- Define a polygon that encompasses all the grid nodes (triangle vertices) belonging to the inflow boundary. Do not skip nodes. End the polygon by pressing <Enter> on the keyboard.
- In the ensuing **boundary Condition** window, assign the inflow condition a name and click **Ok**. Here, we use **Inflow** (Fig. 4.2, left).
- Select **Fixed Width** in the **Type of inflow boundary**.
- Enter the time series for the inflow discharge by clicking **Edit**. In this example, only one value is entered (Q = 247 m^3/s), because we seek a steady state flow solution (Fig. 4.2, right). Click on the button **Add** to add a row on the left (white) window, for data entry. Click on the respective time and discharge cells to enter the numerical values. Click **Ok** when done.
- The defined inflow boundary is shown in Fig. 4.3.
- If the grid has been rotated, the display can be returned to its original state of zero rotation by clicking on the button located in the main toolbar.

Fig. 4.1 shows the grid colored by bed elevation. This is accomplished by checking **Grid | Node attributes | Elevation** in the **Object Browser**.

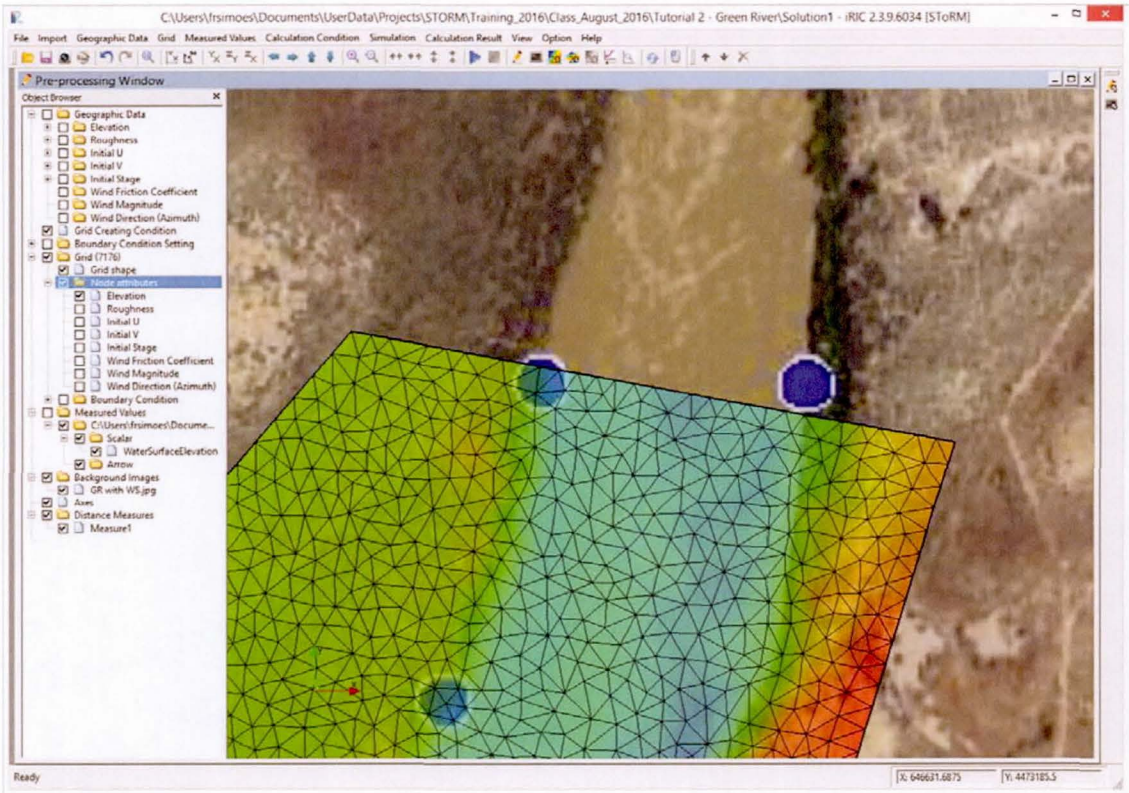


Figure 4.1

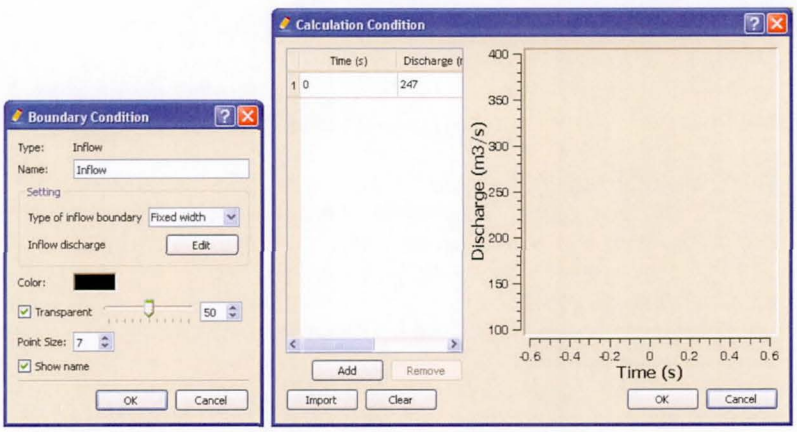


Figure 4.2

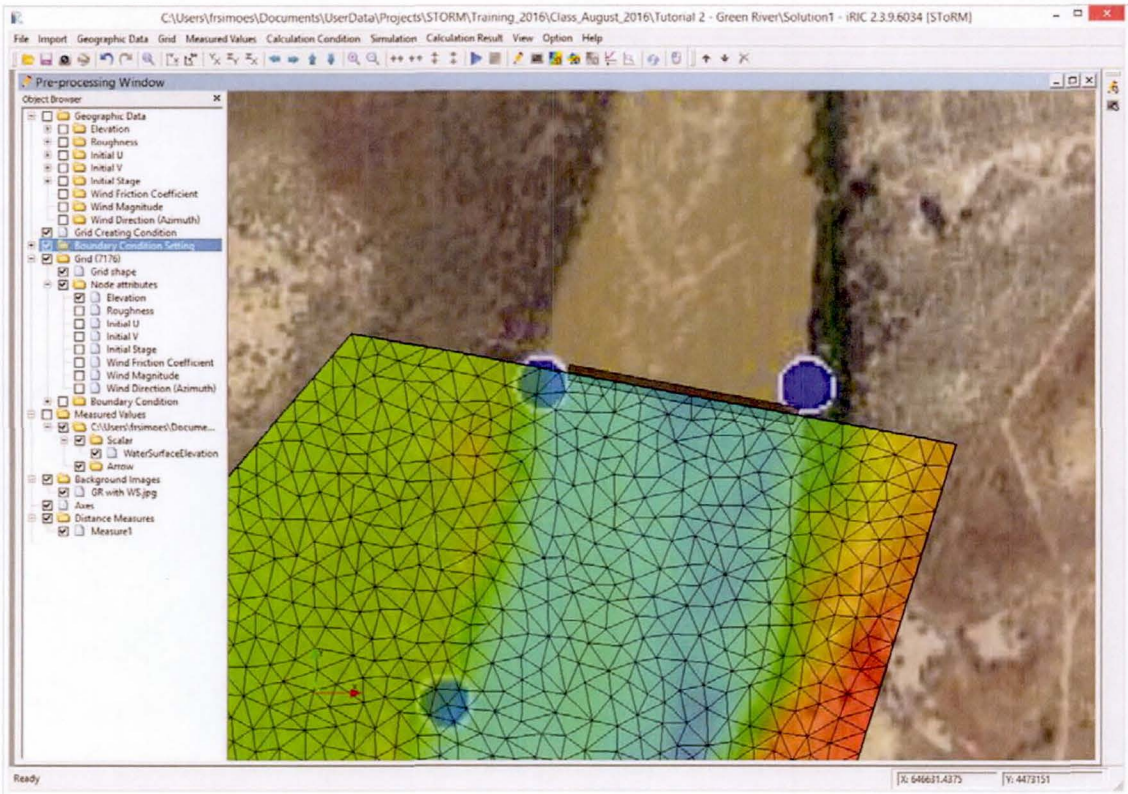


Figure 4.3

The inflow discharge is entered at a time series, with time (seconds) in the left column and discharge (m^3/s) in the right column. In this case there is only one constant value of the discharge, therefore time data is not important, and a value of 0 (zero) was entered.

In this example, the discharge is distributed uniformly across the string of nodes defined as the inflow boundary. A synthetic flow velocity field is then generated with velocity vectors that are perpendicular to the string. Due to the artificial nature of this velocity distribution, it is recommended that this boundary should be placed away from the areas of interest, i.e., from regions where accurate solutions are sought. Try not to include nodes that are thought to be dry or, at least, try keeping them to a minimum. This may require a trial-an-error procedure, where successive runs are performed with increasingly accurate boundary node strings. This process is illustrated in Fig. 4.4, which depicts three inflow boundaries, with the computational grid in gray, the physical river margins in thick black lines, the computed river margins in thinner black lines, and the arrows showing flow paths. In (A), too many nodes (small squares) are assigned to the inflow boundary, resulting in an artificially larger river width. In addition to the inaccuracy, potential problems associated with this type of boundary are caused by large flow velocities on shallow depths over the margins, which may induce numerical instability and require unnecessarily small time steps to march the solution in time (time stepping is discussed in a later section of this tutorial). The opposite circumstance is shown in (B), where too few inflow nodes cause the flow to accelerate in the middle of the channel and produce spurious eddies near the river margins. These eddies occur because triangle edges at the border of the grid are treated as solid walls, therefore the flow is computed as if there are small groynes blocking the flow at each side of the inflow cross section. Finally, the ideal boundary string is shown in (C): it does not extend past the river margins, neither does it fall short.

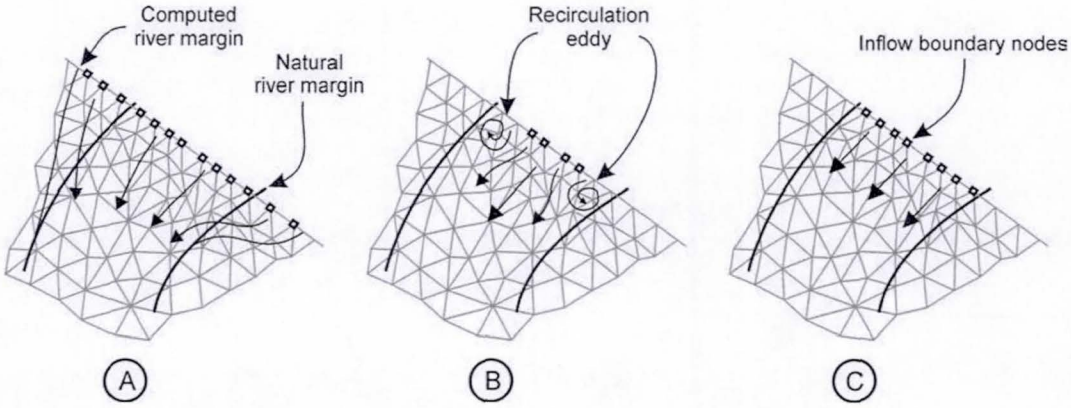


Figure 4.4

To deal with the issues described in the previous paragraph, SToRM offers two different implementations of the inflow boundary condition. The first (used in the example in this tutorial) uses the static inflow boundary, where the flow is distributed along every segment in the boundary node string, as described in the previous paragraph. This option is the **Fixed Width** in the **Type of inflow boundary** option (Fig. 4.2, left). This option is useful for those cases where the top width of the inflow cross section does not change significantly during the hydrograph, but it may require trial-and-error to set properly (as shown in Fig. 4.4). The second type of inflow boundary condition is the **Variable Width** in the **Type of inflow boundary** option (Fig. 4.5). When this option is selected, of all the inflow nodes in the inflow boundary string, only the boundary edges associated with wet triangles will have inflow. In this case, the inflow cross section may widen when the stage at the inflow rises, and contract when it lowers, which may be useful for inflow hydrographs with large amplitude, i.e., with large magnitude changes. Its disadvantage is that it cannot be used to start simulations of the flooding of dry areas. It may also cause spurious oscillations in the solution due to alternate wetting and drying of triangles at the edges of the inflow cross section.

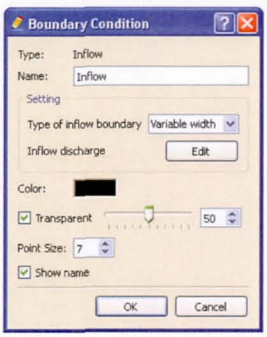


Figure 4.5

The outflow boundary can be thought as the downstream-most cross section of the river reach, and is where the water surface elevation (stage) is needed for the backwater calculations. This boundary is defined in similar steps as those used to define the inflow boundary:

- Zoom to the area of interest, which is the northwest end of the computational grid (Fig. 4.6).
- Right-click on **Boundary Condition Setting** item in the **Object Browser** and select **Add Outflow** in the pop-up window.
- Specify the outflow boundary nodes using a bounding polygon, as it was done for the inflow boundary.

- Assign the inflow condition a name and click **Ok**. Here, we use **Outflow** (Fig. 4.7, left).
- Enter the time series for the outflow stage by clicking **Edit**. This process is similar to the one used to define the inflow discharge time series: click on **Add** to create a data entry row and click on each cell to enter the corresponding value.
- In this example, only one value is entered (1447.4 m), because we seek a steady state flow solution (Fig. 4.7, right). Time is, therefore, not important and the value 0 (zero) is used here. Click **Ok** when done.
- The defined outflow boundary polygon is shown in Fig. 4.6.

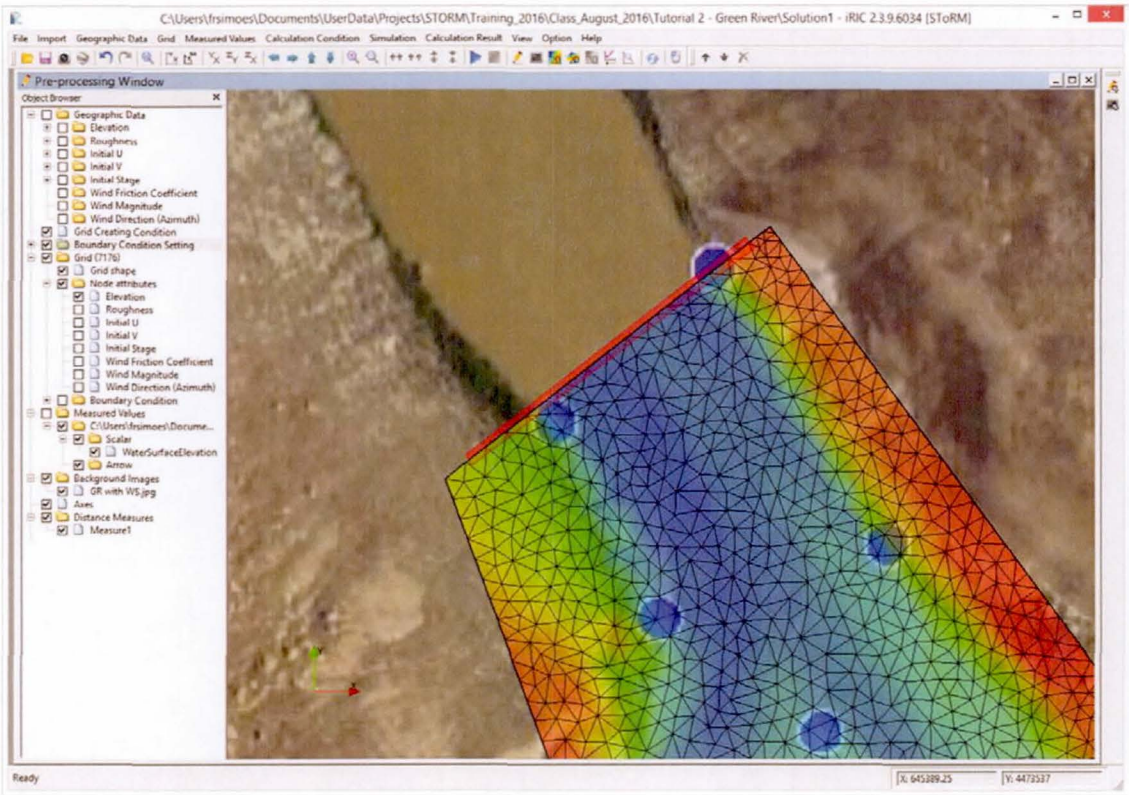


Figure 4.6

Outflow boundaries need not be defined as rigorously as inflow boundaries, and assigning nodes that are known to be dry is acceptable. Outflow boundaries that are too narrow, however, will have similar issues with flow obstruction and unrealistic recirculation areas. Ultimately, it is better to have an outflow boundary that is too wide than one that is too narrow. The outflow boundary should be defined as a cross section placed perpendicularly to the flow and away from recirculation zones. Ideal places are gaging stations in the river, where stage information is known and can be used as boundary condition. However, in most cases that is not be possible, in which case the data collection program should include the surveying of appropriate water levels for the discharges of interest.

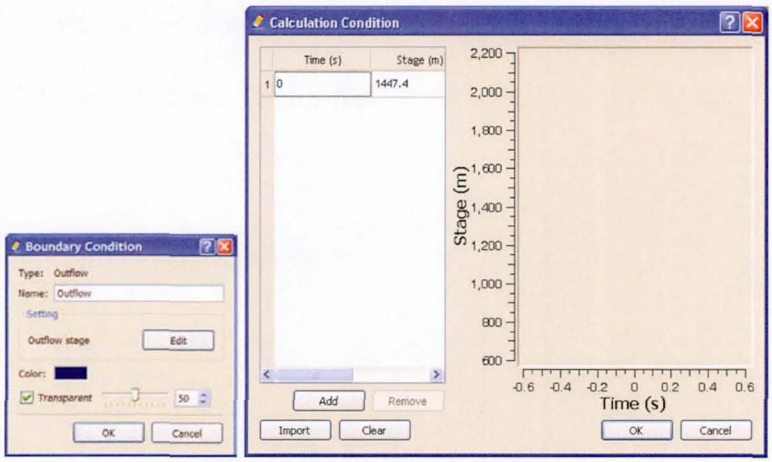


Figure 4.7

It is relatively easy to make an error creating a boundary-node string. Any string can be deleted by right-clicking on its name in the **Object Browser** and selecting **Delete** in the corresponding pop-up menu. Note also that the inflow and outflow boundaries are grid nodes defined on the border of the computational grid. Any string can be defined as inflow or outflow, and it is up to the user to define them correctly to ensure that the flow is in the right direction.

After all the boundaries are created, map the boundary polygons to the grid by executing the command **Grid | Attributes Mapping | Execute...** from the main menu. The boundary nodes will not be properly identified by SToRM until this step is completed.

This would be a good point to save your work.

5. Roughness and Initial Conditions

Roughness is set by drawing polygons (sometimes called coverage polygons) over the areas of interest. Follow these steps to set the roughness:

- In the **Object Browser**, under **Geographic Data** right-click on **Roughness** and select **Add** and **Polygon** in the respective pop-up menus.
- Draw a polygon covering the entire computational grid. The polygon is entered by left clicking the mouse on the canvas to enter each vertex of the polygon, in sequence. Finish by double-clicking the last vertex of the polygon (or by pressing <Enter>).
- Enter the value of the roughness, as in Fig. 5.1.
- Map the roughness values to the computational grid by executing the command **Grid | Attributes Mapping | Execute** from the main menu.
- The final polygon may look like what is shown in Fig. 5.2.

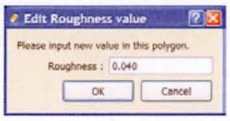


Figure 5.1

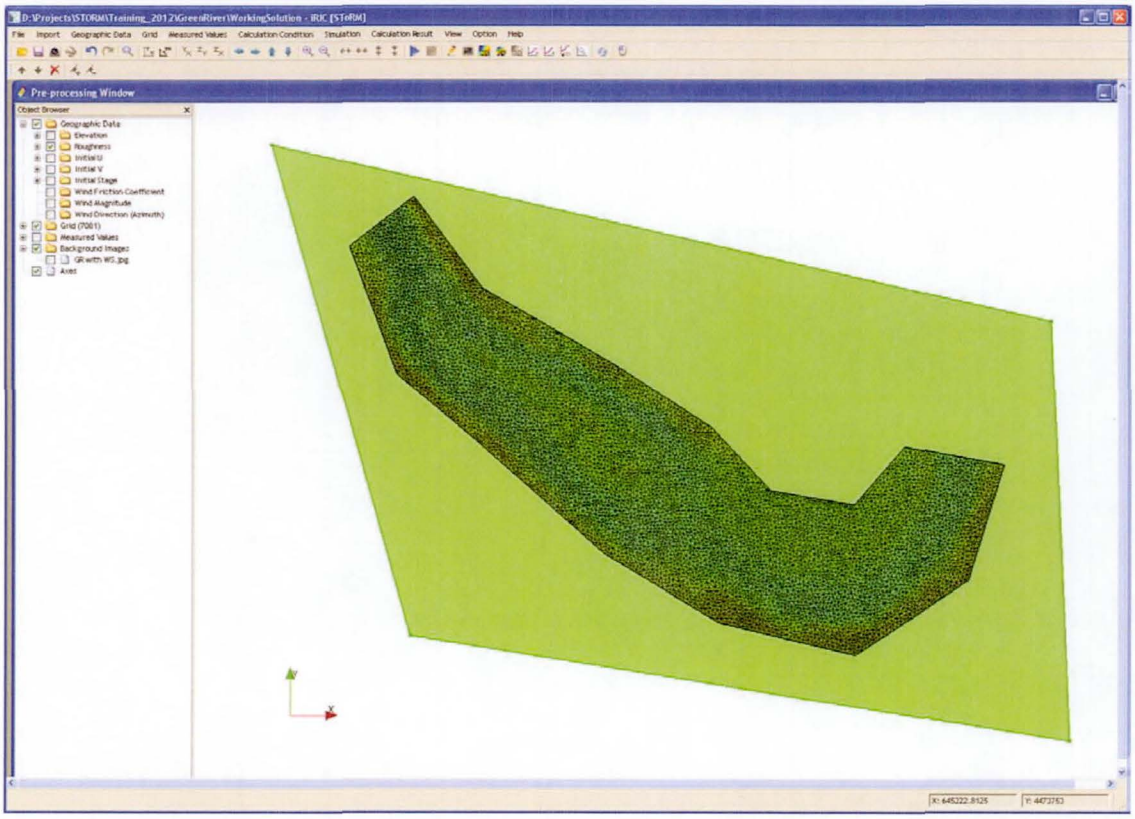


Figure 5.2

In SToRM, roughness can be represented by Manning's n values, by Chézy's C coefficients, or by drag coefficients c_f. The default is Manning's n, which is used in this tutorial, but it will be shown later how to select the others. There is a conceptual difference between roughness in one-dimensional and in two-dimensional modeling frameworks. For example, recirculation areas are not modeled by one-dimensional models, therefore their contribution to flow resistance is incorporated in a bulk roughness factor n, which also includes flow resistances from many other phenomena. Two-dimensional models, however, are able to calculate recirculation areas with greater detail, therefore their effects do not need to be included in n. The same happens for a number of other phenomena (such as super-elevation), which results in the same river cross sections having different values of n depending on the model used. In general, the Manning's roughness values are lower in two-dimensional models than in the one-dimensional counterparts.

The term "initial conditions" reflects the state of the flow at the beginning of the simulation, i.e., at time $t = 0$ s. A simulation run starts with a known flow field, which is a known velocity and water depth throughout, and advances it in time (also called time marching or time stepping in the modeling community). Often times, the initial flow field is unknown and it must be guessed in an appropriate manner. In steady state computations, a good initial guess is closer to the final solution, therefore needing shorter run times to reach it. In this tutorial, a number of polygons are used to define a set of initial conditions to run SToRM.

One of the ways that can be used to define initial conditions is by the use of coverage polygons, identical to what is done above for roughness data entry. For each quantity U (component of the flow velocity along the x axis), V (component of the flow velocity along the y axis), and η (water surface elevation), a polygon is used to define a region with a uniform distribution of the respective variable. The polygons do not need to be identical for each quantity and may overlap. In the present example, three areas are defined, as shown in Fig. 5.3: the inflow stretch (1), the

elbow region (2), and the island (3). This classification is arbitrary and many others could be devised.

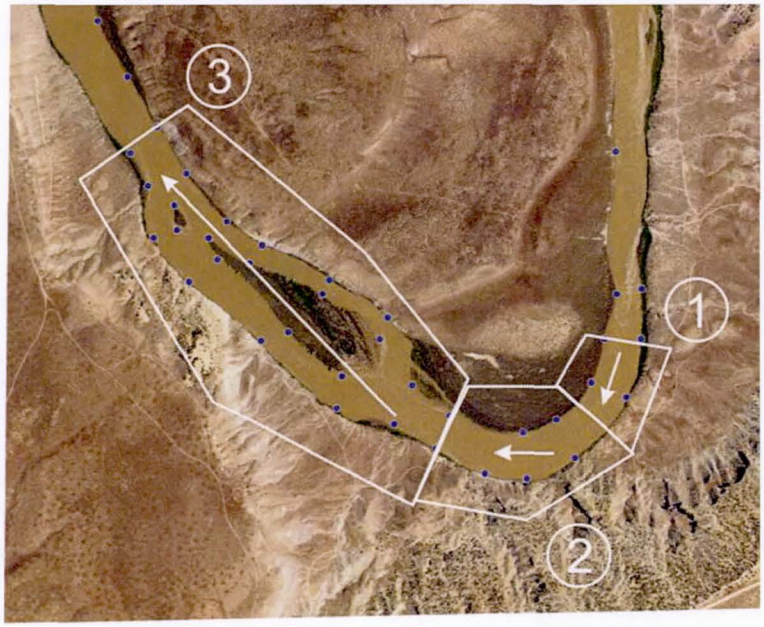


Figure 5.3

The steps to define each polygon are described here using Fig. 5.3 as a guide. For the *U* velocity:

- In the **Object Browser** of the **Pre-processing Window**, right-click on **Initial U** and select **Add** and **Polygon** in the respective pop-up menus.
- Define the first polygon by left clicking the mouse on the locations of its vertices. Double click or press <Enter> to terminate the polygon and assign it a value of -0.50 m/s (Fig. 5.4).
- Repeat the steps to define the second polygon and assign it a value of -1.54 m/s (Fig 5.5).
- And repeat the steps to define the third polygon and assign it a value of -1.09 m/s (Fig 5.6).

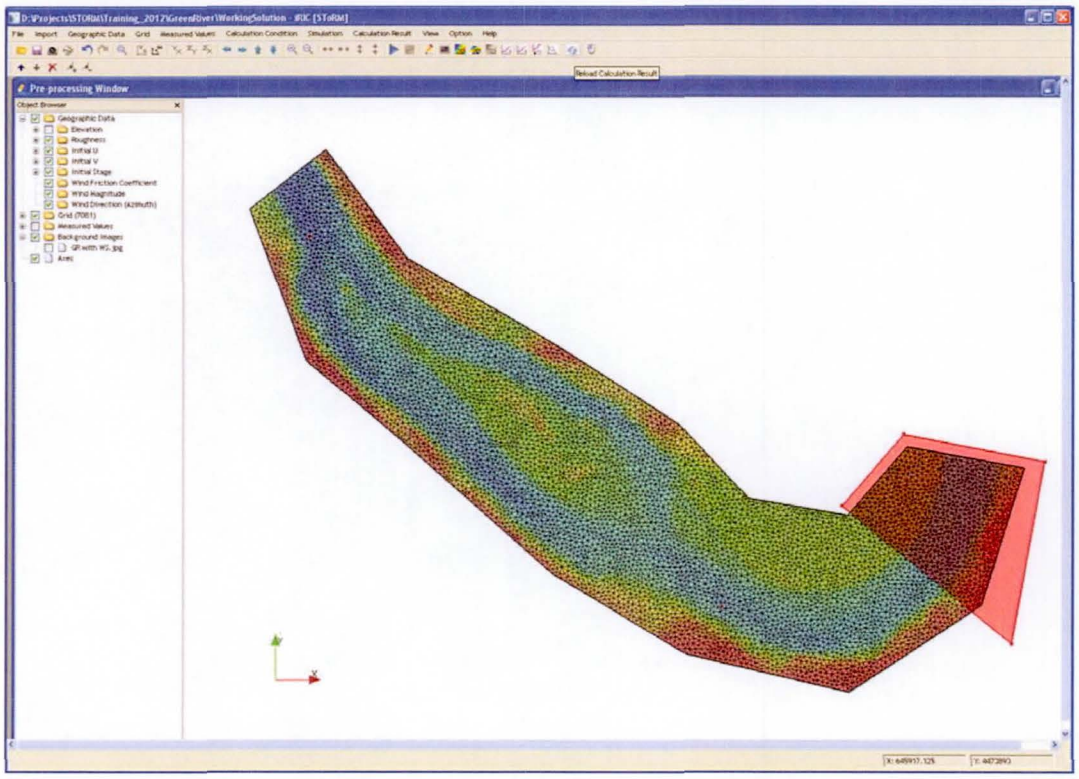


Figure 5.4

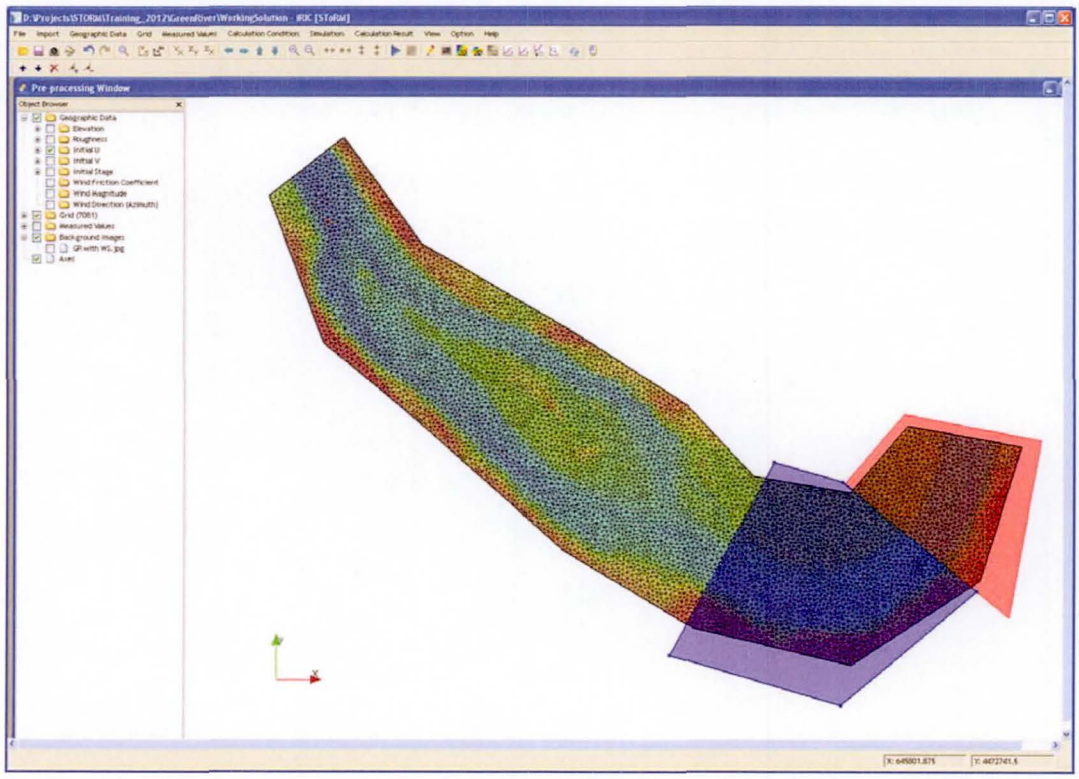


Figure 5.5

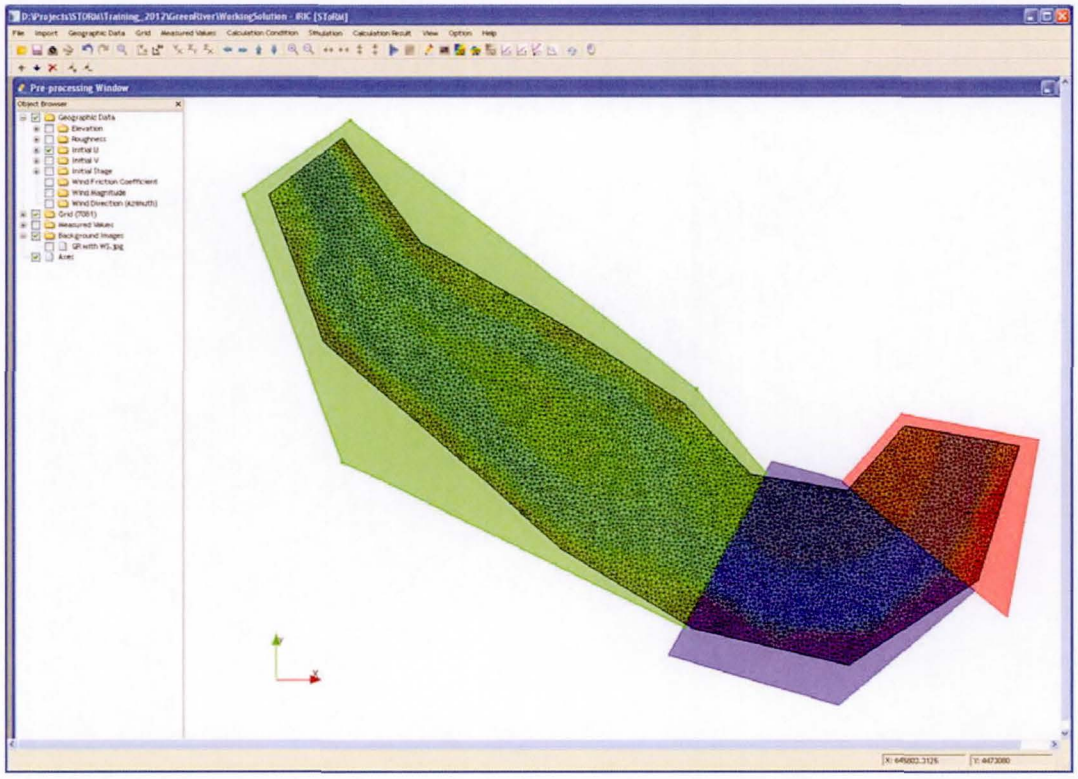


Figure 5.6

Repeat similar steps to define identical polygons for the *V* velocity:

- In the **Object Browser** of the **Pre-processing Window**, right-click on **Initial V** and select **Add** and **Polygon** in the respective pop-up menus.
- Define the first polygon similarly to what was done for the U component and assign it a value of -1.46 m/s (Fig. 5.7).
- Repeat the steps to define the second polygon and assign it a value of 0 m/s.
- Finally, define the third polygon and assign it a value of 1.09 m/s.

Similarly to what is done above for the components of the flow velocity, water surface elevation can be defined in the same three regions of the computational domain. Values of the stage for each area were averaged from existing experimental data.

- In the **Object Browser** of the **Pre-processing Window**, right-click on **Initial Stage** and select **Add** and **Polygon** in the respective pop-up menus.
- Define the first polygon similarly to what was done for the velocity components and assign it a value of 1448.7 m (Fig. 5.8).
- Repeat the steps to define the second polygon and assign it a value of 1448.51 m.
- Finally, define the third polygon and assign it a value of 1447.75 m.

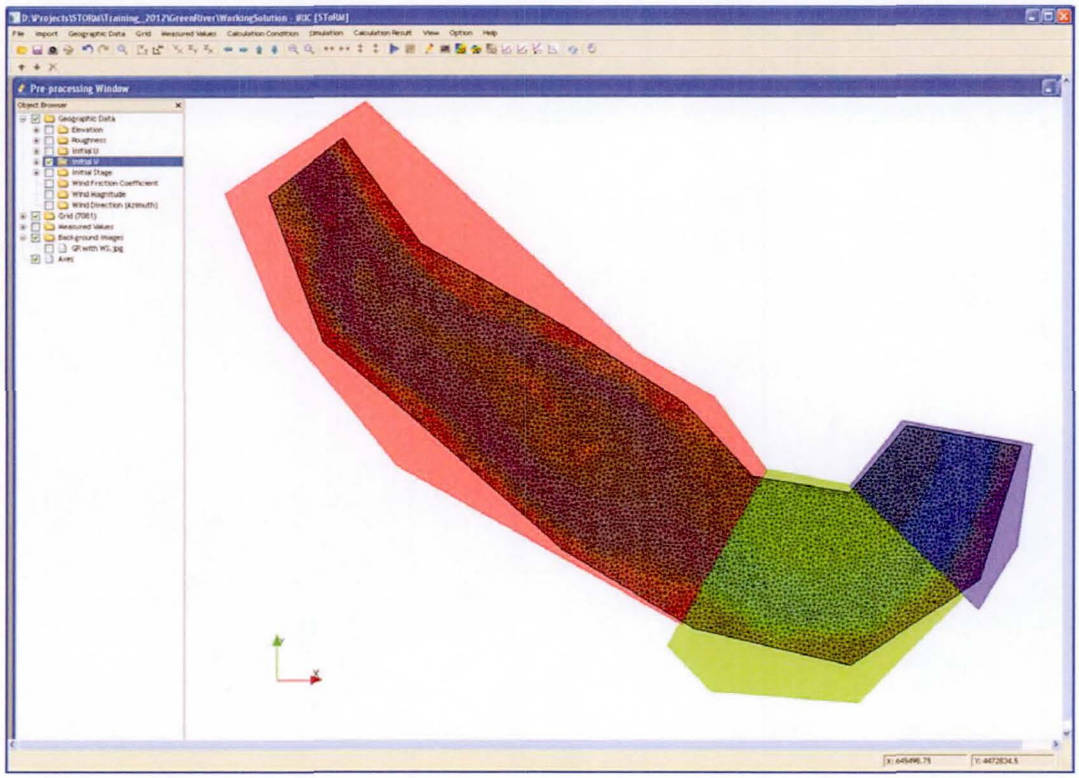


Figure 5.7

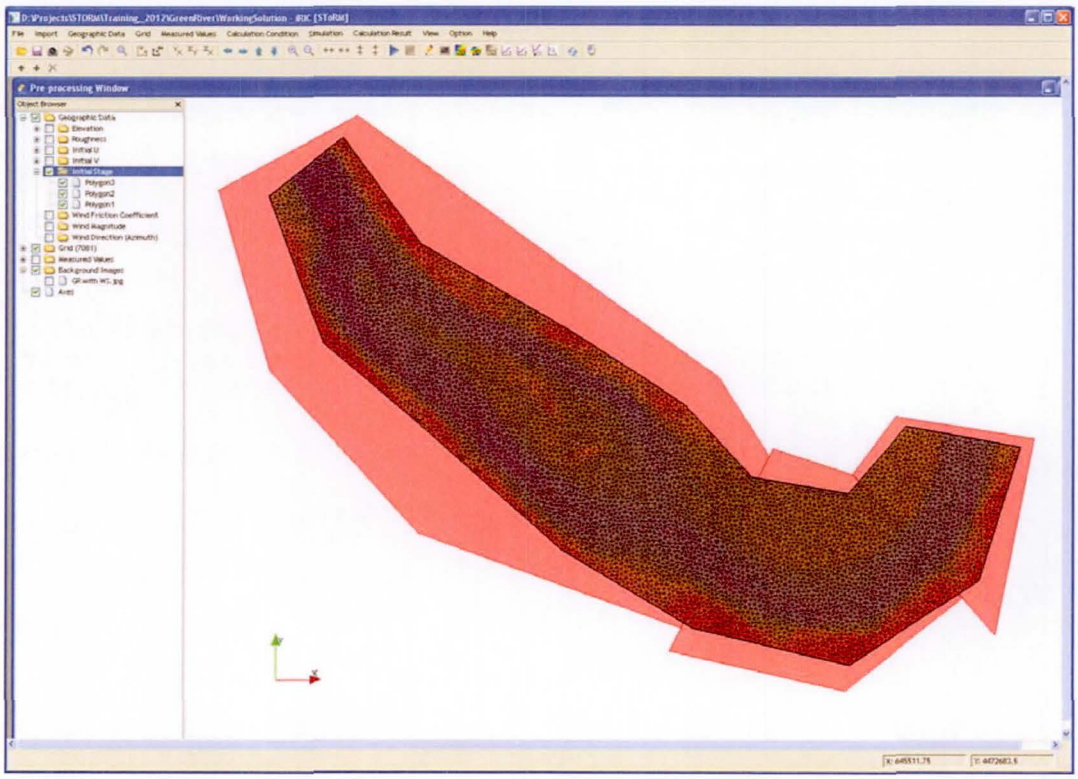


Figure 5.8

Try to make the corresponding *U* and *V* polygons match each other as closely as possible. After all the polygons are defined, transfer their properties to the computation grid points by executing the command **Grid | Attributes Mapping | Execute** in the main menu bar. Save your work.

The values of the water surface elevation used above were determined by averaging measurement data taken inside each coverage polygon. There are no measured velocity data points, therefore the values used in the initial guesses above had to be derived some other way. First, a cross section was cut from the topographic data right at the southern-most tip of the elbow, in region 2 of Fig. 5.3. This location is very close to a measurement point for which experimental data is available, and its top width is representative of a large part of the study reach. The cross section is plotted in Fig. 5.9, showing also the experimental stage. The top width in that section is 135 m, and an estimated average depth is 1.5 m. From the known value of the water discharge (Q = 247 m^3/s, as defined in the previous section), an average value of the mean cross sectional velocity can be found: $U_m = Q/A$ = 1.2 m/s, where A is the cross sectional area.

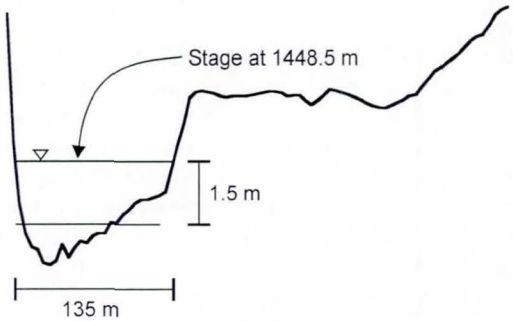


Figure 5.9

Using the flow directions represented by the white arrows in Fig. 5.3, in region (1) the flow makes an angle α of 250 degrees with the *x* axis, 180 degrees in region (2), and 135 degrees in region (3). From the known relations $U = U_m \cos \alpha$ and $V = U_m \sin \alpha$, one can find the values used in this exercise.

Don't forget to save your work.

6. Running the Model

The previous sections defined the region (computational domain) where the simulations are going to take place. Like any other model, SToRM requires the definition of additional parameters. Some of these parameters are physical quantities, others are parameters used to set up the numerical methods used to solve the equations. These must all be defined by the user in a set of entry screens that are going to be covered in this section.

To define the parameters needed for the run, there are a group of entry screens in the **Calculation Condition** window. This window is invoked from the **Calculation Conditions | Settings...** menu from the main pull-down menu bar. The different parameter entry screens are selected by clicking the appropriate text in the left-hand side panel. The first is titled **Initial Conditions** and is shown in Fig. 6.1, left. In this tutorial, the initial conditions were defined using coverage polygons in the previous section, therefore this entry screen must read **From coverage polygons** in the **Set-up initial conditions** pull-down menu.

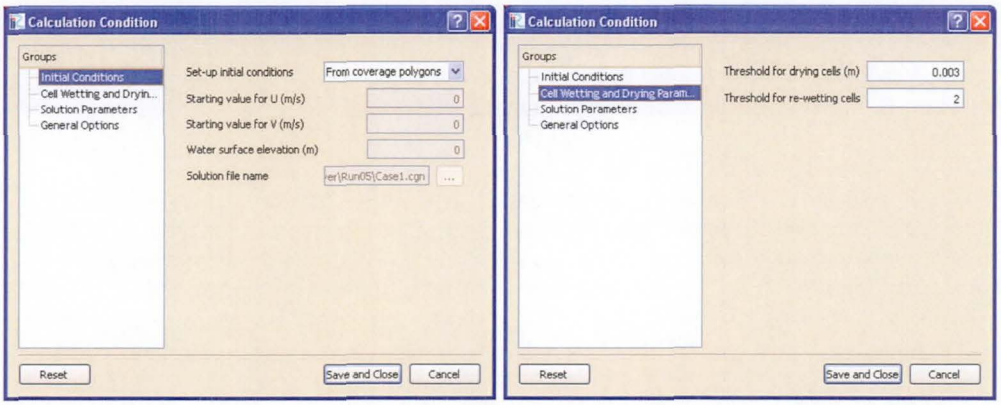


Figure 6.1

The **Wetting and Drying** entry screen shown in Fig. 6.1, right, is used to specify the parameters that control the behavior of the wetting and drying computation algorithm in SToRM. Parameter **Threshold for drying cells** is a value of the water depth (in meters) below which a wet cell becomes dry. In this example it is set to 3 mm. Parameter **Threshold for re-wetting cells** defines a value of the water depth above which a dry cell becomes wet. It is a dimensionless multiplier: this value multiplied by the **Threshold for drying cells** provides the desired value of the water depth. Here, the value of the parameter is 2, meaning that a dry cell is considered wet again when its water depth becomes higher than 6 mm (2 x 0.003). The **Threshold for re-wetting cells** parameter is important because it prevents excessive triggering of wetting and drying of cells at the river margins. This numerically-induced wetting and drying cycle can introduce instabilities in the solution and slow down convergence to steady state. Recommended values are of 2 or higher in the lowest range, and up to 10 or more, depending on the value used for the threshold for drying cells: low drying thresholds require higher re-wetting values.

The **Solution Parameters** entry screen (Fig. 6.2, left) is used to define the parameters used by the time stepping solution algorithms in SToRM. The **Time step size** defines the size of the time step used to advance the solution in time. The time units are seconds. The **Time stepping method** is a pull down menu that selects the time stepping method used in the calculations. There are two methods: **Order 1** and **Order 2**. The first method is computationally more efficient and the second more accurate (at the expense of computer run time). The choice of time stepping method is a compromise: in a typical application, both methods should arrive to the same solution, but it may be necessary to use smaller time steps when using the 1st order method. Experimentation may be needed to choose between each, with the ultimate goal being to reach a solution in the shortest computer run time.

A maximum of 100,000 time steps are sought (parameter **Number of time steps**), with intermediate print-out of the solution every 10,000 time steps (parameter **Plot Increment**). This corresponds to a run of 10,000 seconds (approximately 2.8 hours) with an intermediate solution every 1,000 seconds.

Finally, the **General Options** entry screen (Fig. 6.2, right) allows to select the type of roughness coefficient from the **Type of resistance coefficient** pull-down menu. The other options are not used in this tutorial and can be safely ignored for now.

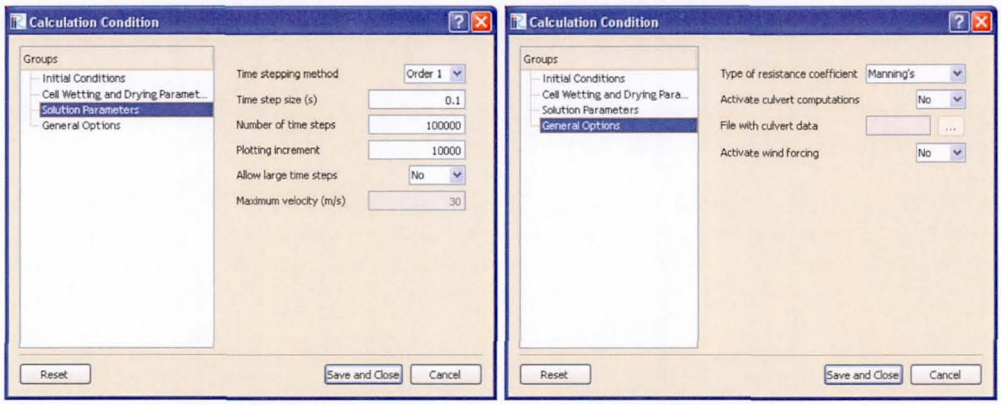


Figure 6.2

After the data preparation, SToRM is run by clicking the button on the main window command bar. A model run can be stopped at any time by clicking on the button. A window pops-up where the progress of the computation can be followed. The first column contains the iteration number; the second column contains the corresponding time, in seconds; the third column contains the value of a quantity that indicates the relative change of the water depth in each time step, defined as

$$\sqrt{\frac{\sum\left(h_i^{t+\Delta t}-h_i^t\right)^2}{\sum\left(h_i^t\right)^2}}$$

where h_i^t is the water depth in triangle i and at time t, Δt is the time step size, and the summation is carried over all the wet triangles; the fourth column contains the inflow discharge of water (m^3/s); and the fifth column contains the discharge out of the computational domain (m^3/s).

Choosing the appropriate time step for the run is an important task. SToRM uses explicit time stepping methods. In practical terms this means that, in order to keep the computations stable, there are constraints to the time step size. If unstable, the computations "blow up," which is a way to say that the variables become uncontrollably large or small — cause divisions by zero or the numbers become larger than the limits of computer representation — so that the water depth may become negative, and the run stops prematurely and its results are meaningless. An indication of these events is the presence of 'Inf' or 'NaN' in the model output window during the run. Usually, in convection-dominated flows, stability is related to flow celerity, size of the time step, and triangle dimension by a dimensionless parameter:

$$C_r = \frac{\Delta t\left(U_m + \sqrt{gh}\right)}{L_T}$$

where Δt is the time step size, U_m is the magnitude of the flow velocity at the center of the triangle ($= (U^2 + V^2)^{1/2}$), h is the water depth at the same point, g is the acceleration due to gravity (9.8 m/s^2), and L_T is a grid-related length scale (for example, the triangle's perimeter). The dimensionless parameter C_r is called the Courant number and is fixed for a certain time stepping method, but usually different for different methods. The above relation translates into the flowing statements:

1. For a triangle of a certain size, larger flow velocity requires smaller time steps.
2. In the same situation, larger depths also require smaller time steps, but this constraint is less severe than the one for flow velocity.
3. For a certain flow field, larger time steps can be used for larger triangles. For smaller triangles the time step size must be reduced accordingly.

This process is succinctly schematized in Fig. 6.3, where the two constraining parameters, flow velocity and grid size, are shown as curves representing the stability of the numerical solution. Considering each parameter individually, the curves represent the maximum stable time step size. For example, the maximum allowed time step size increases as triangle size increases, but it decreases as flow velocity grows.

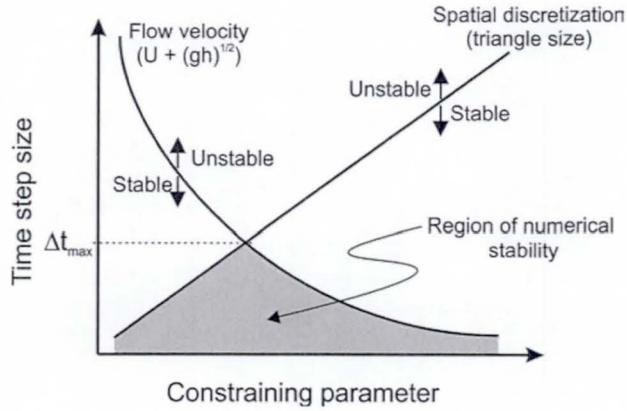


Figure 6.3

A stable time step must lie below both curves. In some cases the limiting factor is velocity magnitude, in others grid size. This means that, for each physical prototype, there is always a limit to the maximum value of the time step that produces a stable numerical solution, which is represented by Δt_{max} in Fig. 6.3. These are important facts to keep in mind when selecting the parameters that go into the **Solution Parameters** entry screen. Unfortunately, things are not that simple. For example, for shallow depths the friction terms become dominant, the flow is no longer convection-dominated, and new restrictions are imposed to the time step to keep the computations stable: small time steps are needed for very shallow depths. Therefore, a trial-and-error cycle is usually necessary to determine the largest time step that retains the stability of the computations. This is case dependent and varies with bed geometry and other boundary conditions.

7. Visualization of Results

Besides the data preparation and model's parameter input, a process usually referred to as pre-processing, iRIC also provides means to view the model's numerical solution. This post-processing stage is done in its independent window, titled **Post-processing (2D):1** and shown in Fig. 7.1. The pos-processing canvas of iRIC is invoked clicking on button 2D or by selecting **Calculation Result | Open new 2D Post-Processing Window** from the main menu. The **Post-processing (2D):1** window works identically to the **Pre-processing Window**, with its own **Object Browser**, and is used to plot the solution variables.

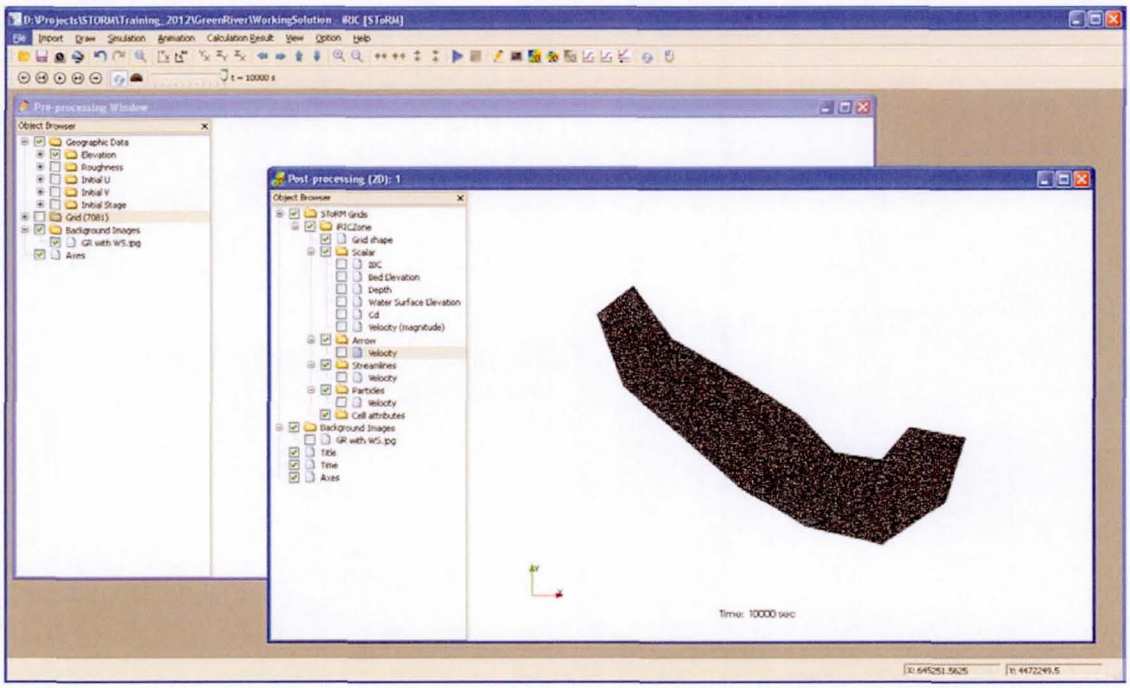


Figure 7.1

The **Post-processing (2D):1** can be resized, and visualization of the diverse solution variables is turned on and off by checking the corresponding boxes in the **Object Browser**. For example, velocity vectors and water depth are visualized in Fig. 7.2.

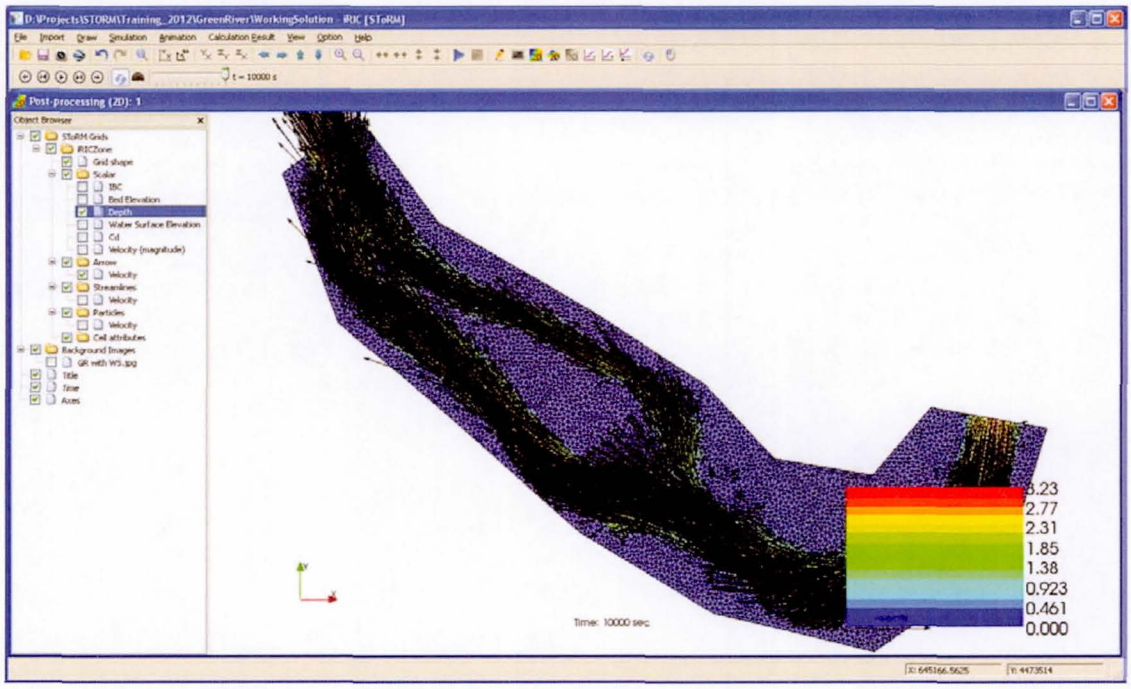


Figure 7.2

As it can be seen, sometimes the plotting parameters are not ideal for the optimum display of the desired quantities. The visualization of each can be fine tuned using the appropriate customization menus. For example, the visual properties of the velocity vectors can be changed from the **Arrow Setting** window (Fig 7.3 left), which is invoked by right clicking the mouse on **SToRM Gids | iRICZone | Arrow** in the **Object Browser**. Similarly, the visual properties for the selected variable, which is water depth in the present case, can be set in the **Scalar Setting** window (Fig. 7.3 right), which is invoked by right clicking the mouse on **SToRM Gids | iRICZone | Scalar** when **Water Depth** is checked. Visualizing all scalar quantities is done the same way and their names are self-explanatory. The user is encouraged to try them all.

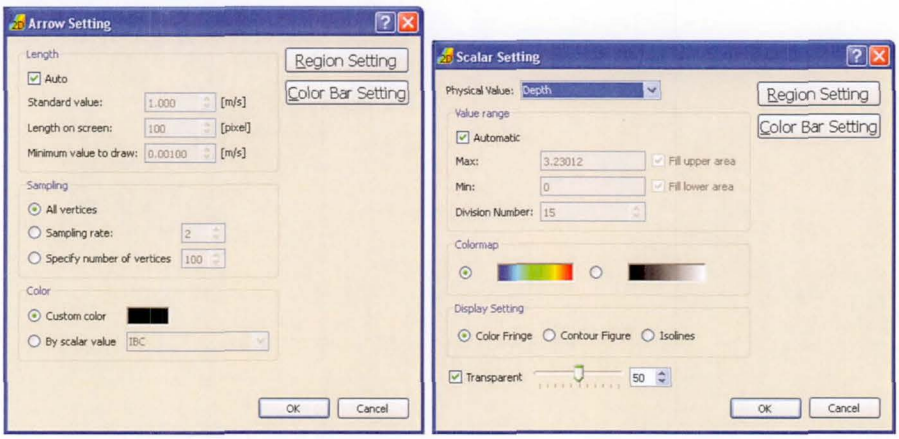


Figure 7.3

There is an important scalar quantity that is different and requires addressing. It is the **SToRM Gids | iRICZone | Scalar | IBC**. This quantity is used like a mask to separate wet from dry cells, and has a value of 1 in wet cells and 0 (zero) in dry cells. Its visualization is shown in Fig. 7.4. It is used to mask the other solution variables when it is desirable to see their values in wet cells only.

To show how to use **IBC** to mask a variable, let's use the water surface elevation. Without masking, the water surface elevation is shown in Fig. 7.5. The display shows water surface elevation in wet cells, but also in dry cells, where stage is not defined but plots with the same elevation as the bed. To restrict the display of water surface elevation to wet cells only:

- Right click the mouse on **Scalar** (henceforth the string **SToRM Gids | iRICZone** will be omitted when there is no source of ambiguity) and select **Property** to bring up the **Scalar Setting** window (Fig. 7.6, center).
- Click on **Region Setting** and select **Active Region** (Fig. 7.6, right). Click OK when done.
- Click OK in the **Scalar Setting** window to complete the process.

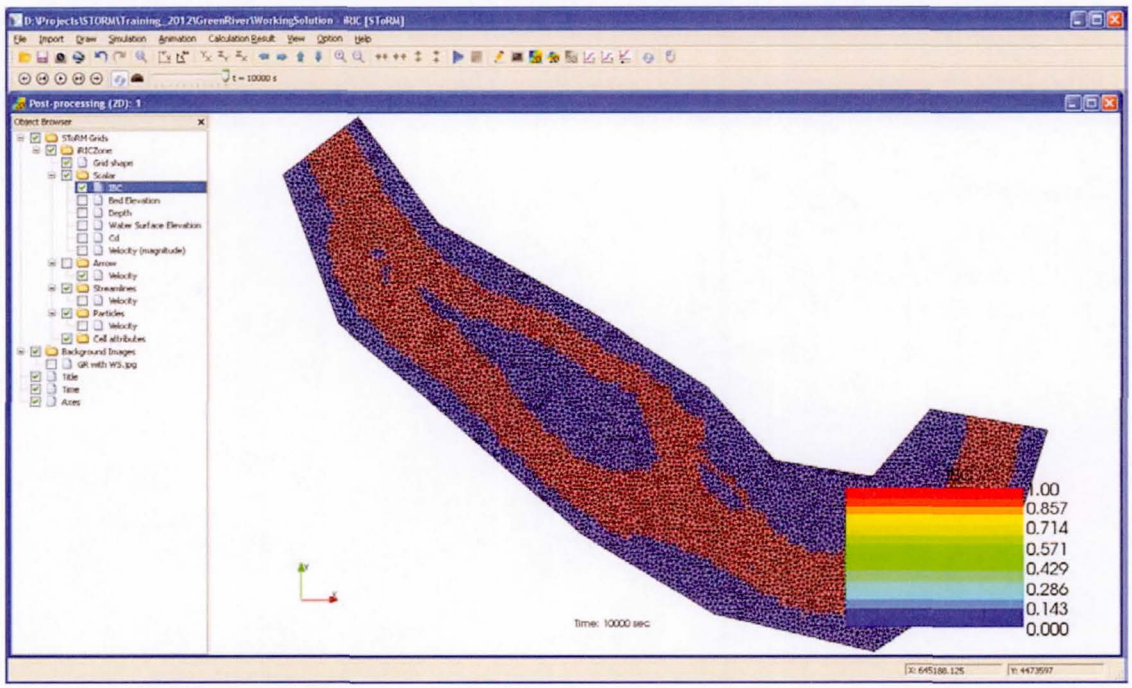


Figure 7.4

The resulting plot is shown in Fig. 7.7, where the legend was removed to improve clarity. To remove the legend:

- Right click the mouse on **Scalar** and select **Property** to bring up the **Scalar Setting** window (Fig. 7.6, left).
- Click on **Color Bar Setting** to bring up the **Color Legend Setting** window (Fig. 7.6, right).
- Uncheck the **Visible** setting. Press OK when done.
- Click OK in the **Scalar Setting** window to complete the process.

Note that there are other attributes that can be changed in the **Color Legend Setting** window. They are self-explanatory and the user is encouraged to try different settings to see how they impact the look of the plotted variables.

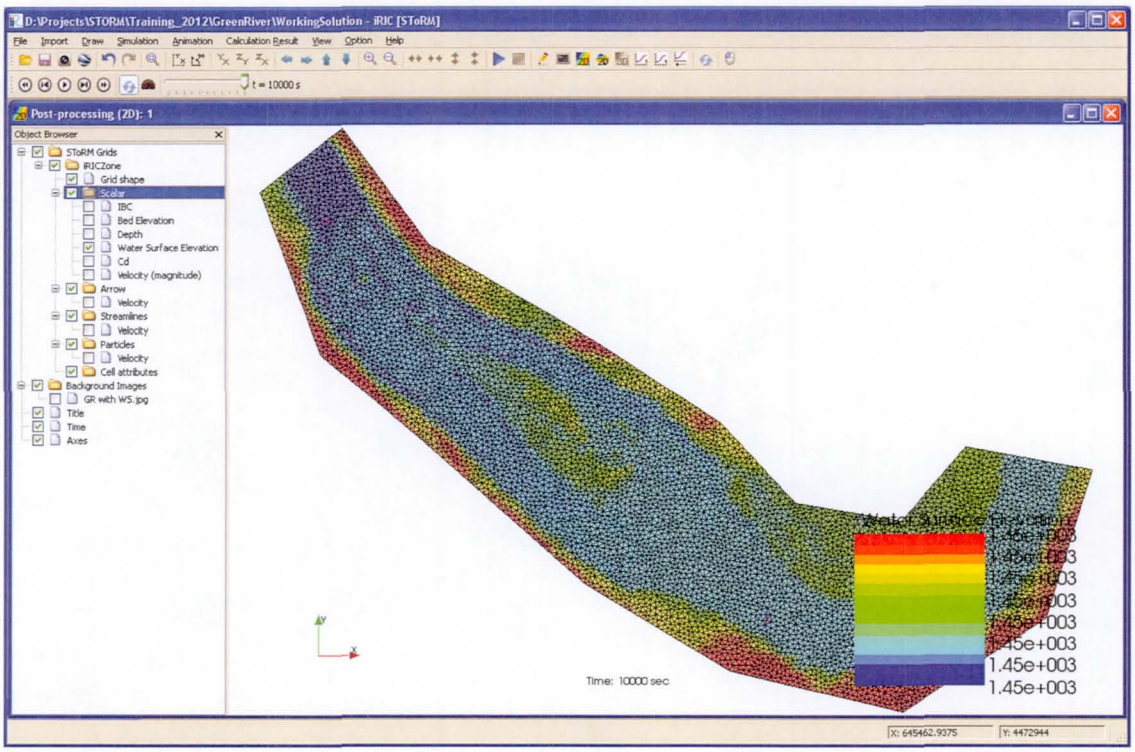


Figure 7.5

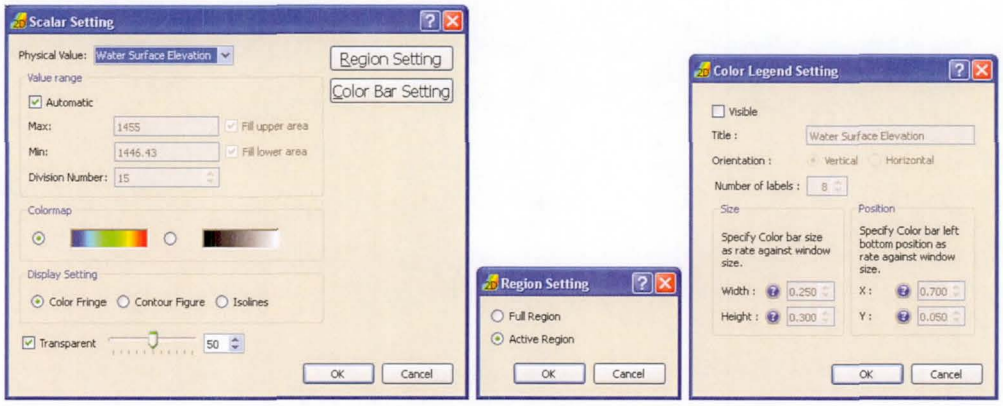


Figure 7.6

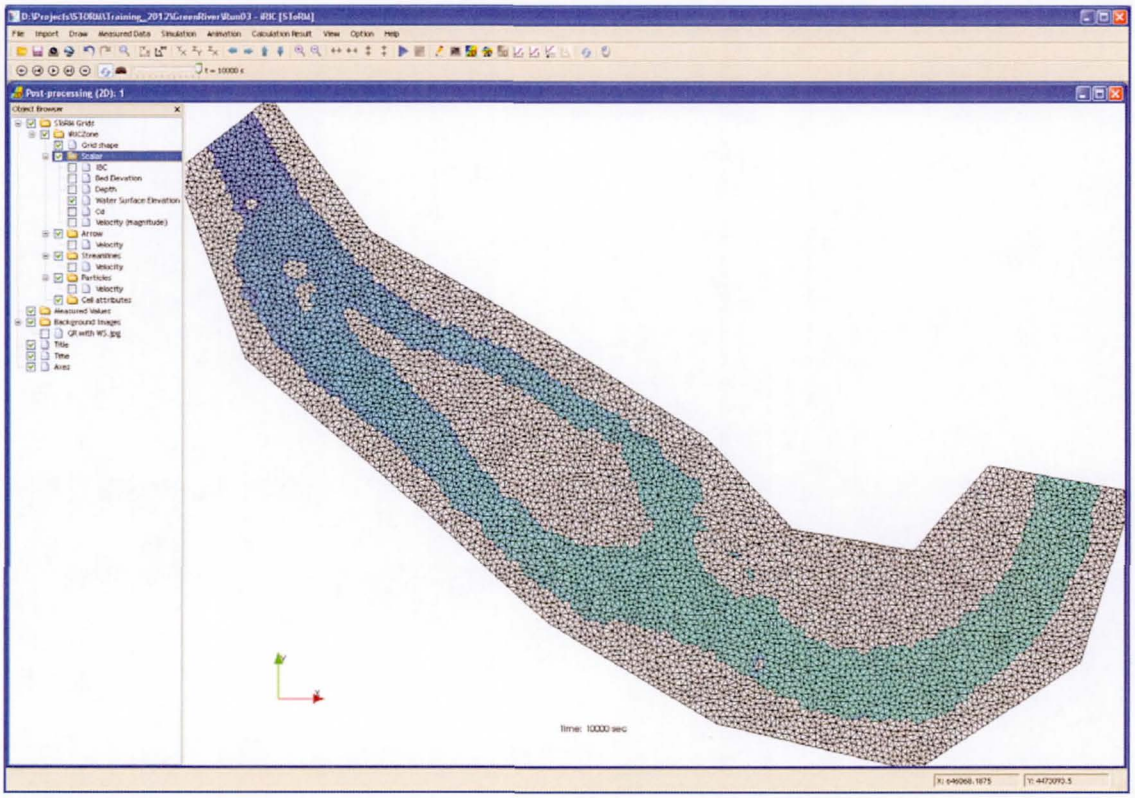


Figure 7.7

In iRIC, it is possible to visualize the flow using streamlines, which are lines are tangential to the flow field (they also represent the trajectories of weightless buoyant particles in steady state conditions). To create streamlines:

- In the **Object Browser** check **Streamlines** and **Velocity**.
- Right click the mouse on **Streamlines** and **Property** to bring up the **Streamline Setting** window (Fig. 7.8).
- Left click the mouse in a desired initial position in the plotting canvas and drag it to the desired final position without releasing the mouse button. Release the mouse button only at the final position. This operation defines a starting line for the streamlines. Set the streamline parameters (color, number of streamlines, and streamline width) as desired.
- Clock OK to create the streamlines.

A view of some streamlines is shown in Fig. 7.9, plotted over water depth.

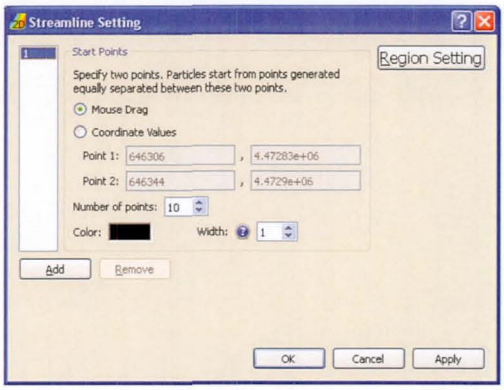


Figure 7.8

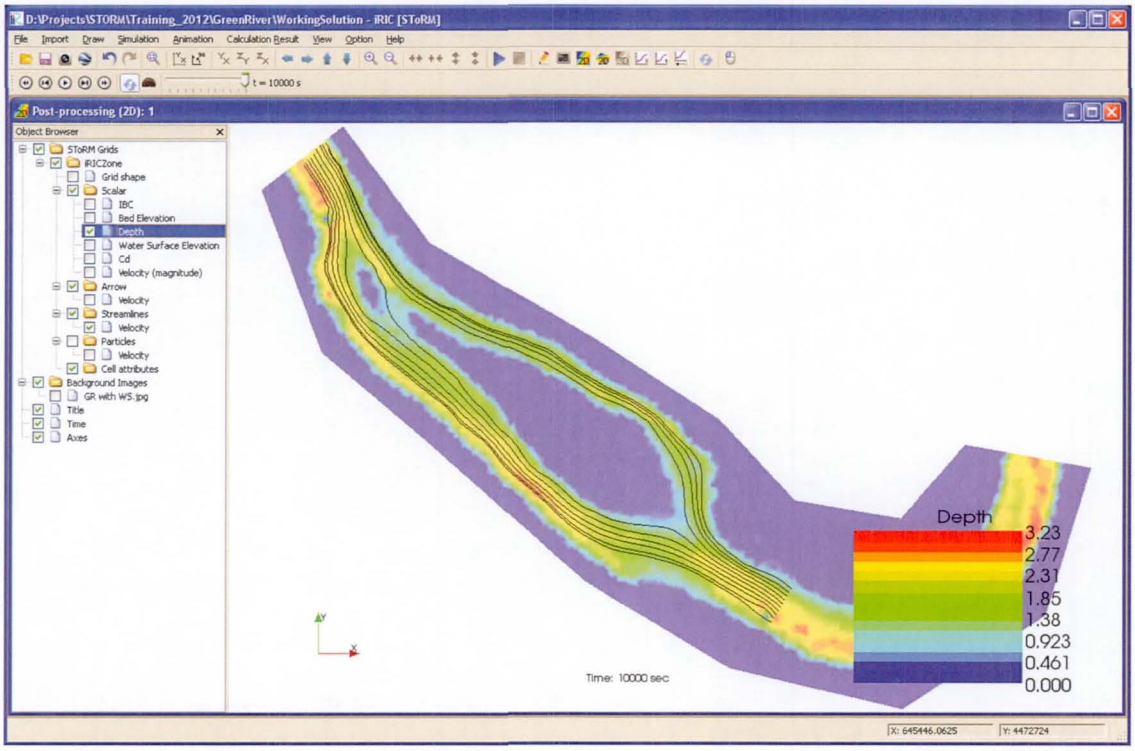


Figure 7.9

8. Refining the Solution

After a solution is found it may be desirable to refine it. There may be several reasons for this: the computational grid may have been deemed too coarse; a sensitivity analysis may be contractually required; the time step may have been too large; quality control; etc. One of the possible refinements may consist in using the 2nd order time marching scheme. This implies obtaining a new SToRM solution. To run SToRM with a 2nd order time marching scheme follow the next steps:

- Close the post-processing window.

- In the main menu, select **Calculation Condition** and navigate to the **Solution Parameters** entry screen.
- From the **Time stepping method** pull-down menu select **Order 2** (Fig. 8.1).
- Click on button **Save and Close**.

Because the same computational grid and boundary conditions and corresponding node strings are used, nothing else needs to be changed. SToRM can, therefore, be run again by simply clicking on button ▶ in the main toolbar. When the run finishes, follow the steps described in the previous section to view the new solution, which is shown in Fig. 8.2. Fig 8.2 shows velocity vectors over water surface elevation, with visualization of the computational grid turned off. Visualization of the solution vectors was customized to the display parameters shown in Fig. 8.3.

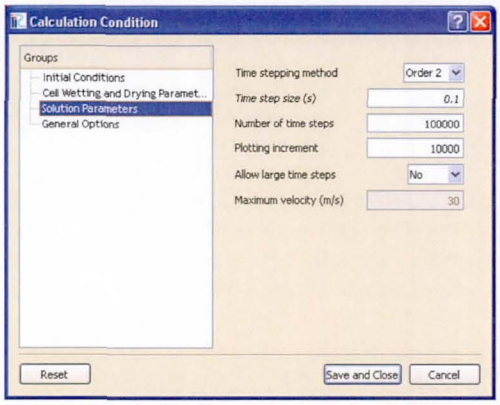


Figure 8.1

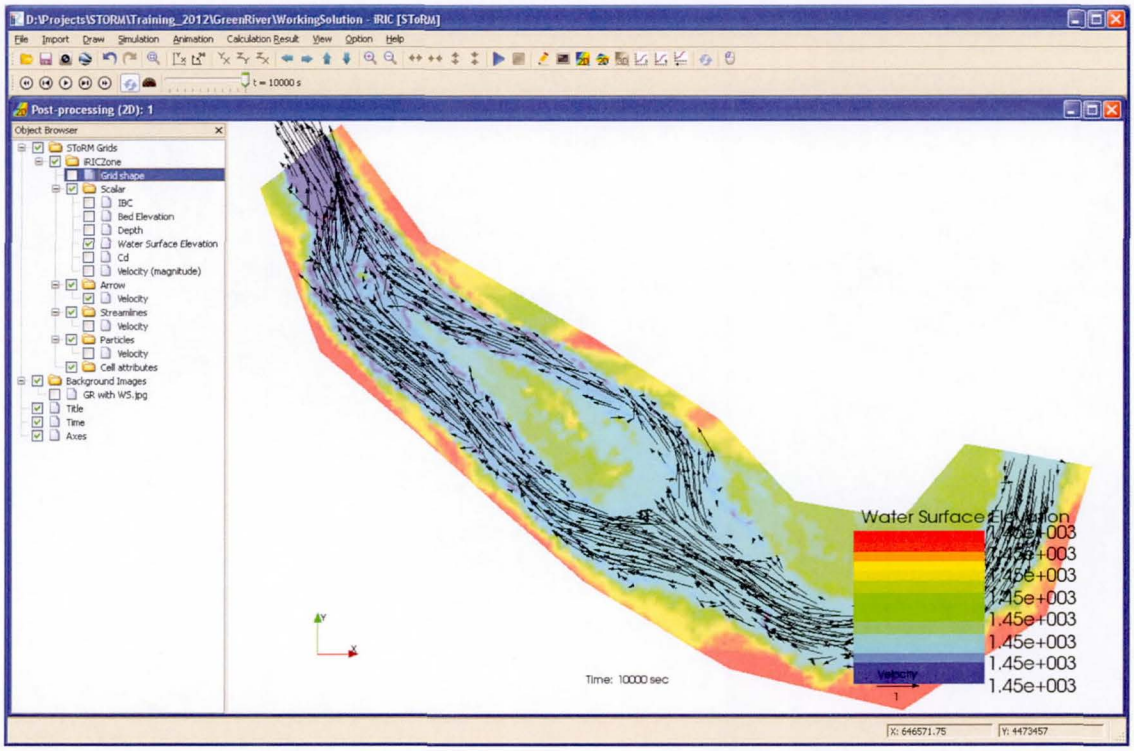


Figure 8.2

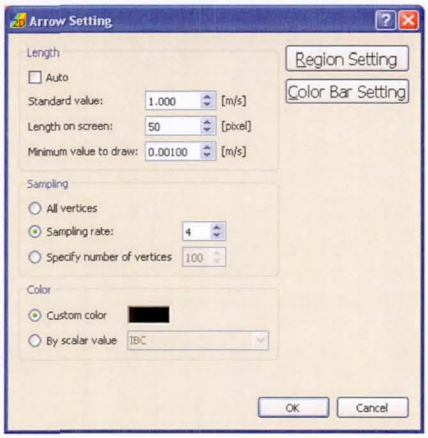


Figure 8.3

It is often necessary to refine (or coarsen) the mesh. Mesh refinement is desirable when more accurate solutions are sought; mesh coarsening is desirable when shorter computer run times are required. Care must be exercised when working on coarser meshes, to ensure that the quality of the solution is not degraded. When refining the mesh, the following steps must be followed:

- Create a new mesh using the mesh generator.
- Define new inflow and outflow boundary strings.
- Define new roughness and initial conditions polygons.
- Re-run the model. In general, the same SToRM parameters can be used, but the time step size may have to be adjusted. For example, the time step size may have to be reduced to ensure stability of the computations.

In essence, one must repeat all the steps previously described in sections 3 through 6. For this example, a new mesh was generated using the same polygonal shape defined in section 3, which needs not be re-entered. Only the mesh generator needs to be re-run, this time using a maximum triangle area of 40 m^2. The resulting mesh is shown in Fig. 8.4, with the new boundary node strings.

The new solution is plotted in Fig. 8.5 in the same manner as the one in Fig. 8.2. One can see an overall more refined solution, with a somewhat better behaved flow field—often a result of the smaller grid size at the margins of the river. The same SToRM parameters were used in this new run, but with a 1st order time stepping method and a time step of 0.1 s, as described previously in section 6.

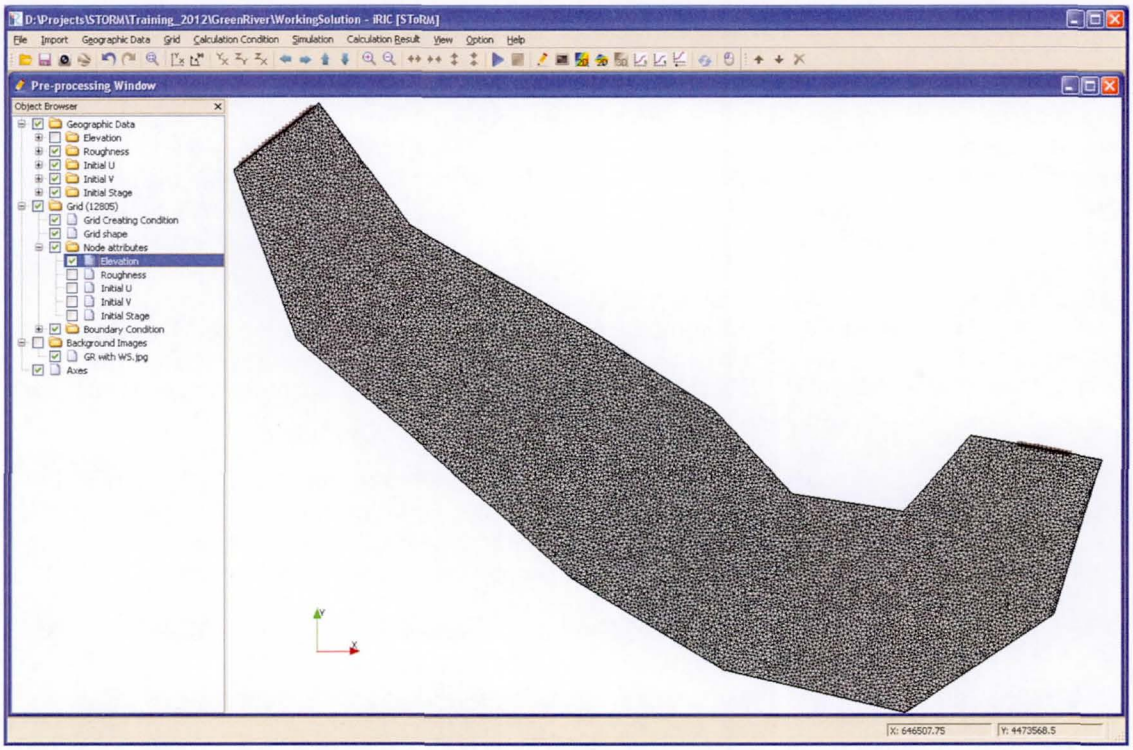


Figure 8.4

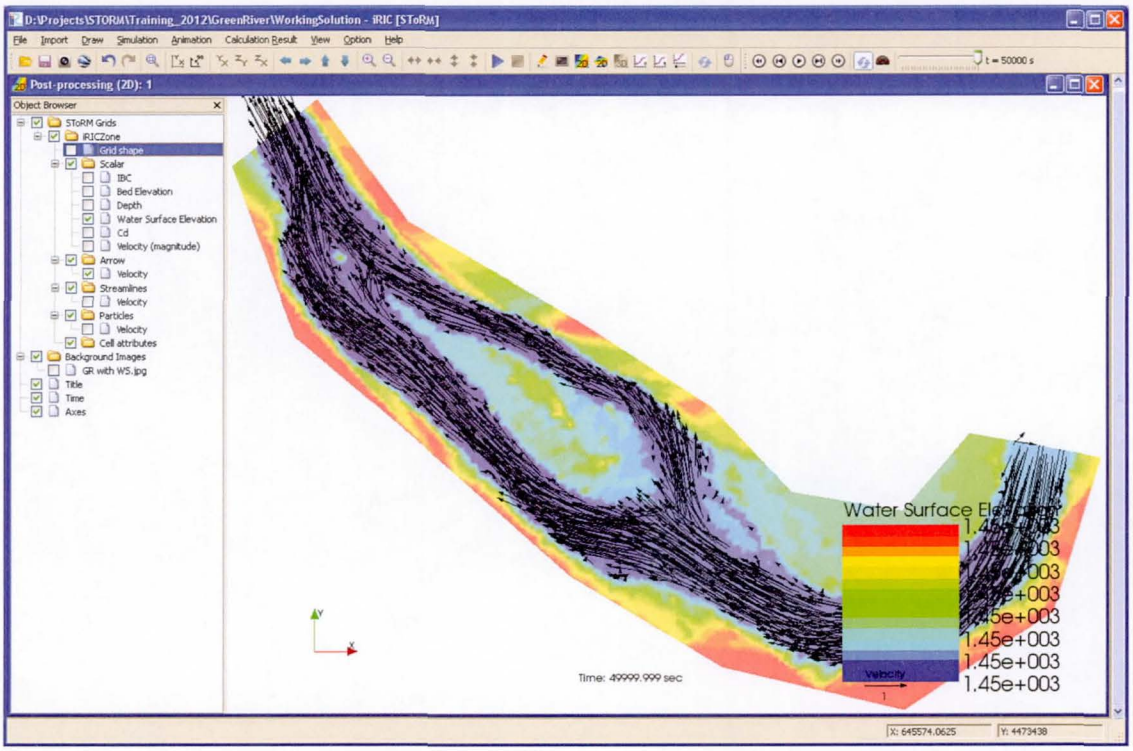


Figure 8.5

9. Thinning areas of dry nodes

Areas of the computational mesh that are dry are excluded from the computations. Nevertheless, they still use computer memory and contribute to an additional memory overhead that may degrade computational efficiency, resulting in longer run times of SToRM. An example of a dry, unused part of the computational grids discussed in this tutorial is the island in the middle of the river. iRIC and its mesh generator provide the tools necessary to thin out that area, a process that involves delineating the area to be thinned (or hollowed out) and regenerating the mesh.

To thin out the island, start by turning all the visualization options off except the background figure, the **Grid Creating Condition**, and the **Node Attributes** (set to **Elevation**). This will result in a display canvas such as the one shown in Fig. 9.1 and will help guide the process. Select **Grid Creating Condition** in the **Object Browser** window. This will activate the editing menu bar . Left click the mouse on the button and draw a polygon around the area to thin out. The polygon is drawn by left-clicking the mouse on the desired points and terminating with a double click or carriage return. Here, an hexagon is used to illustrate the process (Fig. 9.2). Right click the mouse on Grid to generate the final mesh. The resulting grid is shown in Fig. 9.3, where the **Grid Shape** visualization was turned on.

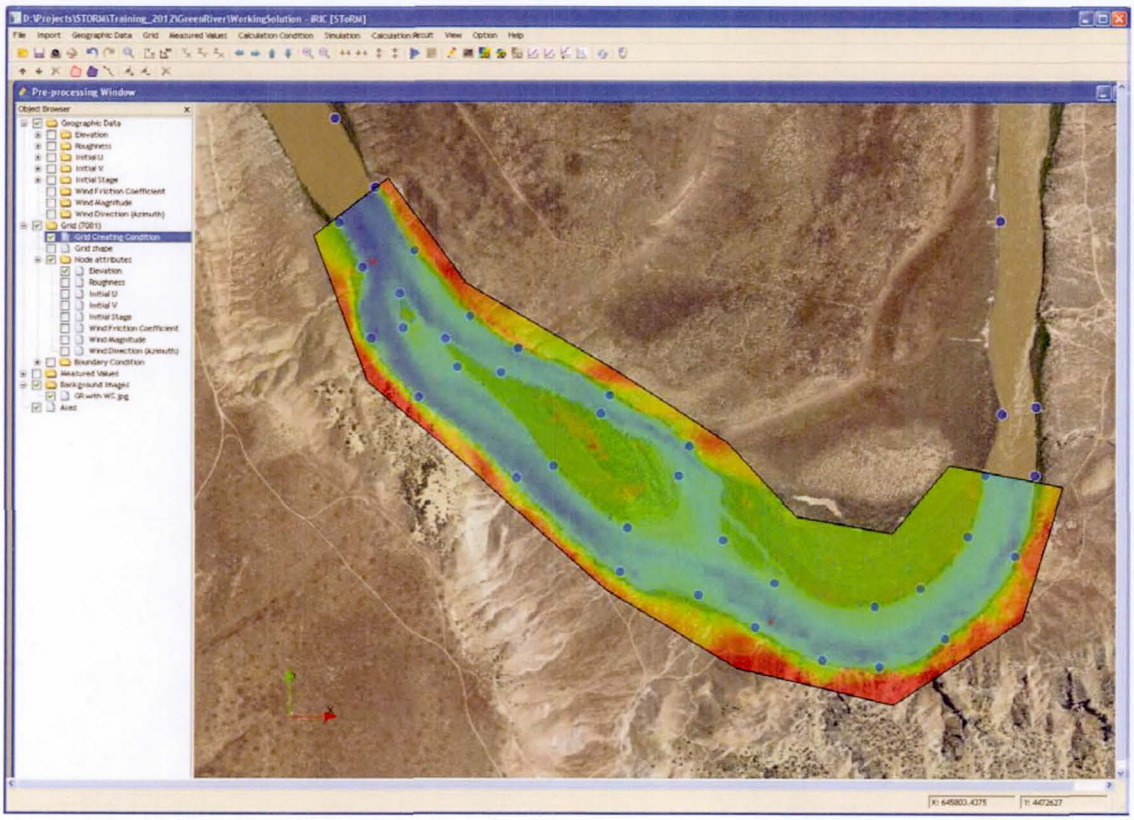


Figure 9.1

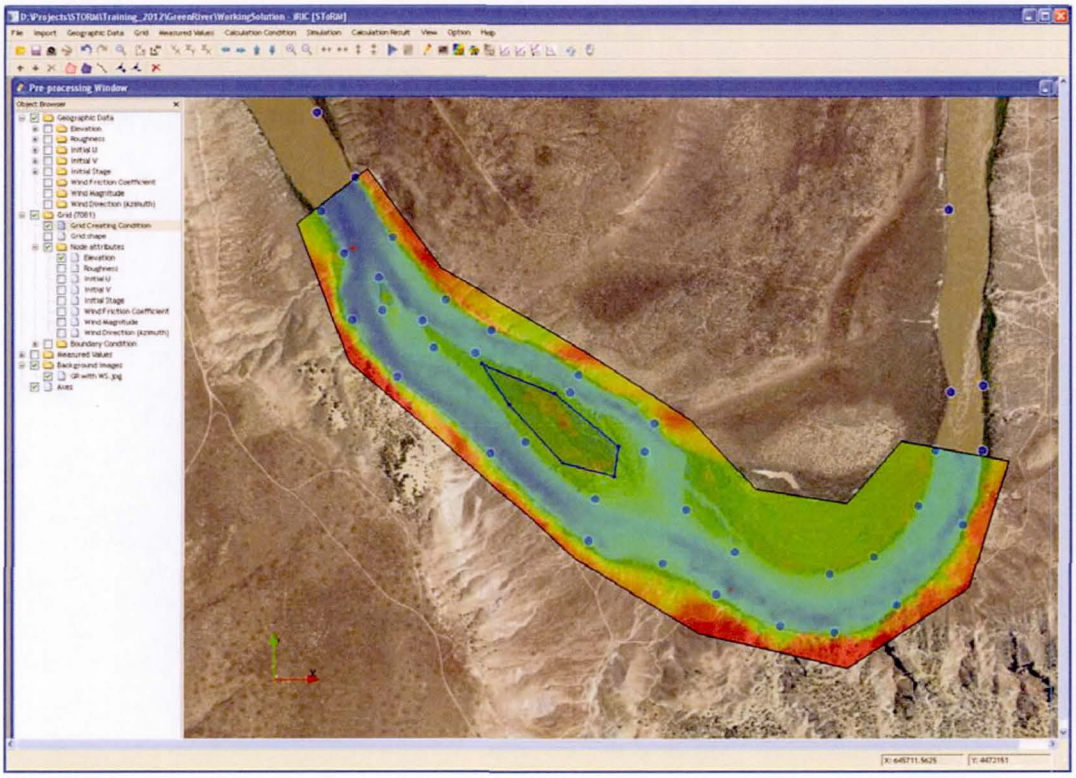


Figure 9.2

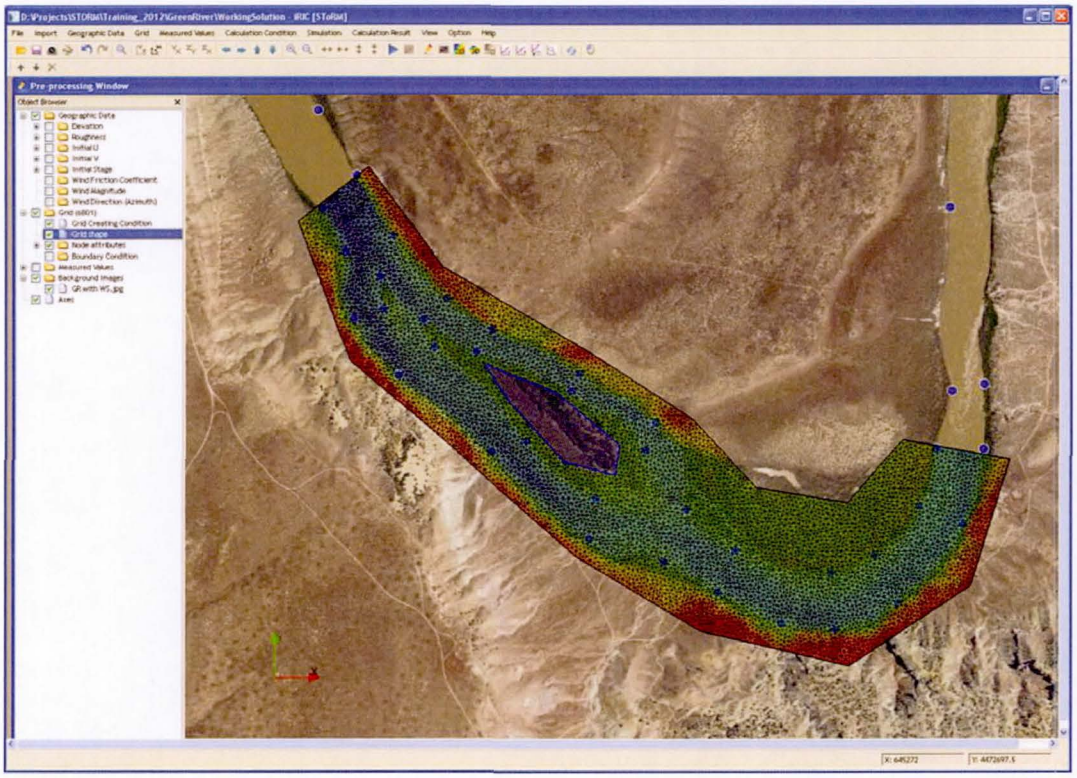


Figure 9.3

It is possible to create as many thinned holes in the mesh as desired. The boundaries of each of these areas are treated as vertical solid walls and the flow will not go through. Vertical bridge piers, dikes, and general emergent obstacles in the flow can, therefore, be represented by thinned out holes.

Please recall that the old inflow and outflow boundaries must be redefined anew every time a new computational grid is created.

10. Model Calibration

In spite of the many tables commonly found in open channel flow textbooks, characterization of hydraulic roughness is still a difficult task to accomplish. One of the difficulties stems from the high variability of surface types encountered in environmental hydraulics: sand, coble, and boulder river beds, vegetated channels, debris and other flow obstructions, all offer different values of resistance to the flow. It is not easy to measure flow resistance itself directly in the field. In practice, a surrogate to flow resistance is provided by the slope of the free surface elevation, which is a quantity easy to measure. In this section, values of the free surface elevation are used to obtain an estimate of the value of bed roughness (Manning's *n* values) for the reach.

iRIC provides a tool to import measured data and to compare it with the corresponding solution quantities computed by SToRM. Here, the experimental measurements are provided in a plain ASCII text file named "Calibration WSE.txt". The first few lines of this file are:

```
X,Y,WaterSurfaceElevation
645360.541958,4473695.035004,1447.375977
645411.952969,4473599.122009,1447.375977
645499.504971,4473469.084000,1447.485962
645430.440029,4473445.841003,1447.524048
645616.889004,4473384.176003,1447.650024
...
```

The first line contains a title, and the following lines contain triplets of data: for each data point, the x-coordinate, the y-coordinate, and the value of the free surface elevation, all using the same datum as the original topographic data and separated by commas. The tool that allows the use of this data in iRIC is accessed through the button in the main toolbar.

To compare measured data with an existing SToRM solution in iRIC, follow the next steps:

- Import the data to iRIC. This can be done from the **Import | Measured Values...** pull-down menu in main menu bar; it can be done from the **Measured Values | Import...** pull-down menu also in the main menu bar; or it can be done by right-clicking on **Measured Values** in the **Object Browser** panel of the **Pre-processing Window**.
- In the **Open Measured Data File** window that pops up, select file "Calibration WSE.txt" and click the **Open** button.
- A data object is created in the **Object Browser** window that allows turning on and off the visualization of the data (Fig. 10.1).
- Click on button in the main toolbar. The **Verification Window** pops up (Fig. 10.2).
- Select the appropriate solution variable from the **Physical Value** pull-down menu. In this case it is the Water Surface Elevation, as shown in Fig. 10.2. Don't forget to drag the **Select the timestep of calculation result** pointer to the desired solution. Click **Ok** to close the window.

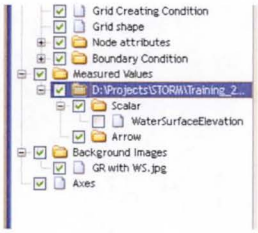


Figure 10.1

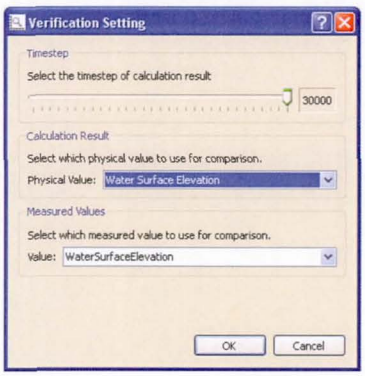


Figure 10.2

- The result is plotted in a calculation vs. measurement graph, as shown in Fig. 10.3. The line of perfect agreement is also plotted.

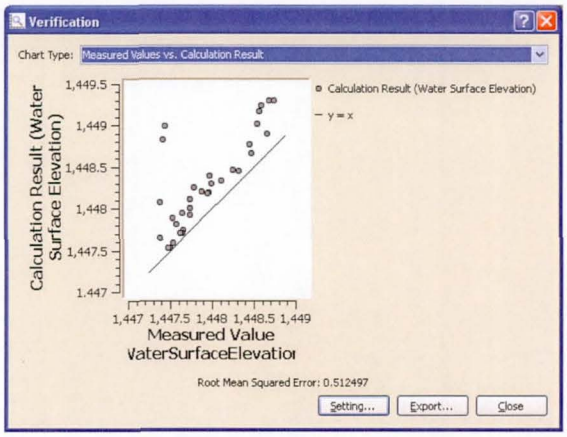


Figure 10.3

The results shown in Fig. 10.3. were computed with a uniform Manning's *n*-value of 0.035, as described in section5. The calibration procedure consists in varying the roughness values in order to obtain the best fit. This is a cyclic procedure, where the roughness is adjusted, SToRM is run, then the predicted water surface elevation is compared to the corresponding measured values. These steps are carried out until the best solution is obtained. Results of this process are shown in Fig. 10.4 for values on the manning's *n* of 0.015 (top), 0.025 (middle), and 0.045 (bottom).

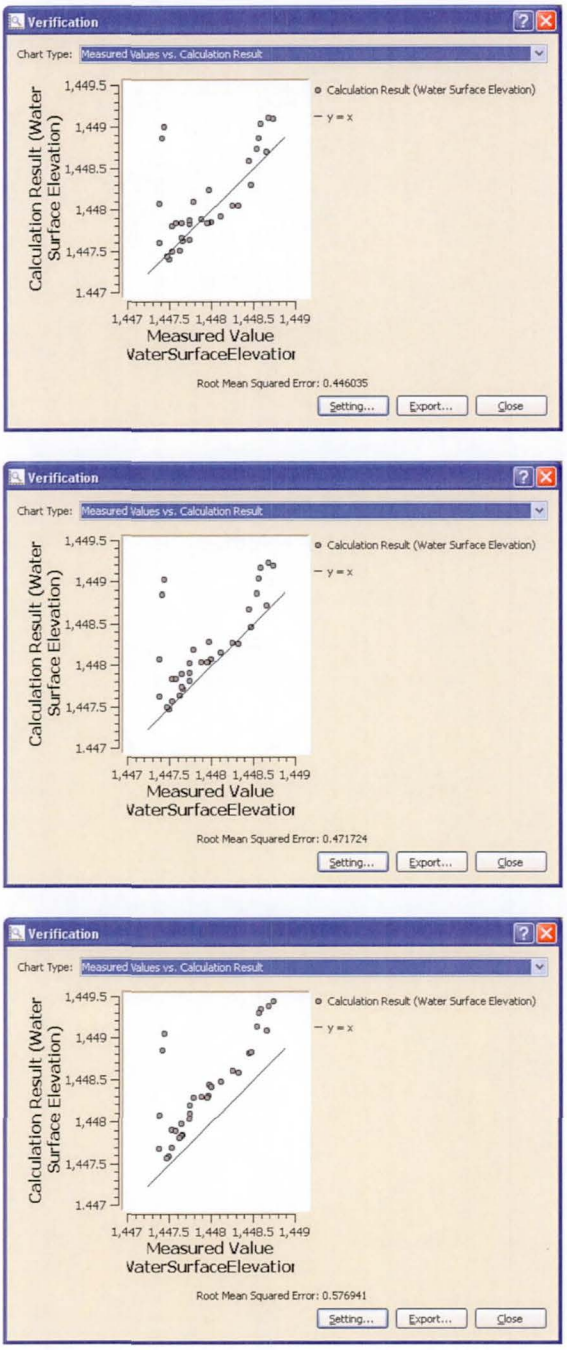


Figure 10.4

In this particular case, some of the solution shows good agreement with the measured data for *n* = 0.025, but other does not. This suggests that a single value of the roughness may not be enough to describe the reach properties, and that multiple roughness polygons are required in different regions.

TUTORIAL 4: 30-Mile Canal, South Platte, NE

SToRm is a time-dependent flow model that was specifically developed to calculate rapidly varying surface water flows with unsteady discharges. The previous tutorials showed how to set-up SToRM using the iRIC[1] 2.3 graphical user interface (henceforth called iRIC only, for simplicity). They cover all the main steps necessary for a successful SToRM simulation, such as using bathymetric data and generating a computational grid, defining initial and boundary conditions, and finding a suitable time step, but steady state conditions were used to simplify the presentation. The present tutorial complements previous information by using fully unsteady data. The bathymetry and boundary conditions used herein are real field data that were collected in 30-Mile Canal—a bypass canal in the South Platte River, NE—and represent a spill from a retention dam through a small canal, with flooding and drying of adjacent floodplains.

Contents:

1 .....Preparing Bathymetric Data and Computational Grid
2 .....Defining Inflow and Outflow Boundaries
3 .....Roughness
4 .....Running the Model
5 .....Visualization of Results
6 .....Final Remarks

1. Preparing Bathymetric Data and Computational Grid

The bathymetric data of the reach used in this tutorial is contained in file GPS_survey_9_16.10.tpo. To import the data into iRIC:

- Use command **Import | Geographic Data | Elevation** from the main menu bar and select file GPS_survey_9_16.10.tpo. Recall that the file has extension .tpo, which must be set appropriately in the filter of the **Select file to import** window.

The result of the **Import** command is shown in Fig.1.1.

For the present exercise, a computational mesh having a maximum element area of 2 m^2 is used. This size grid is chosen to capture enough topographic detail for the model simulation, but without resulting in very long computation time. In practice, grid resolution must reflect the accuracy sought for the flow field, but it must also be consistent with the data available. Here, the grid used is of finer resolution than the available topographic information, therefore care must be exercised when interpreting the results.

[1] iRIC stands for International River Interface Cooperative.

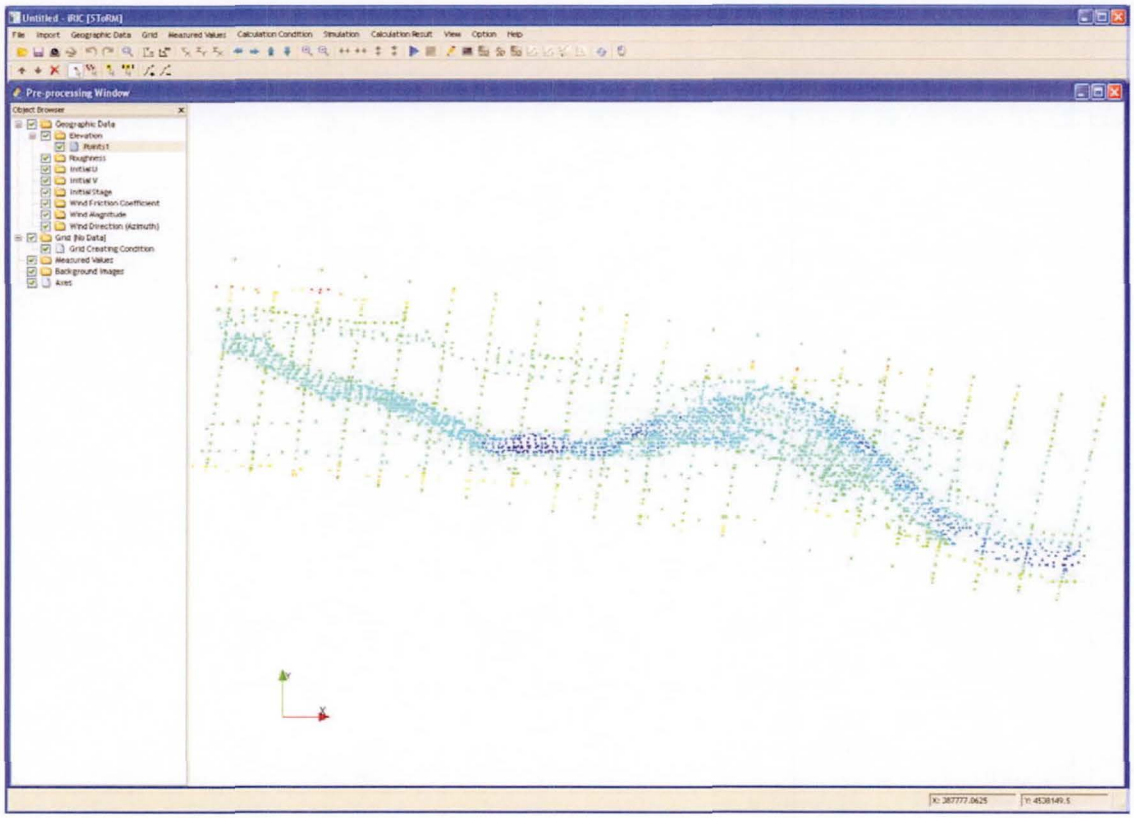


Figure 1.1

The steps to generate the computational grid are identical to those described with detail in previous tutorials:

- Use command **Grid | Select Algorithm to Create Grid...** to select **Create grid from polygonal shape**. Click **OK**.
- The computational domain is defined by a polygon that encompasses the area of interest. Define it by left-clicking the mouse at the desired polygon vertices. Double-click the last point or press <Enter> to finish the polygon. In this example we are interested in the region in close proximity to the confluence of the two rivers. Use Fig. 1.3 to guide you to create the mesh polygon.
- Right-click on **Grid [No Data]** in the **Pre-processing Window** and select **Create grid...** in the pop-up menu. Use the parameters shown in Fig. 1.2 (minimum angle for cell vertices: 32 degrees; maximum area for cells: 2 m^2). Click <OK>.

The resulting mesh is depicted in Fig. 1.3. Further data visualization follows the same steps as described in previous tutorials.

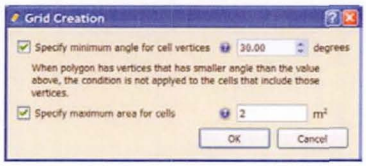


Figure 1.2

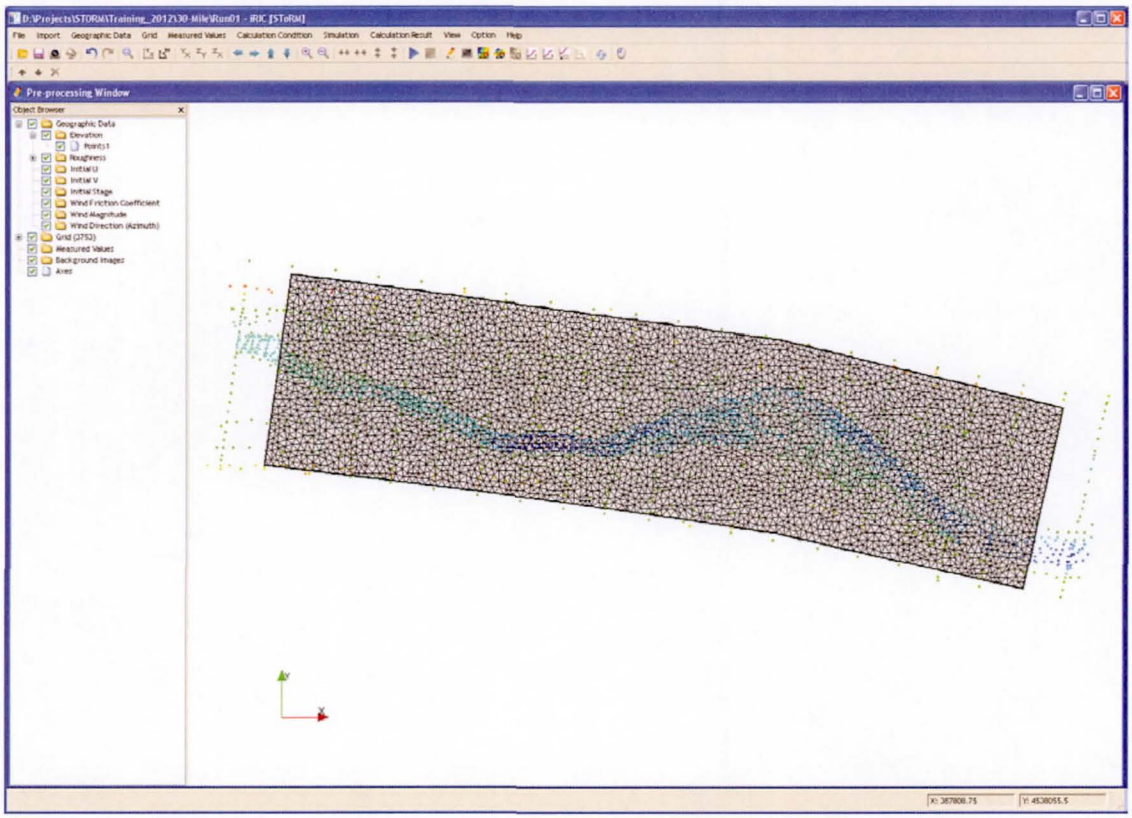


Figure 1.3

2. Defining Inflow and Outflow Boundaries

The unsteady boundary conditions used in this section were derived from flow measurements *in loco* during a release flow event at the 30-Mile Canal. An inflow discharge hydrograph is used at the upstream-most end of the grid, which, in this case, is the western boundary. A corresponding stage hydrograph is used at the downstream-most end (the eastern boundary).

To define the inflow boundary:

- Zoom in the area of interest in the mesh boundary.
- Right-click on **Boundary Condition Setting** item in the **Pre-processing Window** and select **Add Inflow** in the pop-up window.
- Create a polygon surrounding the nodes belonging to the boundary. Only external nodes (nodes on the boundary of the mesh) should be included in this polygon. The polygon creation process is ended by double-clicking the last point or by simply pressing the <Return> key. The nodes belonging to the inflow boundary are shown in Fig. 2.1.
- Assign the inflow condition a name and click **Ok**. Here, we use **Inflow** (Fig. 2.2, left).
- Enter the time series for the inflow discharge. Each row contains a time (left column) and a discharge (right column), and is created by clicking on button **Add**. Here, the hydrograph is defined by 5 points, therefore 5 rows need to be created. The cell values are entered by clicking on the cell and typing the desired value. Enter the values shown in Fig. 2.2 (right). The shape of the hydrograph is plotted at the right of the same window. Click **Ok** to terminate the input of values.

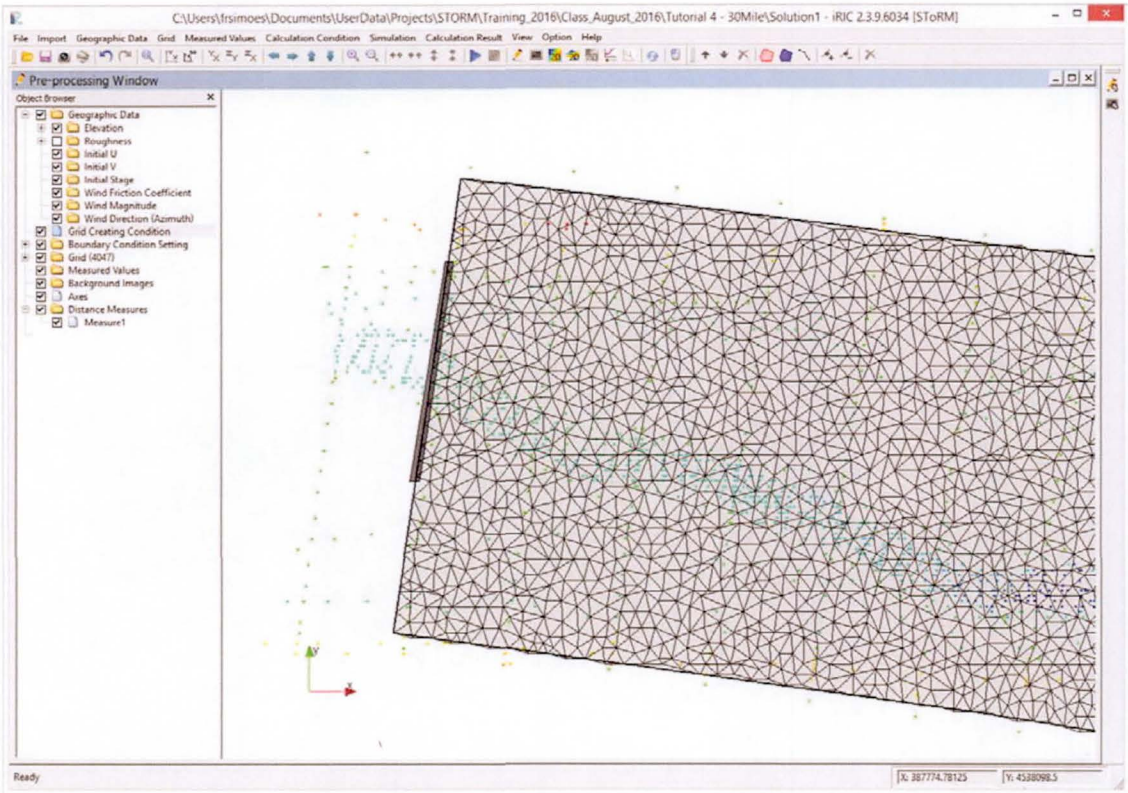


Figure 2.1

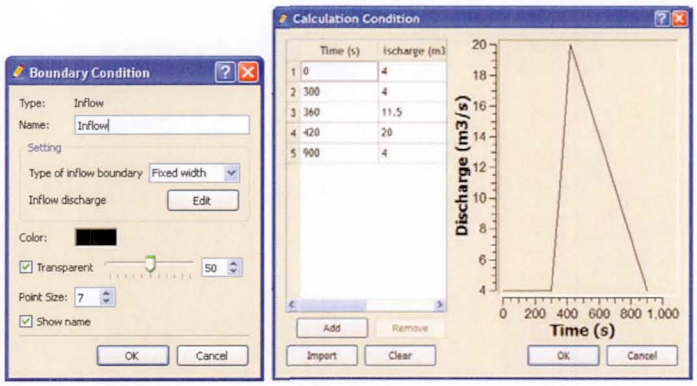


Figure 2.2

Note that the first entry of the inflow hydrograph is set at time 0 (zero). All SToRM simulations start at time 0 and proceed forward, incrementing time one time step at a time. It is not necessary, however, to start the hydrograph at time 0: SToRM uses the first value of the hydrograph if its first entry is set at a time larger than zero, until the defined time is reached. Linear interpolation is used for times between the points defining the hydrograph. For example, if the hydrograph in Fig. 2.2 started at time of 300 seconds, SToRM would used a value of Q = 4 m^3/s until such time was reached. The same approach is used for times after the end of the input hydrograph: for the hydrograph of Fig. 2.2, SToRM will use an inflow of 4 m^3/s if the run is prolonged after the 900 second limit of the hydrograph. The base discharge in the flow channel before the flow release is 4 m^3/s and, in the present exercise, the first 300 seconds are used to let the flow establish itself in the channel from the guessed initial conditions.

The downstream boundary is located at the eastern side of the computational grid. A stage hydrograph matching the inflow discharge must be defined there. Entry of those values is done in the same way as for the inflow hydrograph:

- Zoom in the area of interest in the mesh boundary.
- Right-click on **Boundary Condition Setting** item in the **Pre-processing Window** and select **Add Outflow** in the pop-up window.
- Create a polygon surrounding the nodes belonging to the boundary. Only external nodes (nodes on the boundary of the mesh) should be included in this polygon. End polygon entry by double-clicking on its last vertex or by simply pressing the <Return> key. The resulting polygon is shown in Fig. 2.4.
- Assign the inflow condition a name and click **Ok**. Here, we use **Outflow** (Fig. 2.3, left).
- Enter the time series for the outflow stage, each row contains a time (left column) and a stage (right column), and is created by clicking on button **Add**. Here, the hydrograph is defined by 5 points, therefore 5 rows need to be created. The cell values are entered by clicking on the cell and typing the desired value. Enter the values shown in Fig. 2.3 (right). The shape of the hydrograph is plotted at the right of the same window. Click **Ok** to terminate the input of values.

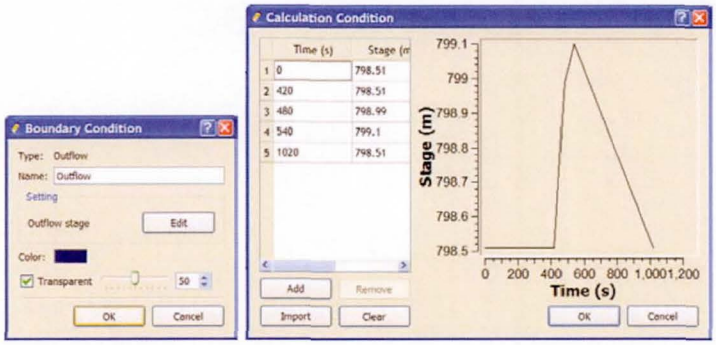


Figure 2.3

The inflow and outflow hydrographs are correlated to each other physically, but they are entered using separate time values. In other words, the values entered in the **Time** columns of the corresponding **Calculation Condition** windows (in the right side of Figs. 2.2 and 2.3) do not need to be the same. And, just like for the inflow hydrograph, the first point of the outflow hydrograph does not need to be set at time 0 (zero). Also, if the run is prolonged for more than 1020 seconds, SToRM will set the outflow stage at 798.51 m.

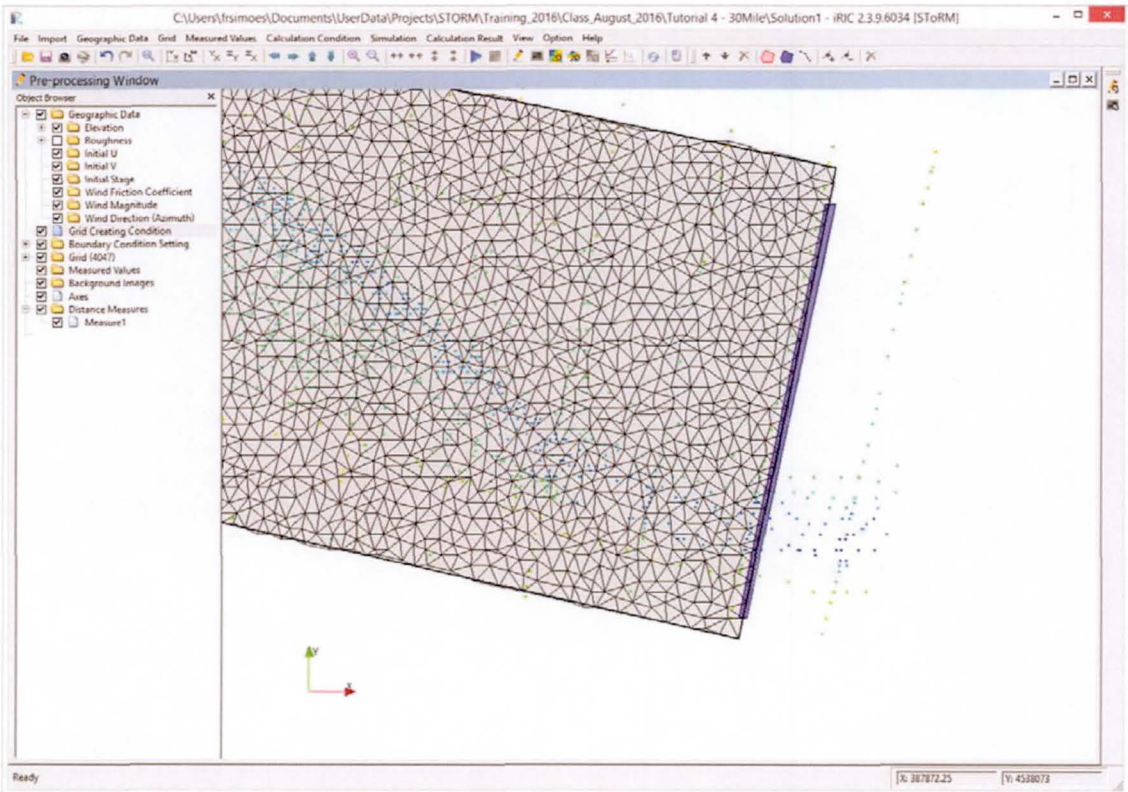


Figure 2.4

3. Roughness

In this example, channel bed roughness is set to a Manning's *n* value of 0.025 using a coverage polygon, as shown in Figs. 3.1 and 3.2. The initial conditions are set in the next section.

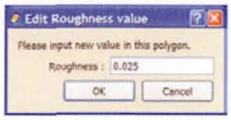


Figure 3.1

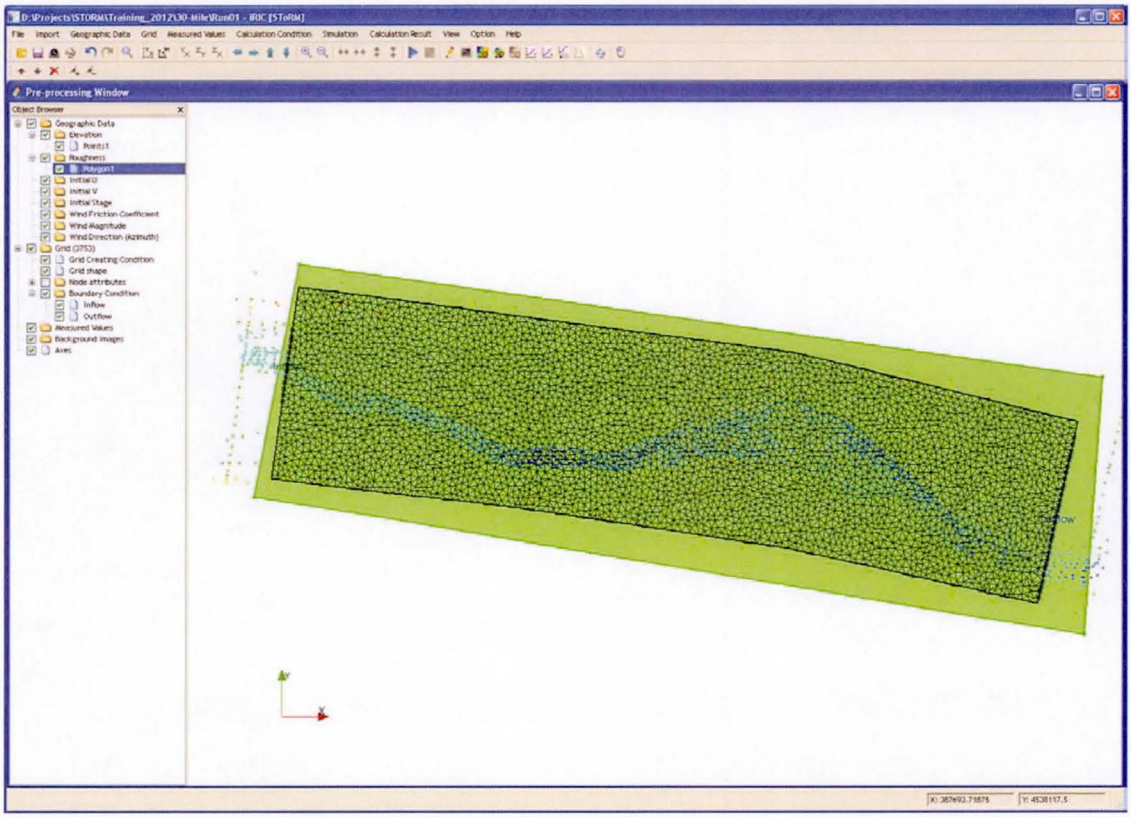


Figure 3.2

4. Running the Model

Unlike in the previous tutorials, initial conditions have not been set yet. This is because iRIC allows the initial conditions to be defined in multiple ways. In this exercise, uniform initial conditions are used and, rather than using coverage polygons, a different option is used. To accomplish this, go to **Calculation Condition | Setting...** to set-up model parameters. The initial conditions are set in the first entry screen. Here, uniform initial conditions are set to U = 0.4 m/s, V = 0 m/s, and water surface elevation to 798.71 m (see Fig. 4.1, left).

- Use the pull-down menu **Set-up initial conditions** and select **Uniform initial conds**.
- Enter the values shown in Fig. 4.1, left.

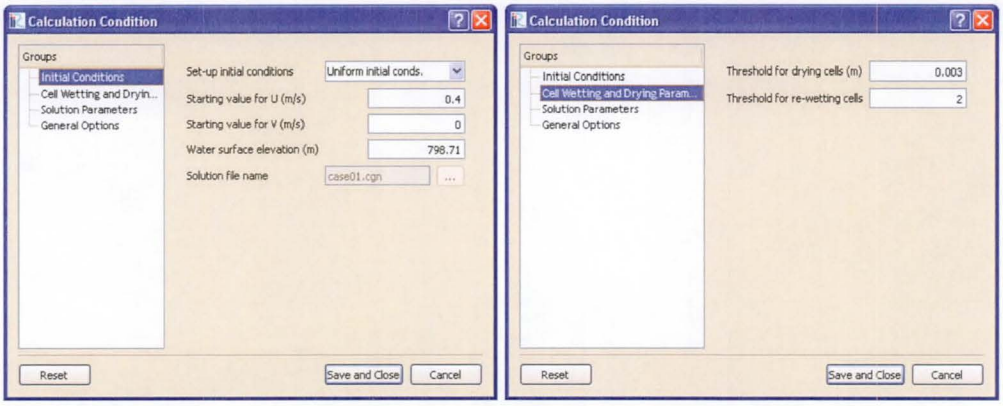


Figure 4.1

Set wetting and drying parameters as shown in Fig. 4.2, right. In this simulation, a 1st-order numerical integration scheme is used, with a time step of 0.005 seconds. The flow event will be simulated in 400,000 time steps. Plotting interval is set to 3 minutes. The above parameters are shown in Fig. 4.2, left. Make sure the roughness coefficient used is set to Manning's *n* (Fig. 4.2, right). Now, run the model using button in the main iRIC toolbar.

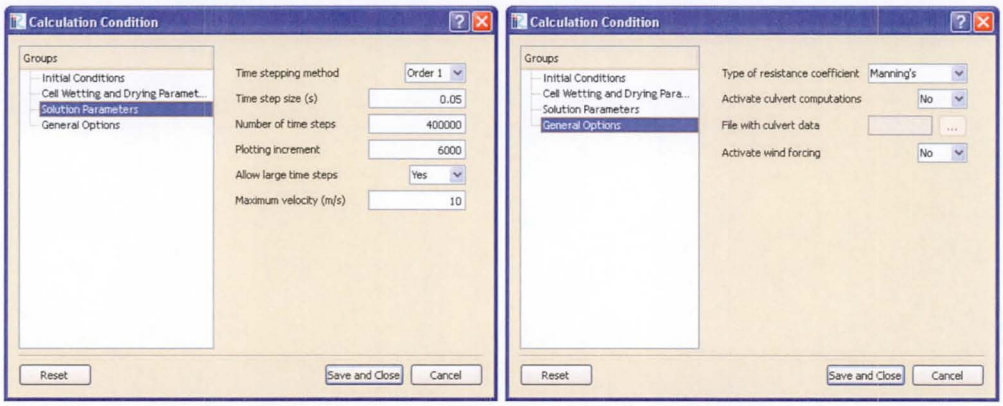


Figure 4.2

5. Visualization of Results

The run takes approximately 28 minutes in a normal desktop PC. Use the button to bring up the visualization window. The results are shown in Figs. 5.1 through 5.3 for the velocity vectors (color denotes velocity magnitude), water depth (under the computational grid), and water surface elevation, respectively.

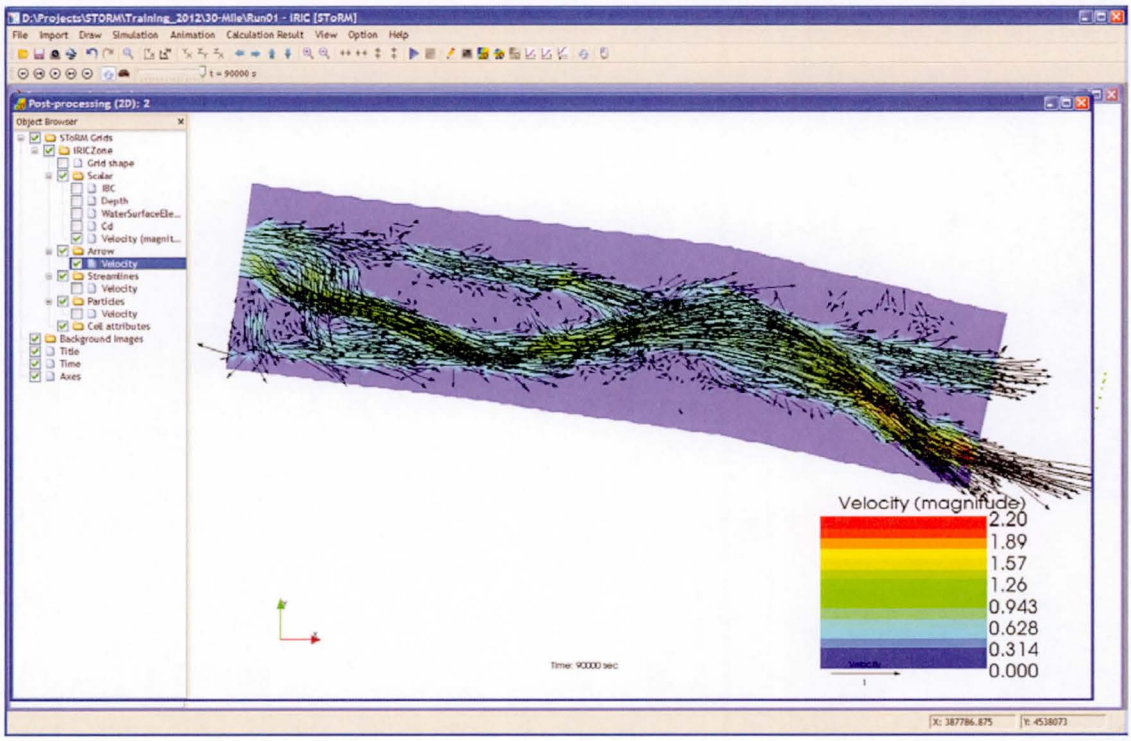


Figure 5.1

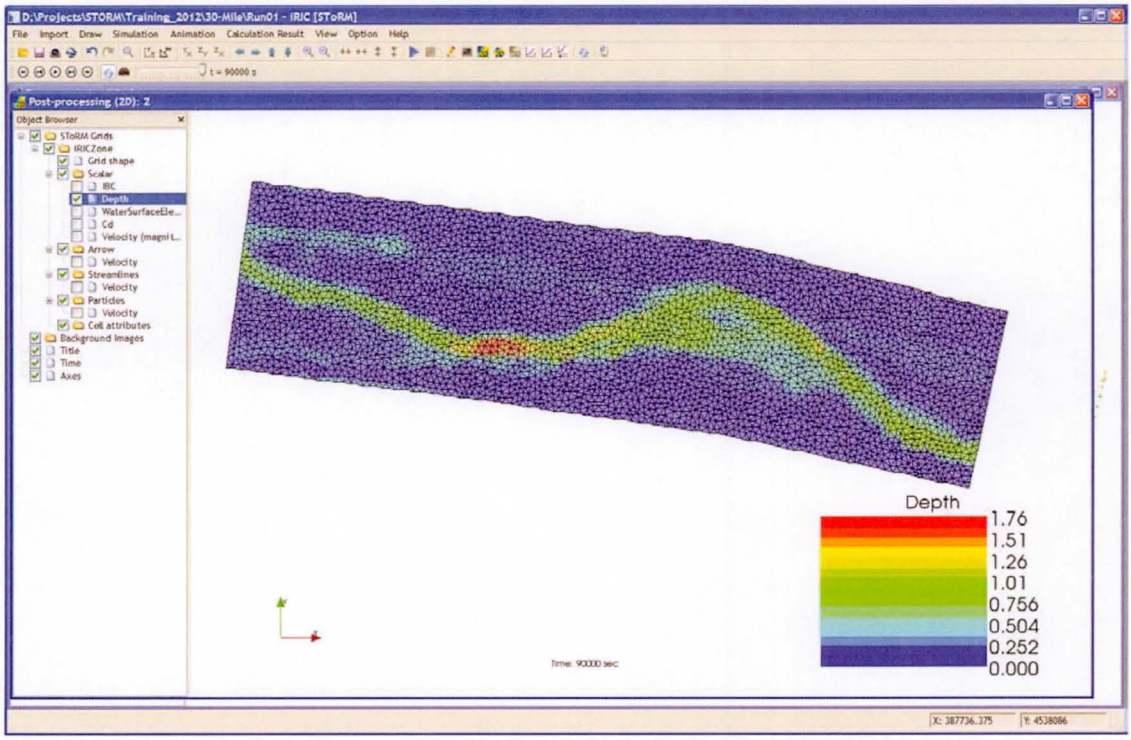


Figure 5.2

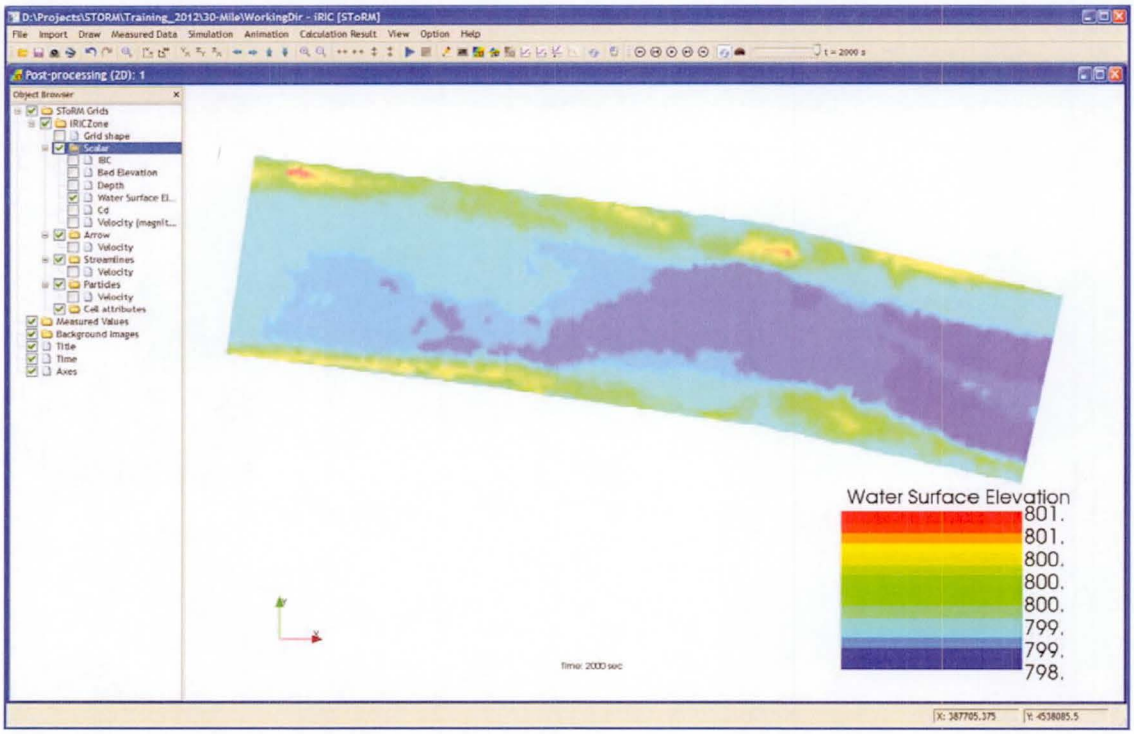


Figure 5.3

iRIC offers the possibility of analyzing the time-dependent data by plotting each output time step (requested in the **Calculation Condition** window of Fig. 4.2), which was set to 3-second intervals. Note the time tools toolbar of Fig 5.4, which appears in the previous figures. It indicates that the data viewed corresponds to a time of 2,000 seconds (which is also printed in the main display canvas in Figs. 5.1 to 5.3). The solution corresponding to the other times steps can be accessed by moving the slider in Fig. 5.4 to the desired position. For example, the magnitude of the velocity for second 840 is shown in Fig. 5.5 (only wet cells are colored).

Figure 5.4

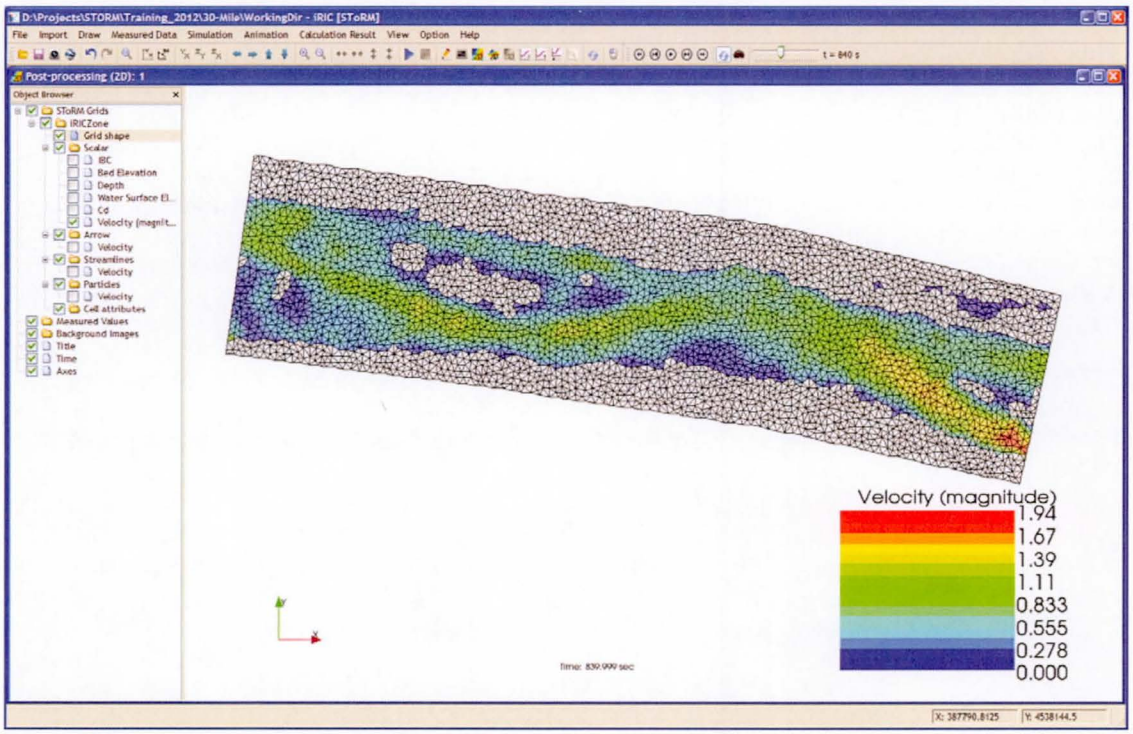


Figure 5.5

The buttons at the left of the toolbar in Fig. 5.4 are used to view the solution in different time steps and can be used to produce a timed animation of the flow solutions. They are easy and intuitive to use, and it is left to the reader to try them out. The buttons are:

- Go to first recorded solution.
- Step back in time by one solution.
- Start animation, starting on currently displayed solution.
- Stop ongoing animation.
- March solution forward in time by one.
- Go to last recorded solution.

6. Final Remarks

In its current distribution, SToRM uses time steps of fixed size, which is set in the **Calculation Conditions** window for each simulation run. Additionally, SToRM uses an explicit type of time marching scheme, which results in numerical stability constraints: simulation runs may become numerically unstable if the time step is too large. Stability is, thus, limited by the most severe condition encountered anywhere during a simulation run. In the example presented in this tutorial, stability issues are encountered in the earliest stages of the simulation, in the first few time steps, but not later. During these earlier time steps, time step size must be very small.

There are two ways to deal with this phenomenon. One way, which is the most accurate, is to run the first few time steps with a very small time step, stop the run after these are overcome, and restart it again with larger time steps. Another method, which is simpler and was used in this tutorial, is to limit the maximum possible velocity to a certain value. This was done in the **Calculation Conditions** window by setting **Allow large time steps** to **Yes** in the pull-down menu

and assigning a value of 10 m/s to the **Maximum velocity (m/s)** option. This has the effect of clipping the highest possible calculated velocity to 10 m/s.

Using the latter approach allows the simulation to progress past the difficult time steps without numerical instabilities, but accuracy may be somewhat affected, depending on the number of cells affected by the time step size restrictions. Fig. 6.1 shows the velocity field (vectors and magnitude of velocity) for the solution obtained in this exercise; Fig. 6.2 shows the same solution obtained with a much smaller time step of 0.001 seconds without velocity clipping. The area shown is near the inflow region. Although at this stage (at 60 seconds) the two solutions are similar, they are some differences. It is up to the user to ensure that the solutions obtained by the model are physically meaningful and accurate.

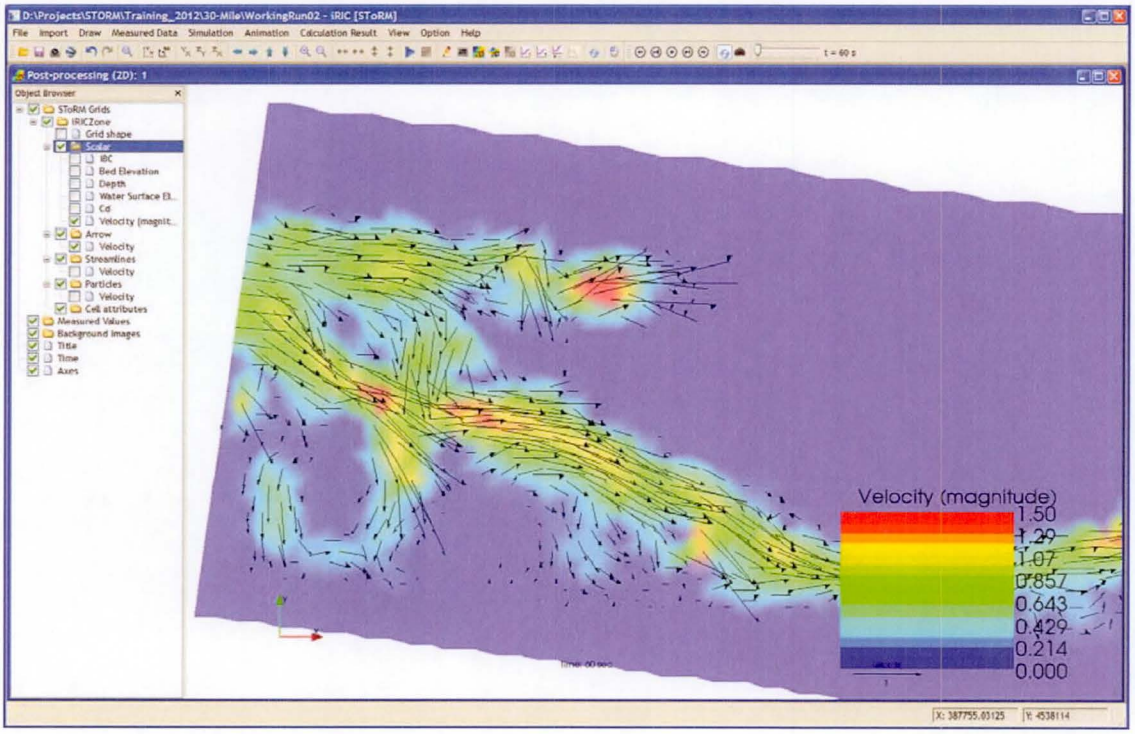


Figure 6.1

If, on the other hand, steady state solutions are sought, the accuracy of the earlier intermediate solutions is not important. In this case, velocity clipping can be used and the final steady-state solution is not affected as long as the maximum computed velocity is lower than the clipping value entered in the **Calculations Condition** menu. That can be easily verified by the user by plotting the magnitude of the velocity and ensuring that its maximum value is lower than that value. For example, in Figs. 6.1 and 6.2 the highest velocity magnitude is about 1.5 m/s, much lower than the prescribed 10 m/s of Fig. 4.2.

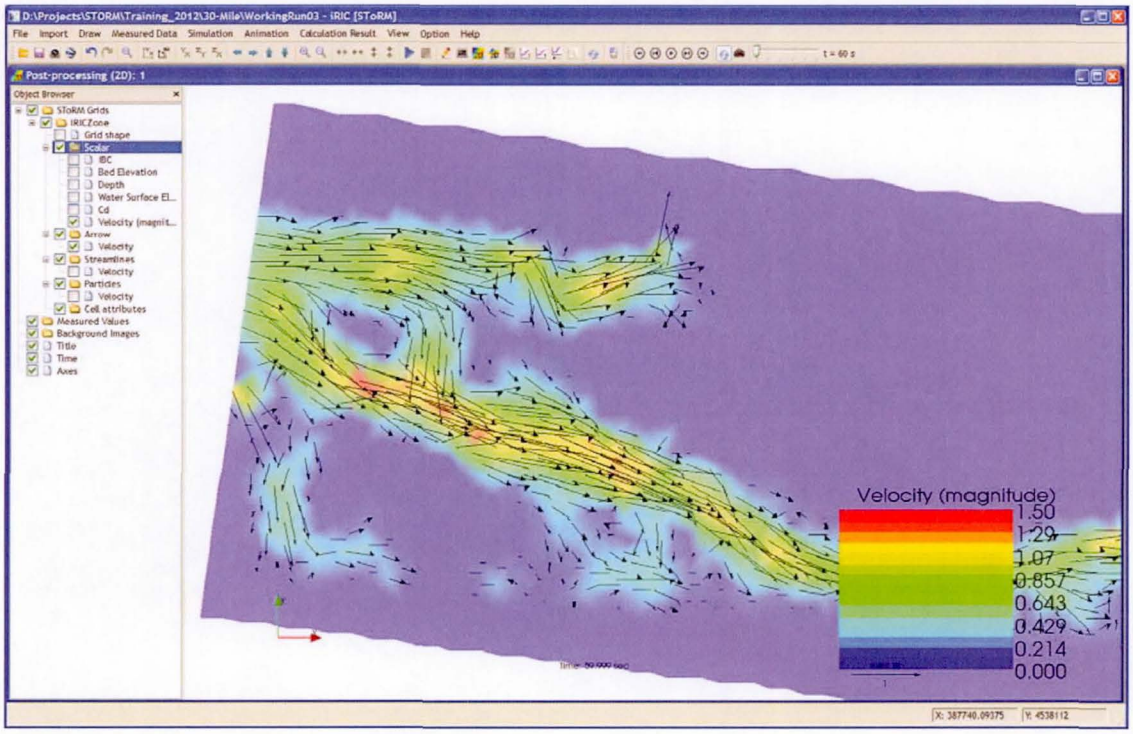
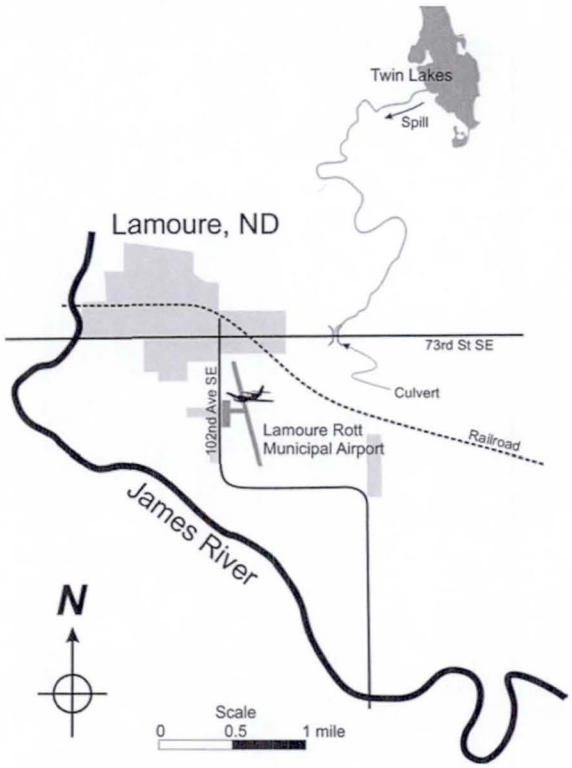

D:\Projects\STORM\Training_2012\30-Mile\WorkingRun03 - iRIC [SToRM]
File Import Draw Measured Data Simulation Animation Calculation Result View Option Help
Post-processing (2D): 1
Object Browser
SToRM Grids
iRICZone
Grid shape
Scalar
IBC
Bed Elevation
Depth
Water Surface El...
Cd
Velocity (magnit...
Arrow
Velocity
Streamlines
Velocity
Particles
Velocity
Cell attributes
Measured Values
Background Images
Title
Time
Axes
Velocity (magnitude)
1.50
1.29
1.07
0.857
0.643
0.429
0.214
0.000

Figure 6.2

TUTORIAL 5: Flooding of Lamoure, ND

SToRM (System for Transport and River Modeling) is a time-dependent hydraulic flow model specifically developed to calculate rapidly varying surface water flows with unsteady inflows and moving wetting/drying fronts. This tutorial shows how to set-up SToRM using the iRIC[1] 2.3 graphical user interface (henceforth called iRIC only, for simplicity). It covers all the necessary steps for a successful SToRM simulation, such as using bathymetric data and generating a computational grid, defining initial and boundary conditions, and setting up model parameters.

This tutorial shows how to determine the flooding conditions in a semi-urban setting. The source of the flooding waters is Twin Lakes, which, under certain extreme hydrologic conditions, may spill over an emergency spillway (see figure below). The downstream spill channel is normally dry and drains all the way to the town of Lamoure, ND. The entire area is initially dry and, as water passes the culvert under 73rd St., it gradually becomes inundated. Flood delineation and time of arrival are two of the aspects investigated in this study.



The example presented here is based on a flood inundation project carried out by the North Dakota State Water Commission for the town of Lamoure, ND. The focus of the project was to determine the time and extent of the flooding affecting Lamoure's Rott Municipal Airport for spills from Twin lakes corresponding to hydrologic conditions with different return periods. The data used here was taken directly from the ND data server (http://lidar.swc.nd.gov/).

[1] iRIC stands for International River Interface Cooperative (http://i-ric.org/en/).

Contents:

1. Getting Started 2
2. Preparing Bathymetric Data 5
3. Importing Ancillary Data 9
4. Creating the Computational Mesh 11
5. Prescribing Boundary and Initial Conditions 19
5.1. Culverts and Other Hydraulic Structures 25
5.2. Boundary Roughness 27
5.3. Initial Conditions 28
6. Running the Model 29
7. Visualization of Results 33

1. Getting Started

iRIC can be launched the usual way: from the Windows Start menu, or by double-clicking on its icon on the desktop. When it starts, iRIC offers several options (Fig. 1.1, left): **Create New Project** and **Open Project File**. In this tutorial, boldfaced words are used to represent iRIC commands, menu items, and general items found anywhere in the iRIC graphical interface. **Open Project File** opens a file that was previously created by iRIC and stored somewhere in your storage devices. Here, we are interested in starting a new project: click **New Project** (Fig. 1.1, left), then select STOrM in the following screen and click **Ok** (Fig. 1.1, right).

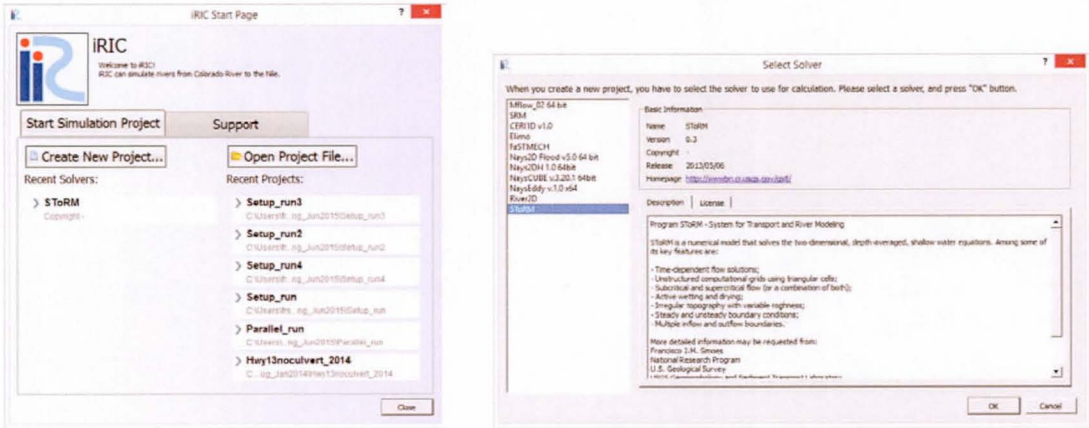


Figure 1.1

The iRIC graphical user interface (GUI) contains multiple windows, buttons, and pull-down menus. This is depicted in Fig 1.2, which shows the general look of iRIC when a new STOrM project is started. Different areas are identified there, which will be used throughout this tutorial.

The **Pre-processing Window** opens by default when iRIC is first started. It is used to display data and prepare several of the quantities needed to run STOrM. It is used for data pre-processing, i.e., for data that needs to be prepared before STOrM can be run. Examples of this type of data are topographic information, computational grid, boundary conditions, and model parameters.

There is also a **Post-processing Window**, which is used to visualize model results. This window, which is independent from the **Pre-processing Window** and that can be displayed concurrently to it, is used after the model is run to process and interpret the data produced by STOrM.

Examples of this type of data are the computed velocity vectors, water surface elevation, and plots of comparisons between computed and measured quantities for model calibration. Examples of these will be shown later in this tutorial.

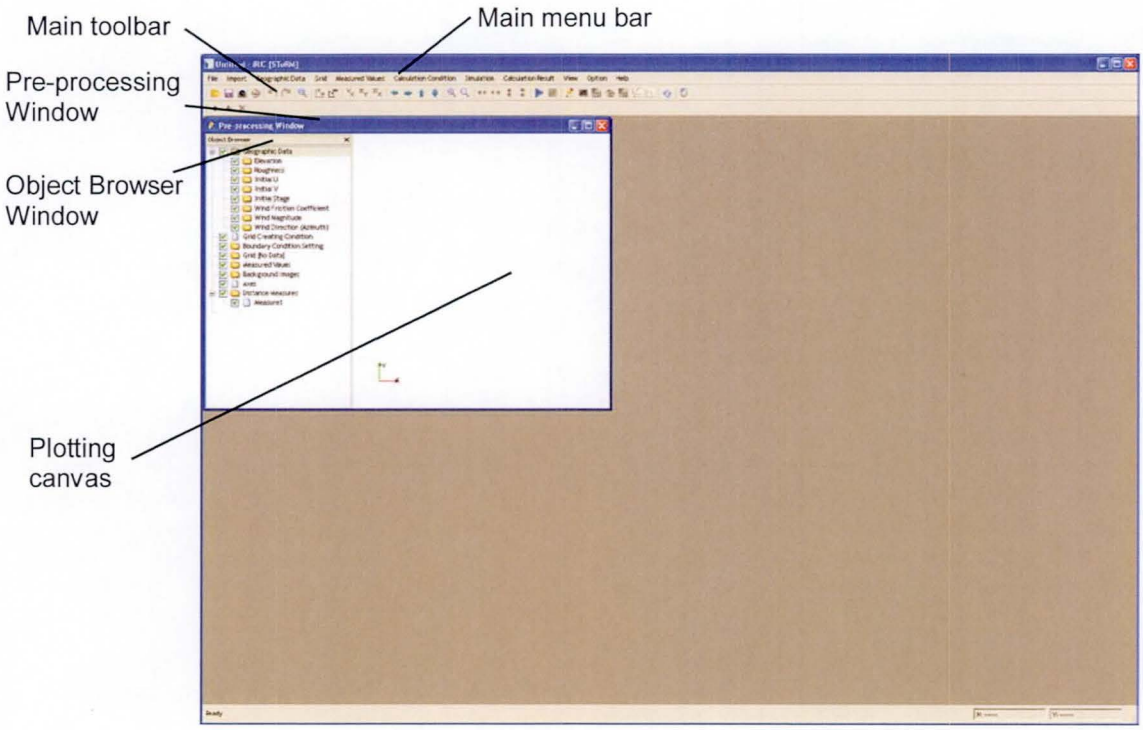


Figure 1.2

The main menu bar contains many of the commands used in iRIC, many of which will be seen later in this document. The main toolbar contains buttons that provide quick access to some of the functions more frequently used. The **Object Browser Window** is used to control the appearance of the items in the plotting canvas of the **Pre-processing Window**. Some of the buttons appearing in the main toolbar are:

- Open an existing project
- Save the current work
- Save a snapshot
- Export data to Google Earth
- Undo last operation
- Redo last undone operation
- Fit all data to the current plotting canvas
- Reset rotation of the viewing canvas to zero degrees, which is the default view
- Rotate the data by 90 degrees counterclockwise (data view can also be rotated by right-clicking the mouse while holding the <Ctrl> key)
- Move data to the left (data can also me moved by left-clicking and dragging the mouse while holding the <Ctrl> key)
- Move data to the right
- Move data up
- Move data down
- Zoom in
- Zoom out

Other tools will be explained later. Note that a help balloon appears when hovering over the icons in the main menu bar. Some of the icons may be grayed out, indicating that the corresponding operation cannot be carried out.

Using iRIC and SToRM in a productive work environment usually results in an iterative cycle in which successive model runs hone into a progressively more accurate solution, each benefitting from feedback from the previous ones. The workflow of an iRIC/SToRM project and consists in the following schematic steps:

1. Import bathymetric data into iRIC.
2. At this point ancillary data can also be imported into iRIC. Ancillary data may consist of items such as field data and orthorectified aerial imagery.
3. Analyze and edit the data.
4. Prepare a computational grid for use by SToRM.
5. Define initial and boundary conditions, including roughness factors and other driving parameters.
6. Specify model run parameters, such as choice of numerical scheme(s) and duration of the run.
7. Run SToRM. May take minutes, hours, or days.
8. Analyze the model results, compare with measurements, all while exercising critical engineering judgment.
9. Are model results grid-independent? If yes, proceed to the next step; if not, backtrack to steps 4.
10. Is the model properly calibrated? If yes, proceed to the next step; if not, backtrack to steps 5 or 6.
11. Are model results satisfactory? Do they pass critical scrutiny? If yes, proceed to the next step; if not, backtrack to steps 4, 5, or 6.
12. Production run completed.

This process is schematically represented in the flow chart of Fig. 1.3. The use of the technical tools comprised in iRIC 2.3 facilitates accomplishing these tasks.

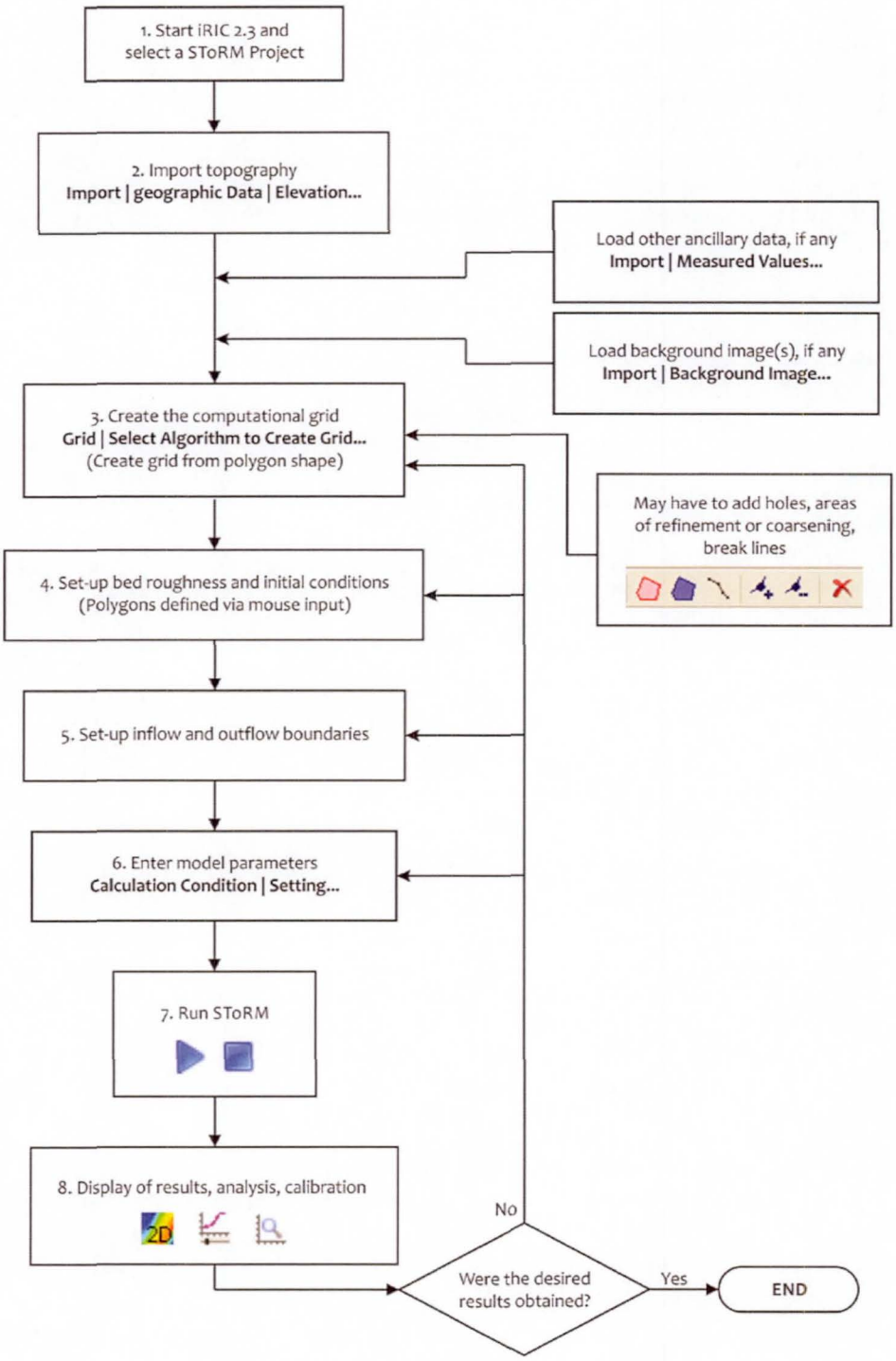


Figure 1.3

2. Preparing Bathymetric Data

Bathymetric data contains the terrain information (bathymetry is the underwater equivalent of topography; because under different flow conditions the terrain can be dry or submerged, both

terms are used interchangeably in this tutorial). This data is usually the result of a field survey program and must be referenced to some common datum. An ideal dataset covers all the area of interest with sufficient spatial resolution for the desired application.

The first step consists in loading the topographic data into iRIC. The data used in this tutorial is located in ShiftedDataSet-3m.tpo. This file type is an ASCII editable file that contains the coordinates and elevation of each data point, in metric units[2], with 3-meter resolution. The beginning of this file looks like this (only the first 7 lines of data are displayed):

```
2217392
449.336 3951.541 397.688
452.336 3957.541 397.719
452.336 3954.541 397.717
452.336 3951.541 397.68
452.336 3948.541 397.693
455.336 3963.541 397.664
...
```

The first line indicates the number of data points contained in the file. The triplets x,y,z that constitute the coordinates of each point (space-delimited) are then given one to a line. Note that iRIC can import several other data formats. To load this data into iRIC:

- Use pull-down command **Import | Geographic Data | Elevation...** from the main menu bar.
- *Find the data file in the folder where it is located and click OK in the pop-up window.*
- The **Filtering Setting** dialog window opens (Fig. 2.1). This window allows resampling very large data sets by reducing the number of points imported to iRIC to alleviate computer memory limitations. It works by defining the sequence of data points read into iRIC in the **Filter** variable: 1 means "read every point;" 2 means "read every second point;" 3 means "read every third point;" etc. Depending on the capability of a computer, filtering may or may not be necessary.

(Note about notation used in this tutorial: bold faced words separated by a vertical bar denote a cascade of menus and submenus in iRIC, or a hierarchy of objects in the **Object Browser**.)

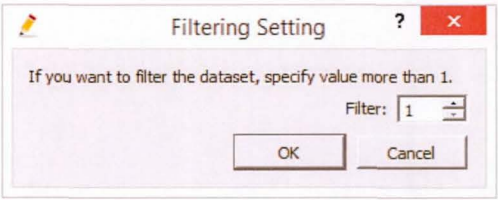


Figure 2.1

The final result is shown in Fig. 2.2 (after resizing the **Pre-processing Window**). The individual survey points are plotted, colorized by bed elevation.

[2] **Important:** all units used in SToRM are metric unless otherwise specified.

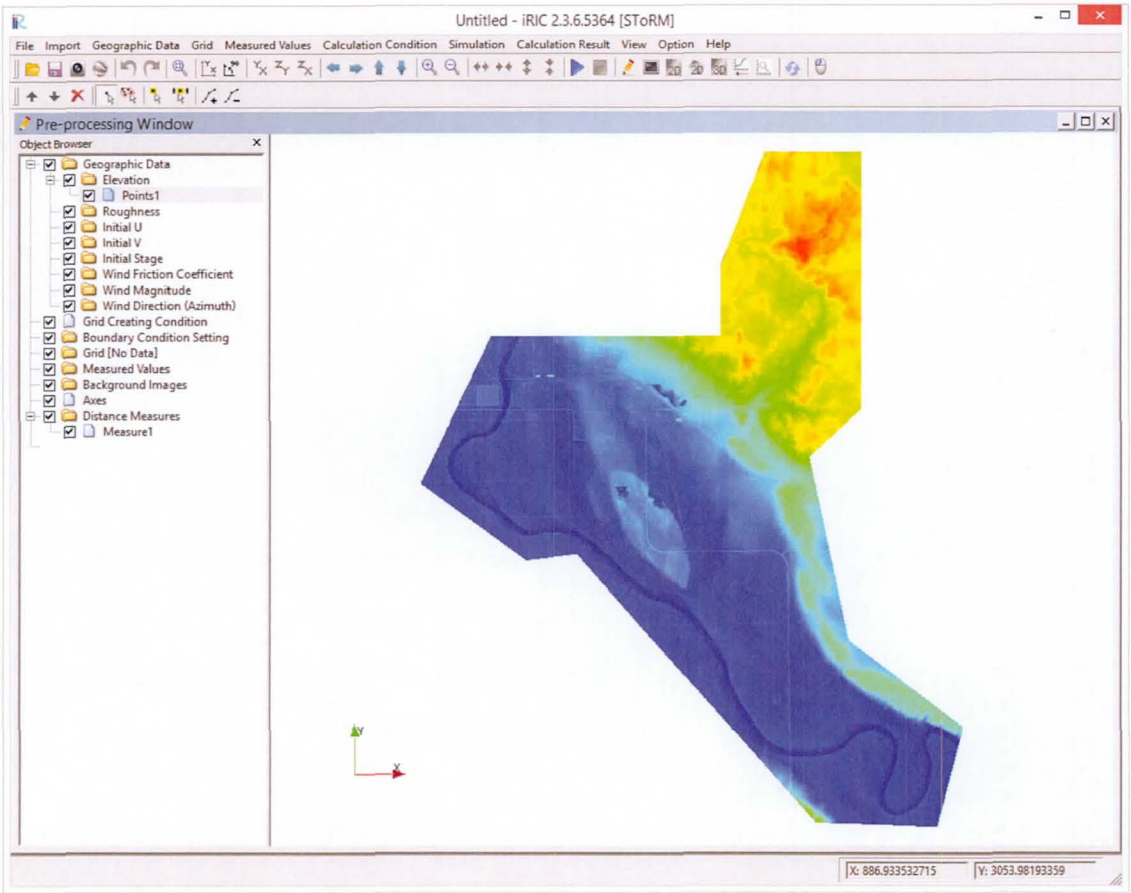


Figure 2.2

Note the objects in the **Object Browser** window (Fig. 2.3). Among other things (which will be seen later), this window can be used to turn specific aspects of the data on or off in the main display canvas. For example, click on the checkbox to the left of **Points1** to uncheck it and the data plot will disappear. Click on it again to check it and the data plot will reappear. This is true for all the graphic entities created during the workflow. The color range is calculated automatically from the data, but it can be changed from an appropriate control window (Fig. 2.4, left), which is brought up by highlighting **Points1** in the **Object Browser** window (highlighting of any item in the **Object Browser** window is accomplished by left-clicking the mouse on the desired object) and then right-clicking the mouse on **Geographic Data | Elevation** and selecting **Color Setting** from the pop-up menu. The same pop-up menu has other functionality, such as controlling the display color legend. The use of this menu is simple and straightforward.

Right-clicking the mouse on **Points1** directly brings up another pop-up menu with different functionality. For example, it is possible to change the name of the dataset (**Points1** is the name assigned to the data by default) and the points' appearance. To change how the topographic data is plotted:

- Right-click the mouse on **Geographic Data | Elevation | Points1** and select **Property**. The window in Fig. 2.4 (right) will appear. Click OK when finished.

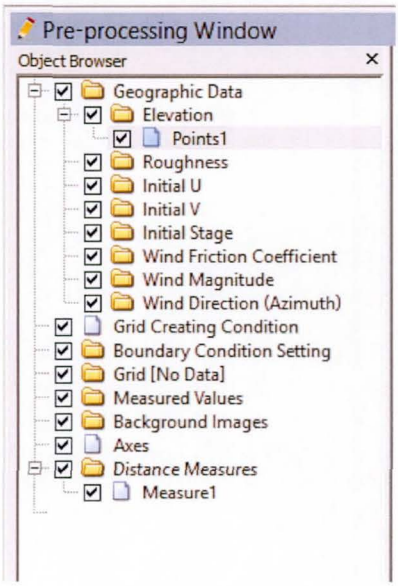


Figure 2.3

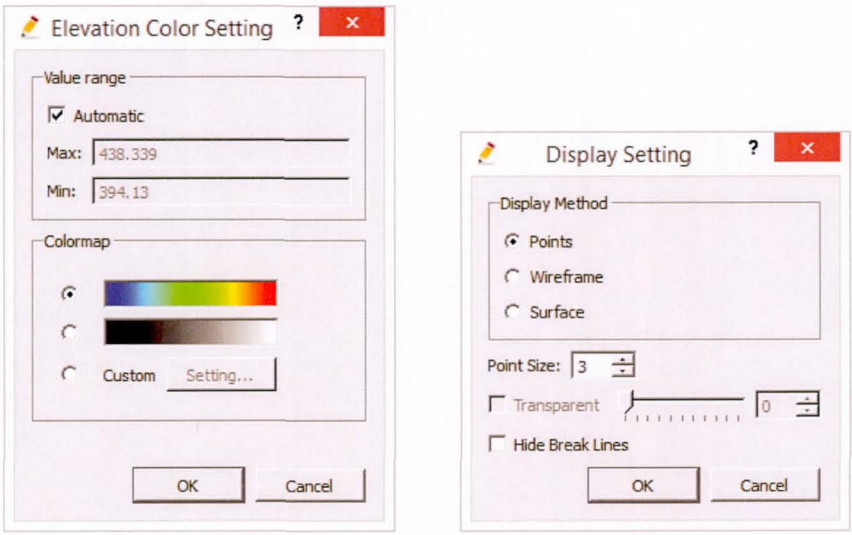


Figure 2.4

It is now possible to carry out a number of exploratory analyses of the topographic data in iRIC. Using the mouse buttons, it is possible to interactively pan, zoom, and rotate the viewport: panning of the viewport is accomplished by clicking and holding the left mouse button and the control keys simultaneously; zooming in and out is accomplished by the use of the mouse wheel; rotation is done holding the control key and the right mouse button. The viewport can be restored using the button in the **Main Toolbar**.

The display characteristics of the bathymetry data points can also be changed and customized. This is accomplished by selecting **Geographic Data | Elevation | Points 1** (left-click on it in the **Object Browser** window) and right-clicking on it to access a pop-up menu that allows to edit the name of the data, export it, delete it, and adjust its properties, such as point size and display method.

A legend can be added to the display canvas by selecting **Geographic Data | Elevation**, right-clicking on it, and selecting **Set Up Scalarbar** in the resulting pop-up menu. Make sure that **Elevation** is displayed in the drop-down menu and the **Visible** box is checked. The **Edit** button gives access to additional features that control the legend display. These menus are intuitive and easy to use, and the student is encouraged to try them to see their effects in the way data is presented.

To conclude this section, save your work. To save the project:

- From the main menu, chose **File | Save as Project**. Never use command **Save as File (*.ipro)** when using SToRM.
- A pop-up window (Fig. 2.5) lets you select the location where the data will be saved. Select the desired location and click on **Make New Folder**. Assign it a name and click OK.
- This operation creates a new folder containing all the data used by iRIC and by SToRM. The data generated later by SToRM run is saved in a CGNS-formatted file with extension .cgn. CGNS (CFD General Notation System) is a data format used widely in the computational fluid dynamics community.

SAVE YOUR WORK OFTEN.

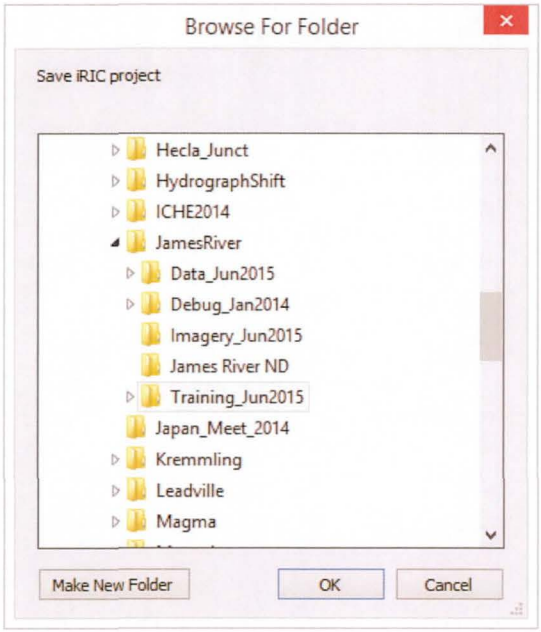


Figure 2.5

3. Importing Ancillary Data

It is possible to use different types of geo-referenced data in iRIC. These data can be scalar data, vector data, and images. Importing data is easy and the necessary steps and file formats have been shown elsewhere. Here, we are only interested to import a geo-referenced image to be used as auxiliary backdrop. This provides a much more interesting visual feedback when working with iRIC. To import a background image:

- Select **File | Import | Background Image** from the main menu. Alternatively, **Import | Background Image** can also be used.

- In the **Open Image** dialog window, select BaseImageShifted.png[3].

The image is now displayed in the main drawing canvas under the topographic data, as shown in Fig. 3.1, and has its own entry in the **Object Browser** window. The image used in this tutorial has a corresponding world file that positions each image correctly in the coordinate space—file BaseImageShifted.pgw, which is loaded automatically by iRIC. Images without such information can also be used in iRIC, but they must be adjusted manually by translating, rotating and scaling them appropriately:

- To adjust the location of an image, select its name under **Background Images** in the **Object Browser**.
- When an image is selected, the left mouse button drags the image, the middle mouse button scales the image, and the right mouse button rotates the image.

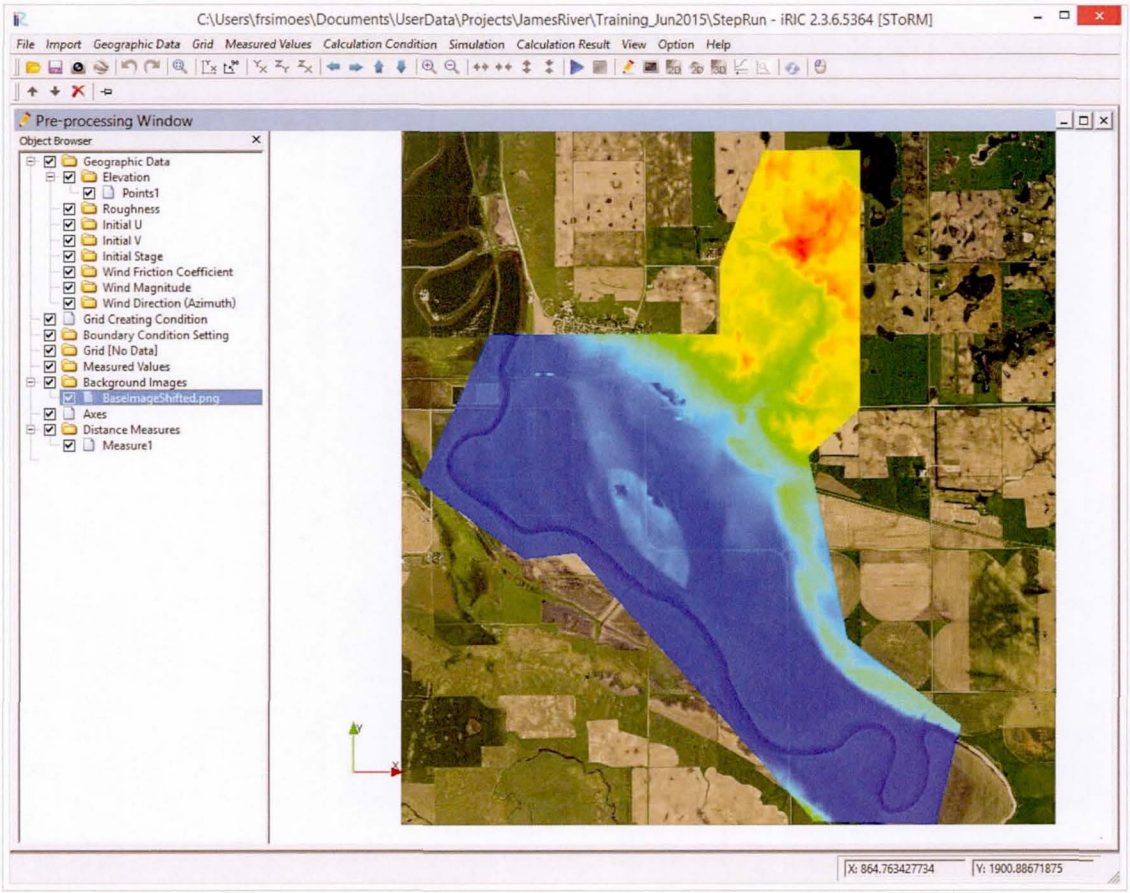


Figure 3.1

At this point it may be convenient to save your work.

[3] A smaller sized image can be found in file BaseImageSmallerShifted.png. It may be beneficial to computer performance to instead load this image, especially if graphics memory is limited.

4. Creating the Computational Mesh

Most numerical models for surface water flow use a computational mesh (here, the term grid is used interchangeably) to calculate the flow solution. The mesh consists in a set of points (the term nodes is also used interchangeably) with some type of connectivity among them. For example, that connectivity provides a link between a point and all of its neighbor points, i.e., it is used to define relations of proximity between nodes. Models compute the flow solutions on the discrete points of the mesh, and use interpolation methods to find flow parameters in the areas between the points. There are many different types of mesh configurations possible, but they can all be classified in one of two broad classes: structured or unstructured. In structured grids, the nodes are placed in an ordered row-column matrix, as shown in Fig. 4.1 (left). In unstructured grids, the nodes are placed in a scattered cloud without apparent symmetry. There are many ways to define an unstructured grid. STORM uses a lattice of contiguous non-overlapping triangles and computes the flow variables (velocity and depth) at the geometric centers of those triangles, marked by circles in Fig. 4.1 (right). This section is devoted to show the necessary steps to build an adequate grid that can be used for STORM computations.

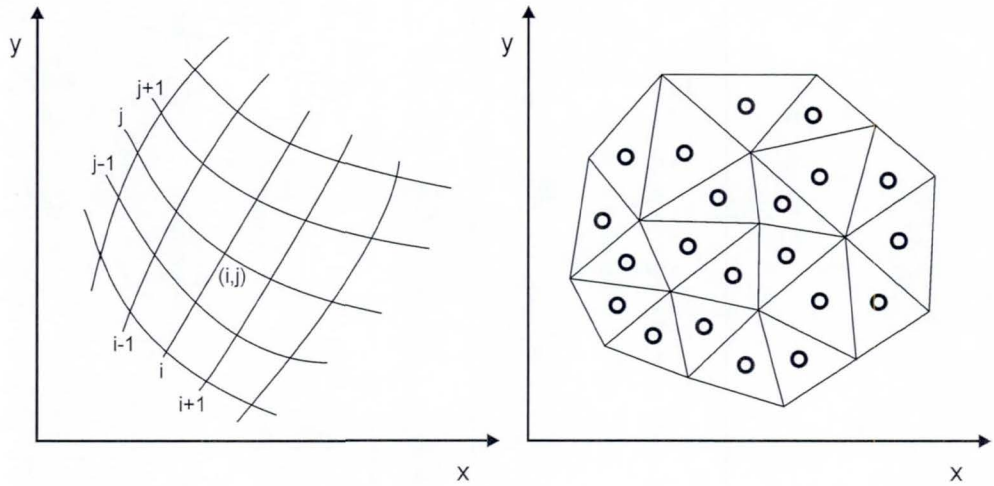


Figure 4.1

The area of interest for the current flooding study is centered at Lamoure's airport. It is convenient to place the computational grid outline far enough from the area of interest to minimize boundary condition effects, therefore we expand the area of interest to also include the 73rd St culvert, the city of Lamoure, and an area south of the airport, as outlined by the red circle in Fig. 4.2.

In iRIC, the computational grid is created automatically by a triangulation program using a few parameters defined by the user. These parameters are: a polygon encompassing the area of interest, maximum area of the triangles, and their minimum allowed internal angle. The steps needed to create the grid are:

- Zoom and pan to the area of interest, which can be accomplished by using buttons in the main toolbar. Zooming can also be done by rotating the mouse wheel and panning by right-clicking the mouse and moving it on the display canvas while pressing the <Ctrl> key.
- From the **Object Browser**, right-click on **Grid Creating Condition** and on **Select Algorithm to Create Grid...** in the ensuing pop-up menu. Click on **OK** in the **Select Grid Creating Algorithm** window.

- The computational domain is defined by a polygon that encompasses the area of interest. Define it by left-clicking the mouse at the desired polygon vertices, in the proper sequence. iRIC provides guiding visual feedback during this process. Double-click the last point or press <Enter> on the keyboard to finish the polygon. Use Fig. 4.3 to guide you on the shape and location of the polygon.

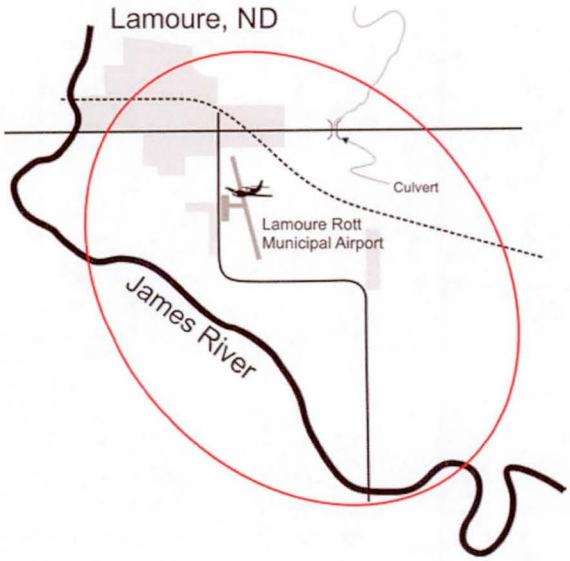


Figure 4.2

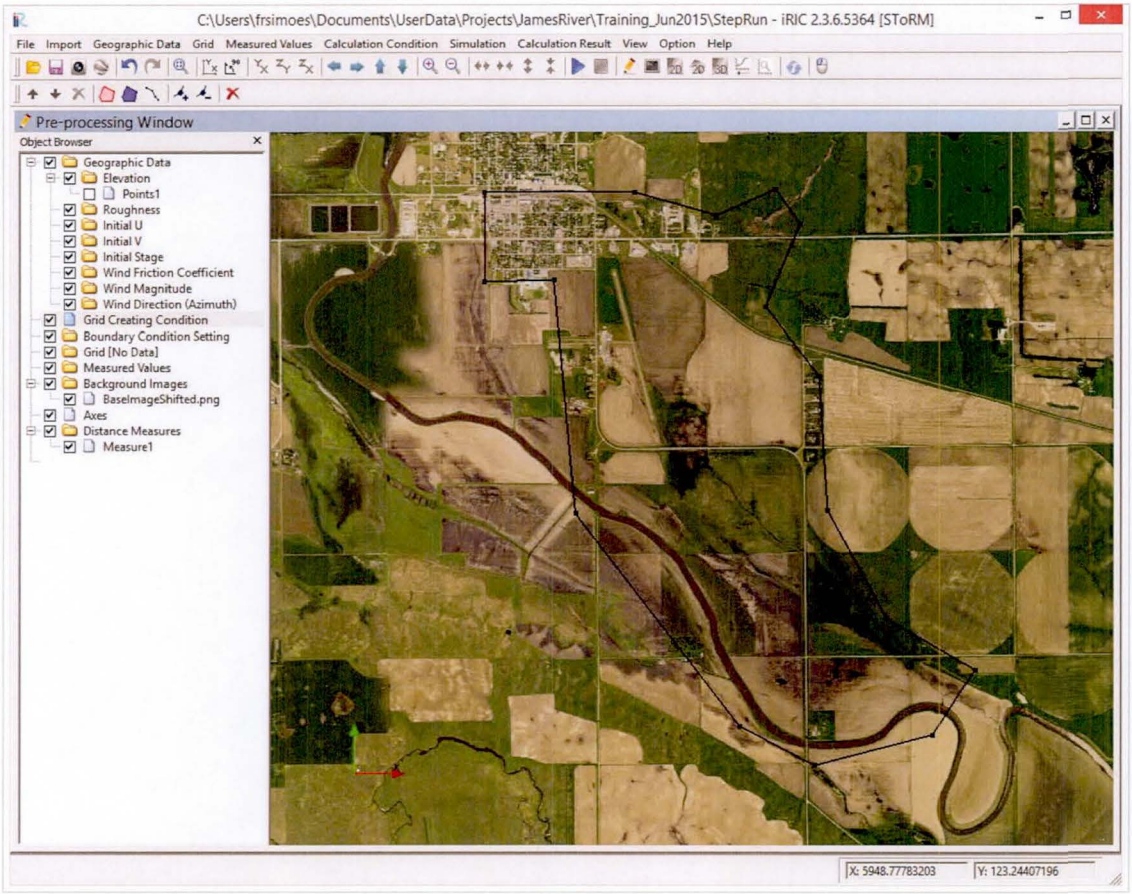


Figure 4.3

It is possible to edit (delete and add) and change the location of the vertices of the grid polygon. To accomplish these operations do the following:

- Left click on **Grid Creating Condition** in the **Object Browser** window. This will activate the editing menu bar . Next, left click on the mesh polygon. The grid polygon is highlighted and its vertices appear as black squares.
- The mouse can now grab individual vertices: when left clicking the vertex, the cursor changes from an open hand shape to a closed hand shape, which means it is grabbing that vertex. Move the mouse without releasing the left button and the vertex will move with it. Release the mouse button at the desired new vertex location. This process is depicted in Fig. 4.4, where the cursor is placed near a vertex and takes the shape of an open hand (Fig. 4.4, left), then the mouse is clicked and the vertex is moved (Fig. 4.4, right).
- Grabbing near or on top of a vertex moves the vertex; grabbing the frame moves the entire polygon to a new location.
- New vertices can be added to the grid polygon: left click on the button, then left click on the desired location for the new vertex. This location must be on the polygon, and not inside or outside of it.
- Similarly, and existing vertex can be deleted from the polygon: left click on the button, then left click on the vertex to be deleted.
- The above operations can be undone using the undo button in the main toolbar.

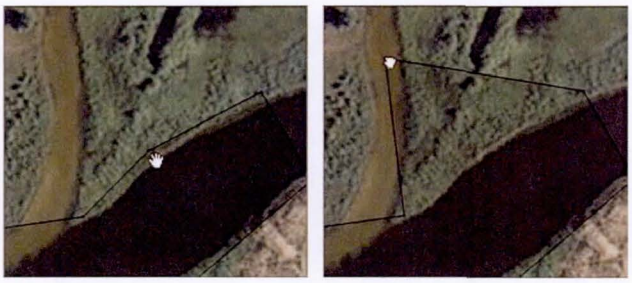


Figure 4.4

In iRIC, the computational grid is generated automatically by a triangulation program that uses quality constraints specified by the user: the minimum allowed internal angle of the triangles, and their maximum area. The objective of the internal angle constraint is to avoid creating triangles that are long and narrow, which have poor interpolating qualities and can impact negatively the accuracy of the model's numerical solutions. Ideally, the value chosen should be close to 60 degrees, which are the internal angles of an equilateral triangle. Unfortunately, the grid generator may become unstable if an angle greater than 32 degrees is prescribed, therefore the use of this value is recommended.

The maximum triangle area constraint is determined from considerations about the required resolution. In other words, it is desirable to have triangles that are small enough to capture all the features that determine the preferential paths that the water will take. A way to determine maximum triangle area is by using the formula for the area of an equilateral triangle whose edges are Δx in length:

$$A = \frac{\sqrt{3}}{4}\Delta x^2$$

Table 1. Area and edge length for various equilateral triangles.

Δx	A
10	43.3
10.7	50.0
15	97.4
20	173.2
21.5	200.2
25	270.6
30	389.7

Using this information, we are now ready to generate the mesh. To start the automatic mesh generator:

- Right-click on **Grid Creating Condition** in the **Object Browser**. This starts the grid generator.
- The **Grid Creation** window is used to define the basic parameters that control grid quality. Enter the values shown in Fig. 4.5.
- When prompted to map geographic data to grid, click **Yes**. This operation transfers the topographic data to the computational grid.
- Visualization of the grid can be turned on and off by left-clicking on **Grid** in the **Object Browser** window.

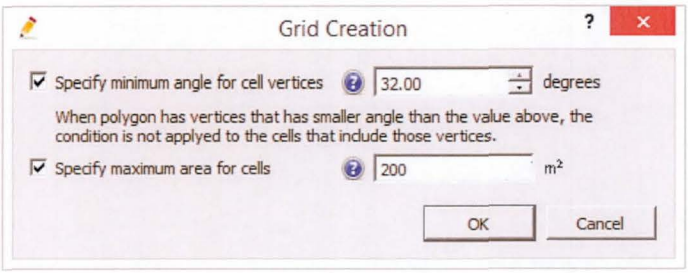


Figure 4.5

One of the means to capture terrain features is provided by break lines. Break lines force the grid generator to create triangles with edges lying on those lines. Placing break lines on certain terrain features allows the grid generator to create grids that capture important topographic details. In the present example, it is important to capture accurately road elevations, because roads, being at a higher elevation than the adjacent lands, pose an important obstacle to the flow of water. The next step is, therefore, to use break lines to improve the grid.

Break lines are open polygonal lines digitized by left-clicking the mouse on the display canvas. Each mouse click defines a vertex in the break line. A break line is ended by double-clicking the last vertex or by pressing <Enter> on the keyboard. To start the break line tool:

- Left-click on **Grid Creating Condition** on the **Object Browser** of the **Pre-processing Window**. This will bring up a toolbar: .
- Use button to create a breakline. Click <Return> (or duble-cklick the last point) when completed.
- The locations of break line vertices can be edited and new vertices can be added using the same procedure described to edit the grid's main defining polygon—see Fig. 4.4 and corresponding text.

Use Fig. 4.6 as a guide to create break lines over the most important roads and airport runway.

Click on button to start each new break line. Break lines must not intersect each other nor the edges of the main grid polygon. After all break likes are entered, generate the new mesh:

- Right-click on **Grid Creating Condition** in the **Object Browser** and select **Create Grid...**
- Click on **Yes** in the warning window.
- Click **OK** in the **Grid Creation** window.
- Click **OK** in the **Confirmation Window** about mapping the geographic data to grid attributes.

Using Table 1, it can be seen that the above choice of parameters provides a resolution of about Δx = 21.5 m, which may be too low for certain localized regions of the domain. iRIC contains tools that allow the computational mesh to be easily refined and/or coarsened. A mesh is refined when the new mesh uses smaller triangles than the original, and coarsened when larger triangles are used. Refinement and coarsening of the computational grid can done selectively, i.e., the mesh can be refined and/or coarsened only in selected areas of the domain. Selective refinement may be advantageous when only a small part of the reach needs a fine mesh, while the solution for the remainder of the reach can be obtained by the use of a larger mesh. Areas where finer grids are needed are areas where the variables (water and/or flow velocity) change rapidly, such as in recirculation areas behind obstacles, around the tip of groins or other salient features, etc. Using selective refinement is a good way to avoid having too many grid triangles when a few are

enough, thus reducing the total number of computational points and resulting in increased computational efficiency (i.e., SToRM runs faster).

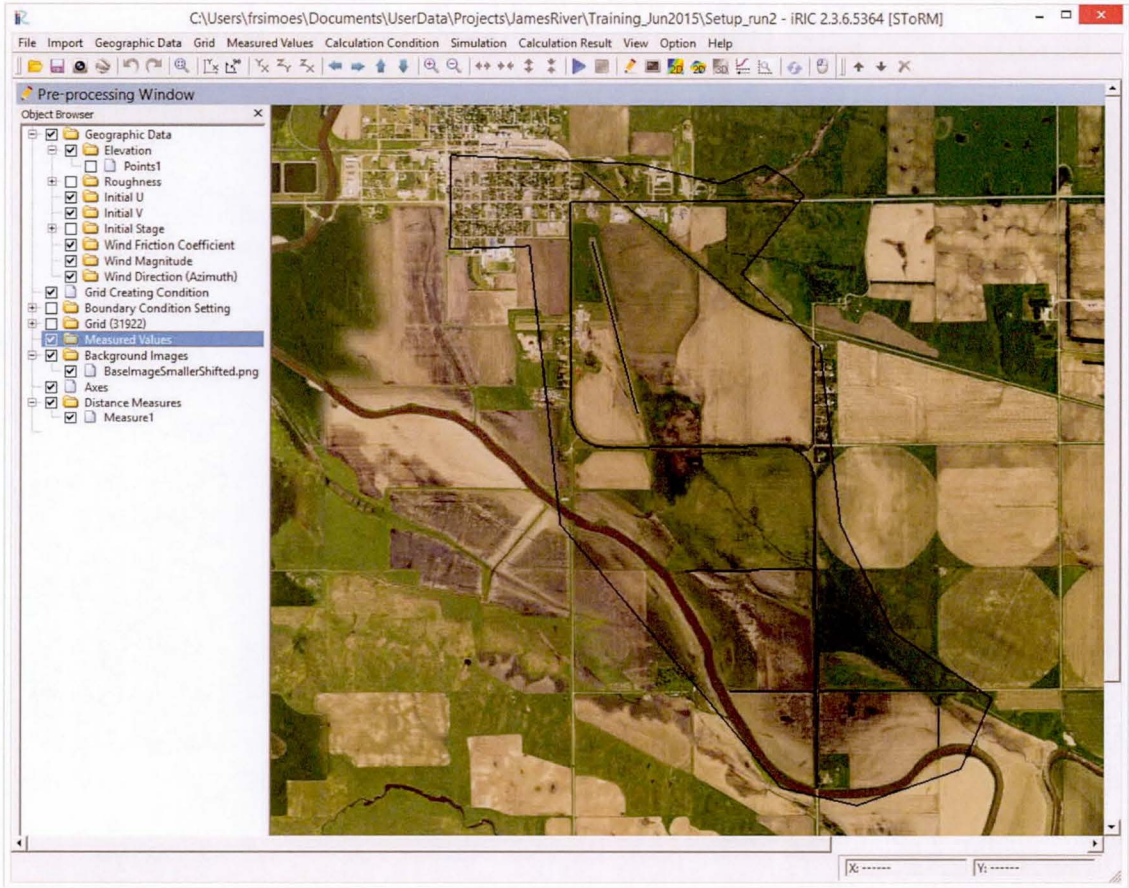


Figure 4.6

Areas of mesh refinement and coarsening are created by defining polygons within the main grid polygon. Here, we are interested in refining the solution in a small area near the upstream boundary and culvert. To help identify the location of the culvert's inlet and outlet, we can load the set of their coordinates to place visual markers on the display canvas. This is done as follows:

- Select **Import | Measured Data** from the main menu bar and select file CulvertShifted.csv.

A new entry will appear in the **Object Browser** window called **CULVERT**. At the same time, two points will appear on the canvas. These show the location of the culvert (see Fig. 4.7) and help identify the areas of interest.

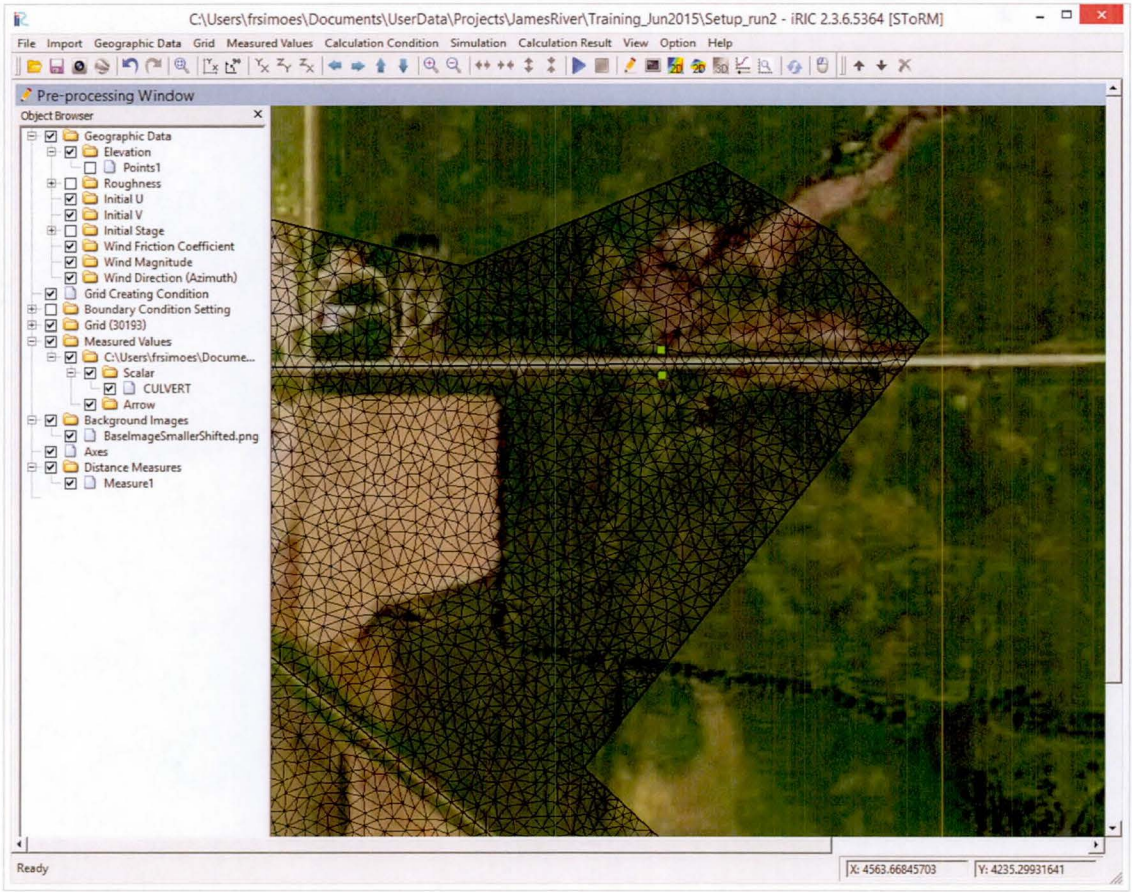


Figure 4.7

We are interested in creating two areas of mesh refinement: one north of 73rd St, covering the culvert inflow; other south of the same, covering the culvert outflow. The refinement polygons are created in the same manner as when creating the main computational grid polygon.

- Left-click on **Grid Creating Condition** on the **Object Browser** of the **Pre-processing Window**. This will bring up a toolbar: .
- Use button to create a refinement polygon. Left-click the mouse on polygon vertices. Click <Return> (or double-click the last point) when completed.
- Enter a refinement triangle area of 50 m^2 in the **Refinement Maximum Area** window and click **OK**.
- Digitize the next polygon in the same manner as above.
- Use Fig. 4.8 as guideline to create the refinement polygons.
- Note that the locations of polygons and their vertices can be edited, and new vertices can be added using the same procedure described to edit the grid's main defining polygon—see Fig. 4.4 and corresponding text.

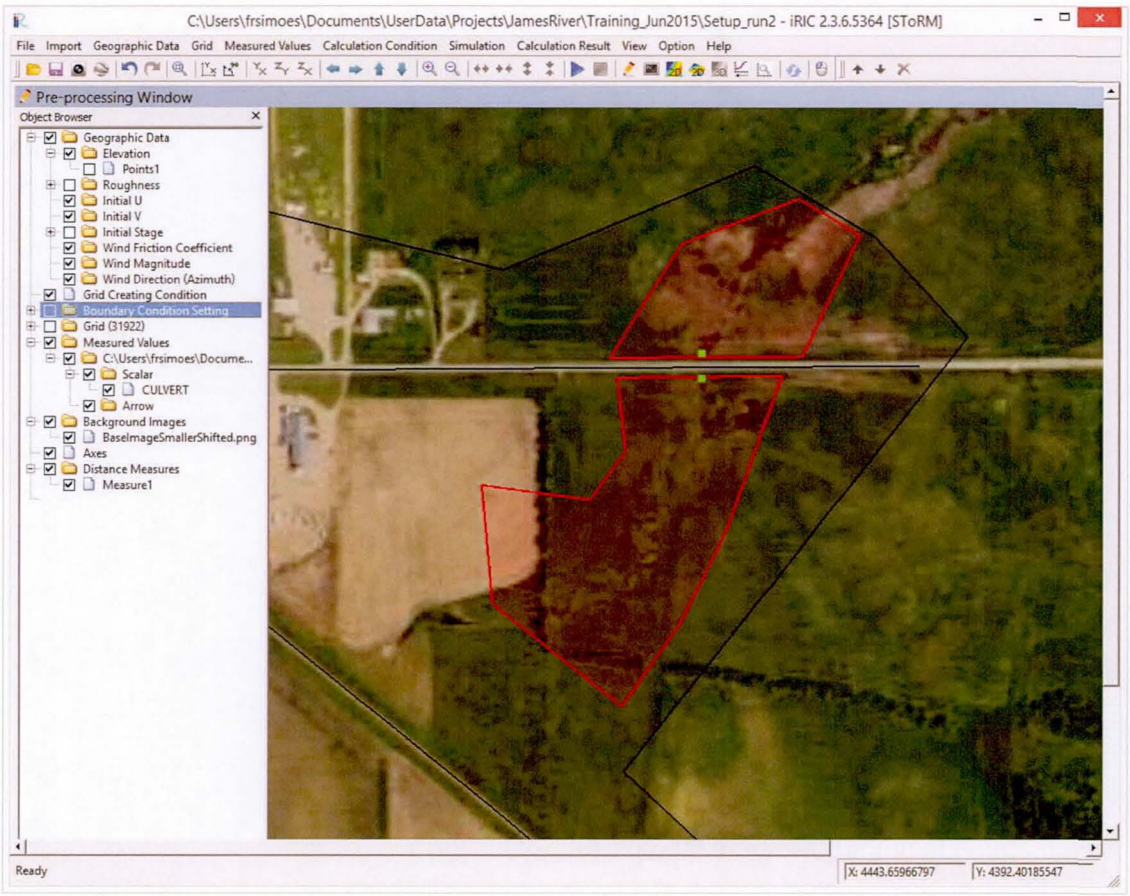


Figure 4.8

The edges of the refinement polygons must not cross over other edges or break lines. After both refinement polygons have been completed, regenerate the computational grid:

- Right-click on **Grid Creating Condition** in the **Object Browser** and select **Create Grid…**
- Click on **Yes** in the warning window.
- Click **OK** in the **Grid Creation** window.
- Click **OK** in the **Confirmation Window** about mapping the geographic data to grid attributes.

An example of the final grid is shown in Fig. 4.9.

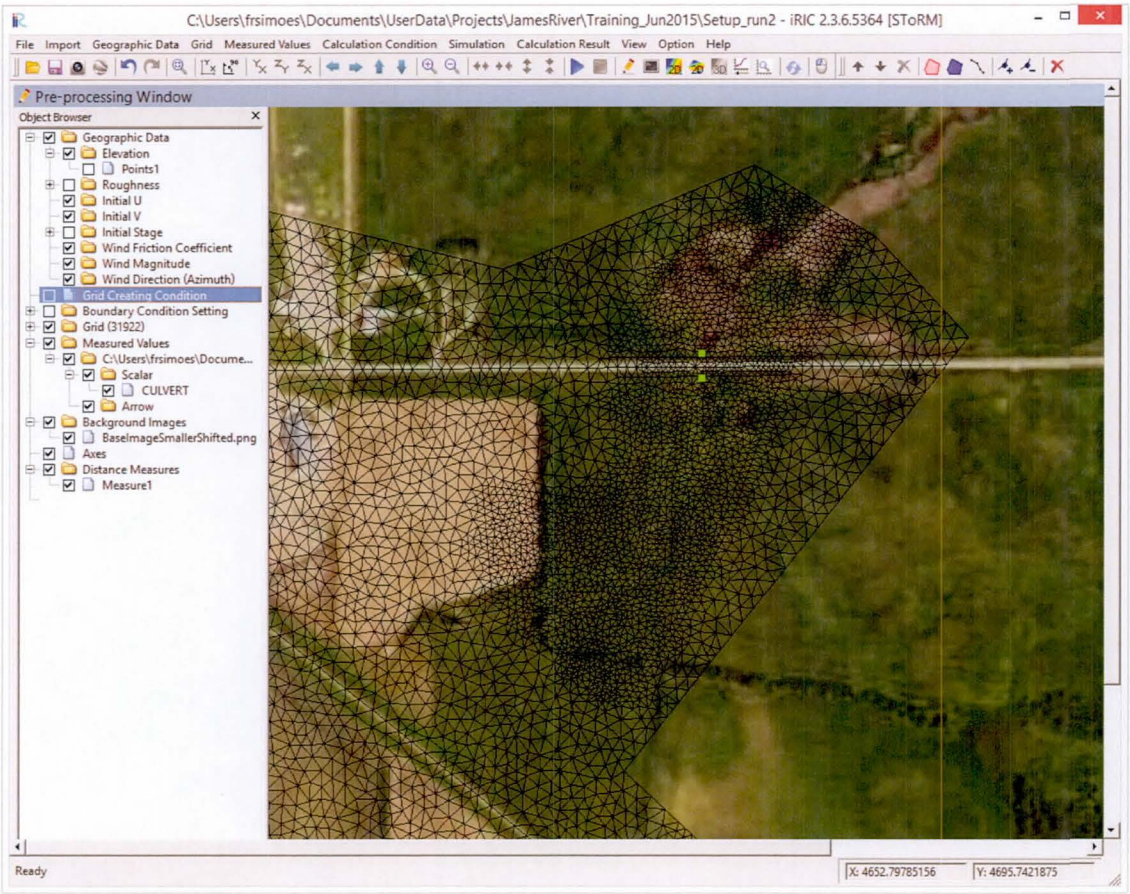


Figure 4.9

5. Prescribing Boundary and Initial Conditions

The computational domain defined by the grid is the only part of the river "seen" by SToRM during the calculations. In other words, the model does not have any sort of information about the region outside the computational grid. To be able to carry out the backwater calculations, it is necessary to know something about what is upstream and downstream from the computational region. Basic hydraulics principles require that, for subcritical river flow, the discharge entering the grid and the water surface elevation at the downstream-most region of the grid must be known. These regions, where the stage and discharge must be specified, are usually called the hydraulic boundaries of the domain. The quantities prescribed there (discharge and stage) are called the boundary conditions. This section illustrates how to define the boundary conditions in iRIC.

First, we are going to define the inflow boundary. This can be thought of as being the upstream-most cross section in the computational domain, and the flow discharge must be specified there. It is convenient to define this boundary as a straight line perpendicularly to the centerline of the river channel. Each inflow (and outflow) boundary is composed of a string of contiguous nodes located on the boundary of the computational grid. The inflow hydrograph was synthesized from a spill from Twin Lakes corresponding to a 100-year rainfall event and is given in file hydrograph+12.txt and plotted in Fig. 5.1. The steps below show how to mark the inflow nodes and assign an inflow boundary condition to them.

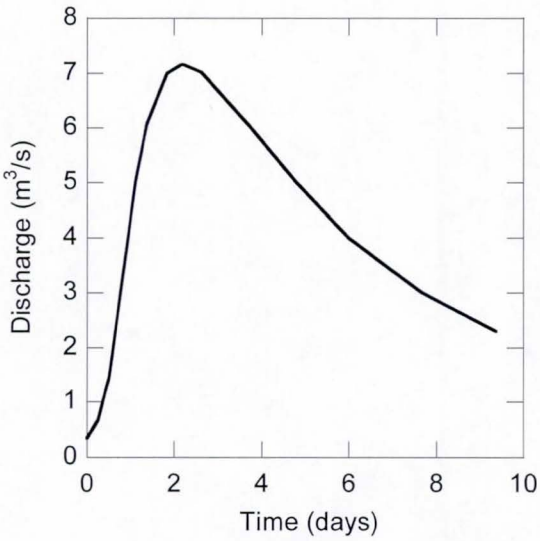


Figure 5.1

Zoom to area of interest, as shown in Fig. 5.2, and:

- Right-click on **Boundary Condition Setting** item in the **Object Browser** and select **Add Inflow** in the pop-up window.
- Define a polygon that encompasses all the grid nodes (triangle vertices) belonging to the inflow boundary by left-clicking the mouse. Do not skip nodes. The polygon must include boundary nodes only, and no internal nodes. End the polygon by pressing <Enter> on the keyboard.
- The desired inflow boundary polygon is shown in Fig.5.3.
- Assign the inflow condition a name and click **Ok**. Here, we use **Inflow** (Fig. 5.4, left).
- Select **Fixed Width** in the **Type of inflow boundary**.
- Enter the time series for the inflow discharge by clicking **Edit**. To use the hydrograph in file hydrograph+12.txt, click **Import** and select the file. The result of these operations is shown in Fig. 5.4, right. Click **Ok** when done.
- If the grid has been rotated, the display can be returned to its original state of zero rotation by clicking on the button located in the main toolbar.

The inflow discharge was entered as a time series table, with time (seconds) in the left column and discharge (m^3/s) in the right column. The discharge is distributed uniformly across the string of nodes defined as the inflow boundary. A synthetic flow velocity field is then generated with velocity vectors that are perpendicular to the string. Due to the artificial nature of this velocity distribution, it is recommended that this boundary should be placed away from the areas of interest where accurate solutions are sought.

To complete the process, map the boundary polygon to the grid by selecting **Grid | Attributes Mapping | Execute...** from the main menu. This is a step that must be carried out after each boundary condition is entered.

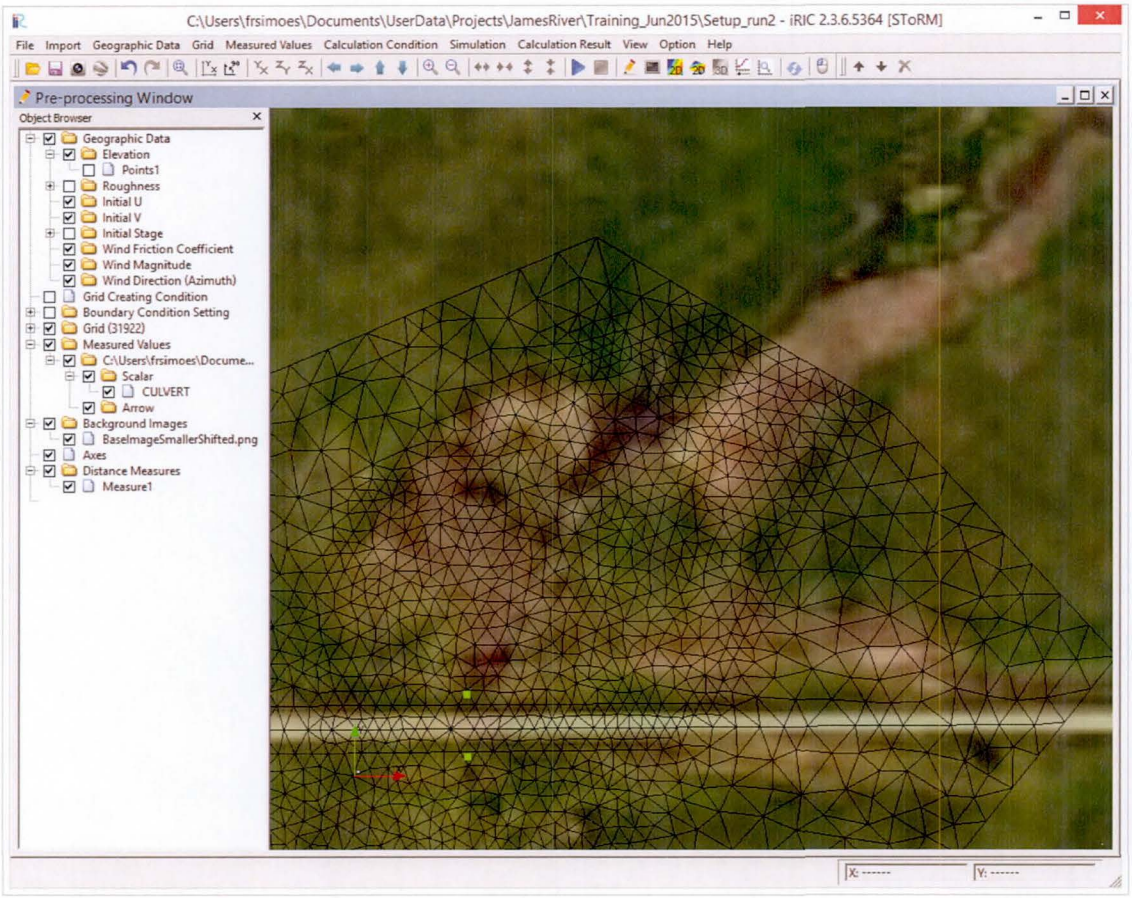

C:\Users\frsimoes\Documents\UserData\Projects\JamesRiver\Training_Jun2015\Setup_run2 - iRIC 2.3.6.5364 [SToRM]
File Import Geographic Data Grid Measured Values Calculation Condition Simulation Calculation Result View Option Help
Pre-processing Window
Object Browser
Geographic Data
Elevation
Points1
Roughness
Initial U
Initial V
Initial Stage
Wind Friction Coefficient
Wind Magnitude
Wind Direction (Azimuth)
Grid Creating Condition
Boundary Condition Setting
Grid (31922)
Measured Values
C:\Users\frsimoes\Docume...
Scalar
CULVERT
Arrow
Background Images
BaseImageSmallerShifted.png
Axes
Distance Measures
Measure1
X: ------
Y: ------

Figure 5.2

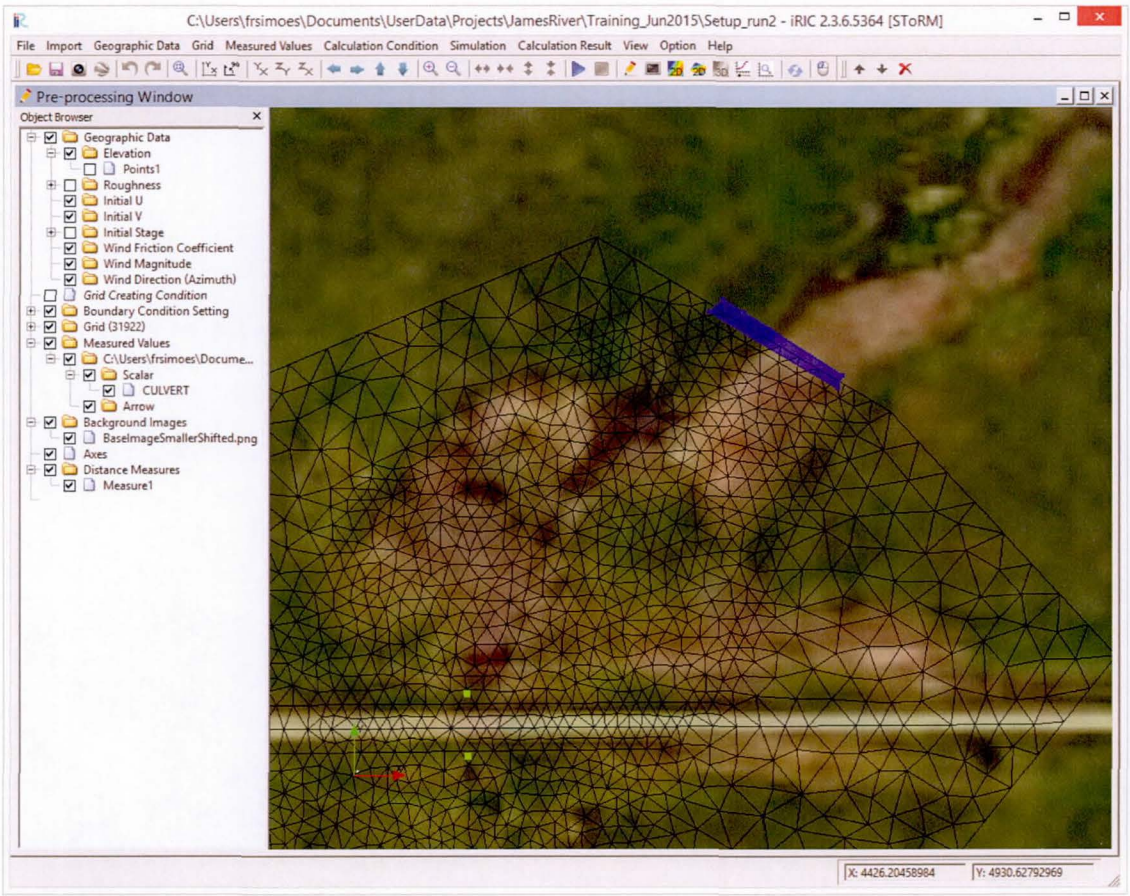


Figure 5.3

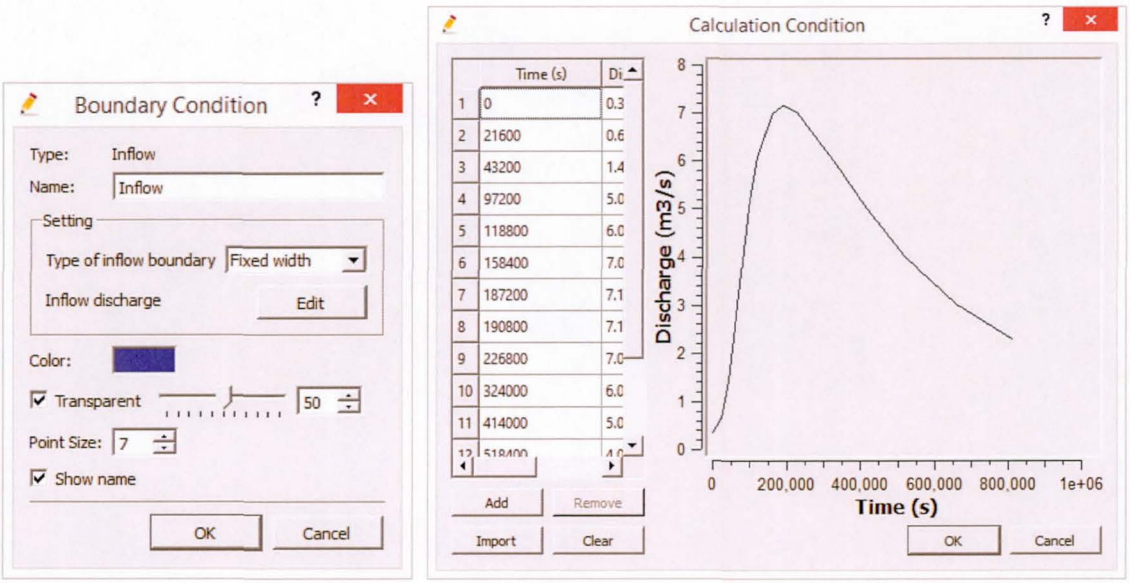


Figure 5.4

The outflow boundary can be thought as the downstream-most cross section of the river reach, and it is where the water surface elevation (stage) is needed for the backwater calculations. An appropriate outflow boundary condition for this exercise can be created by using a very shallow outflow stage. To find an appropriate stage, find the bed elevation at the outflow location using the **Attribute Browser**:

- Zoom to the area of interest, which is the southeast end of the computational grid (Fig. 5.5).
- Right-click on **Node attributes** in the **Object Browser** and select **Show Attribute Browser** from the pop-up menu.
- The **Attribute Browser** window will appear below the **Object Browser** window. It allows to query the values of individual grid nodes. Simply use the mouse to point to the desired node and its corresponding values will be displayed in the **Attribute Browser** window.
- Point to a node at the outflow boundary. The example shown in Fig. 5.5 registers a bed elevation value of 394.19 m. Adding 1 cm to this value yields a value of 394.20 m for the outflow boundary water surface elevation.

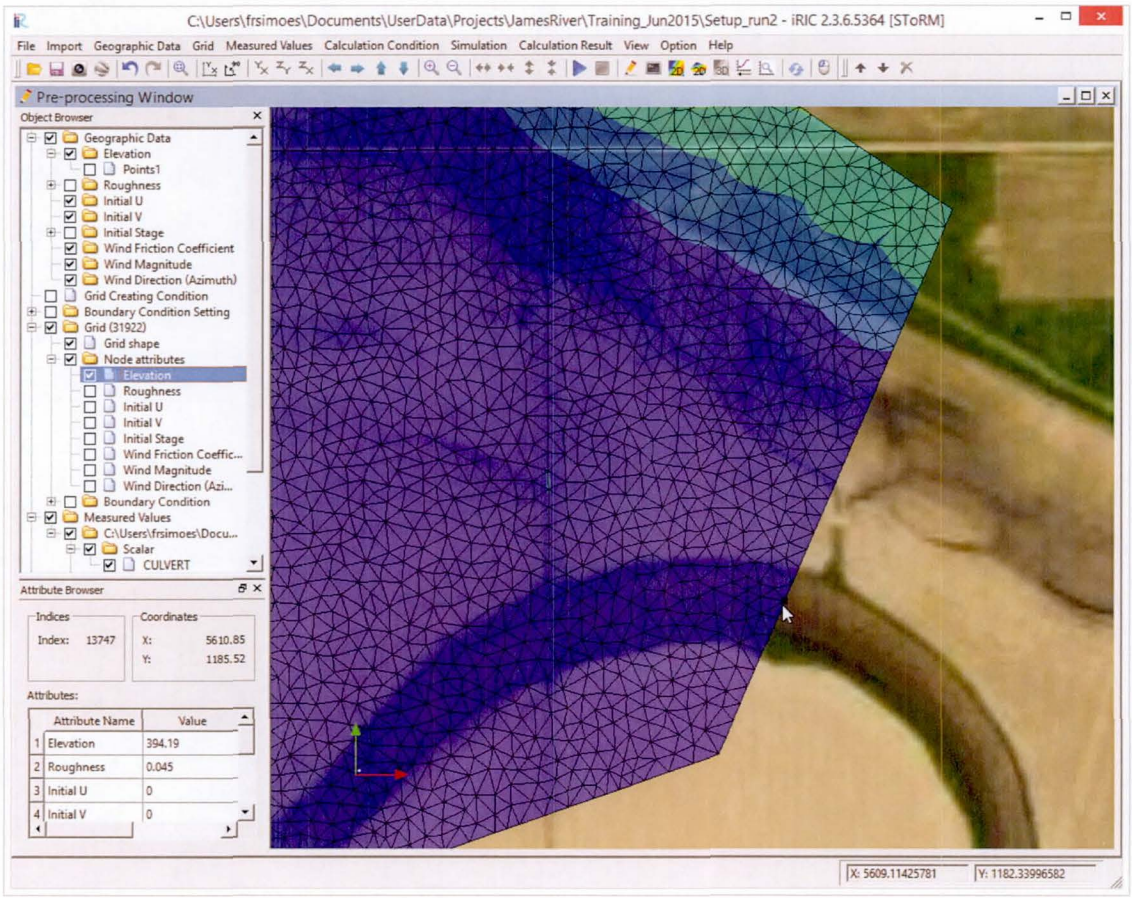


Figure 5.5

The outflow boundary is defined in similar steps as those used to define the inflow boundary:

- Zoom to the area of interest, which is the southeast end of the computational grid (Fig. 5.5).
- Right-click on **Boundary Condition Setting** item in the **Object Browser** and select **Add Outflow** in the pop-up window.

- Define the polygon that encompasses all the grid nodes (triangle vertices) belonging to the outflow boundary by left-clicking the mouse. Do not skip nodes. The polygon must include boundary nodes only, and no internal nodes. End the polygon by pressing <Enter> on the keyboard.
- See Fig. 5.6 for the desired boundary polygon.
- Assign the inflow condition a name and click **Ok**. Here, we use **Outflow** (Fig. 5.7, left).
- Enter the time series for the outflow stage by clicking **Edit**. Click on **Add** to create a data entry row and click on each cell to enter the corresponding value.
- In this example, only one value is entered (394.20 m). Time is not important and the value 0 (zero) can be used here. See Fig. 5.7, right. Click **Ok** when done.
- If the grid has been rotated, the display can be returned to its original state of zero rotation by clicking on the button located in the main toolbar.
- Map the boundary polygon to the grid by selecting **Grid | Attributes Mapping | Execute...** from the main menu.

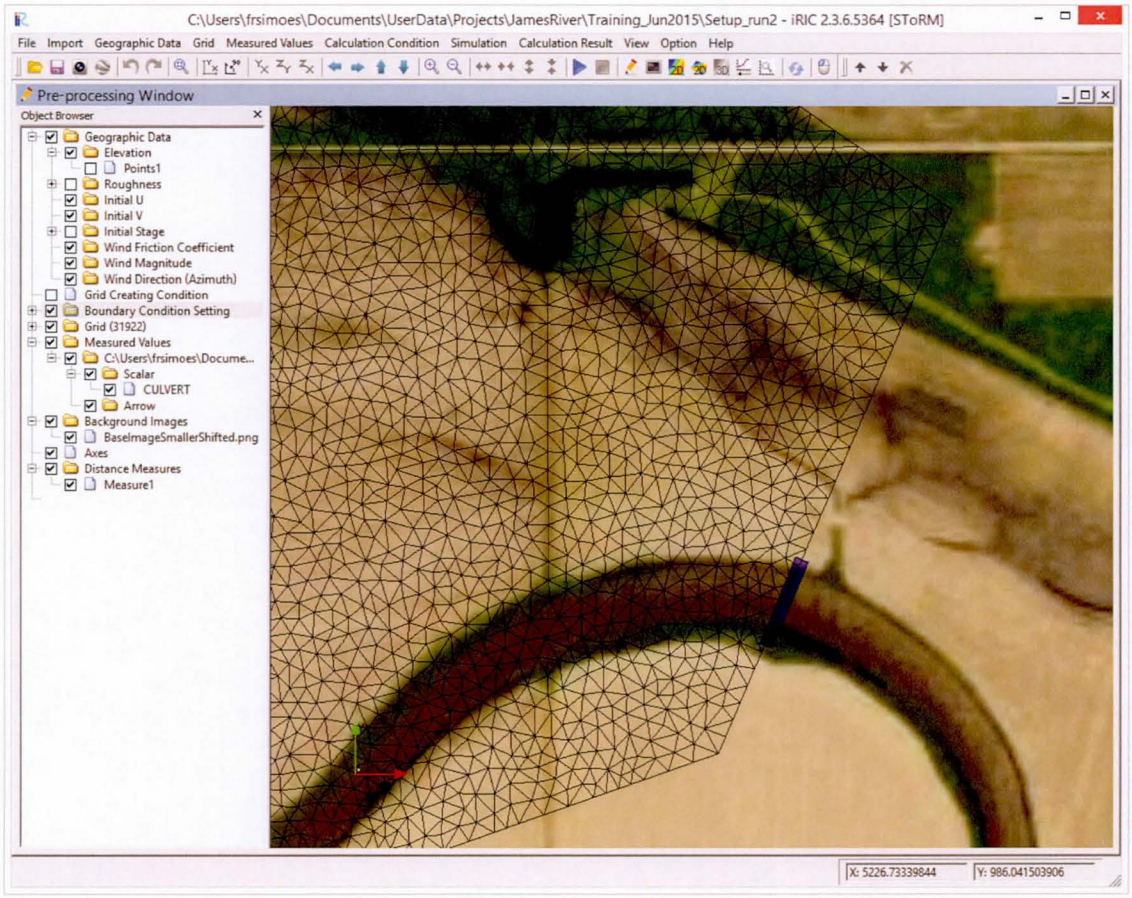


Figure 5.6

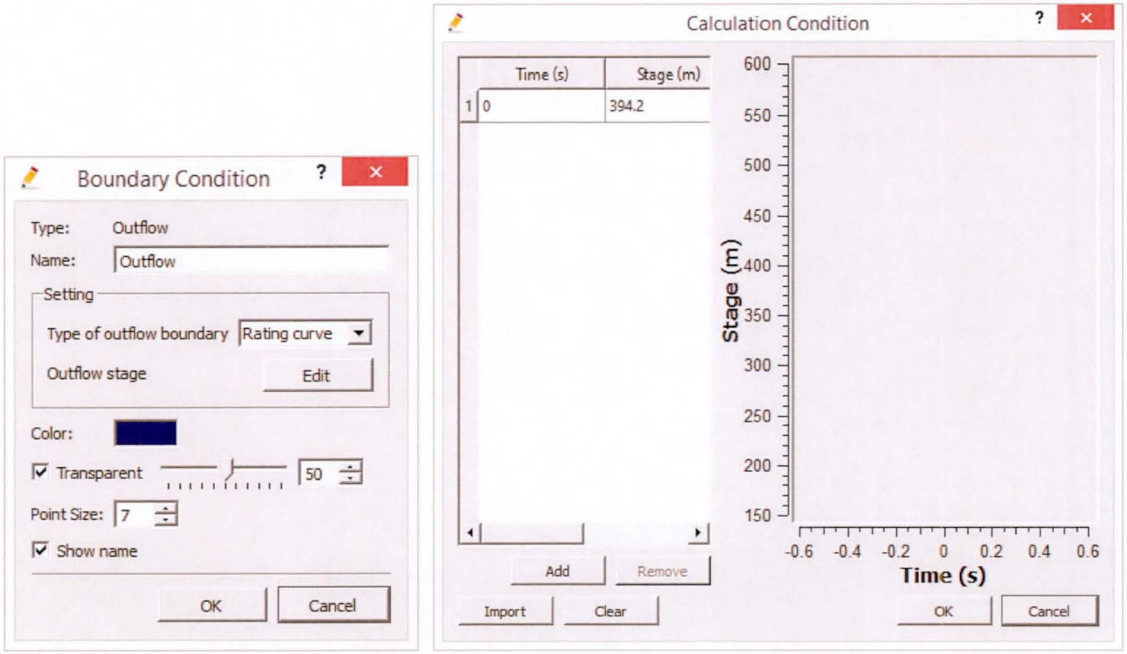


Figure 5.7

5.1. Culverts and Hydraulic Structures

SToRM allows the inclusion of hydraulic structures defined by performance rating curves. A performance rating curve is the depth versus discharge relation for the flow through the structure. The depth is defined at the invert of the structure's head. A structure is defined by a rating curve, by the (pointwise) location of its inlet and outlet, and by the invert of the structure at the inlet. SToRM transfers water from the inflow to the outflow, removing the appropriate amount of water from the triangle where the inflow is located and transferring it to the triangle where the outlet is located. This transfer is instantaneous, but it enforces conservation of mass.

The performance curve must contain only the flow that passes through the structure. In some cases, however, there may be flow passing through the structure as well as above it. This may be the case in overflowing highway culverts, when the culvert is at maximum capacity and there is also flow passing over the roadway above it. In this case, the culvert flow rate is calculated from the performance curve, and the free surface flow over the roadway is calculated normally by SToRM. Therefore, performance curves must not contain the structure's overflow. This is illustrated in Fig. 5.8, where a typical culvert performance curve is shown. Only the inlet and outlet control parts of the performance curve are used in the SToRM input data, and the roadway overtopping contribution must not be included. Similar considerations must be made for other hydraulic structures, such as for submerged gates and drain pipes.

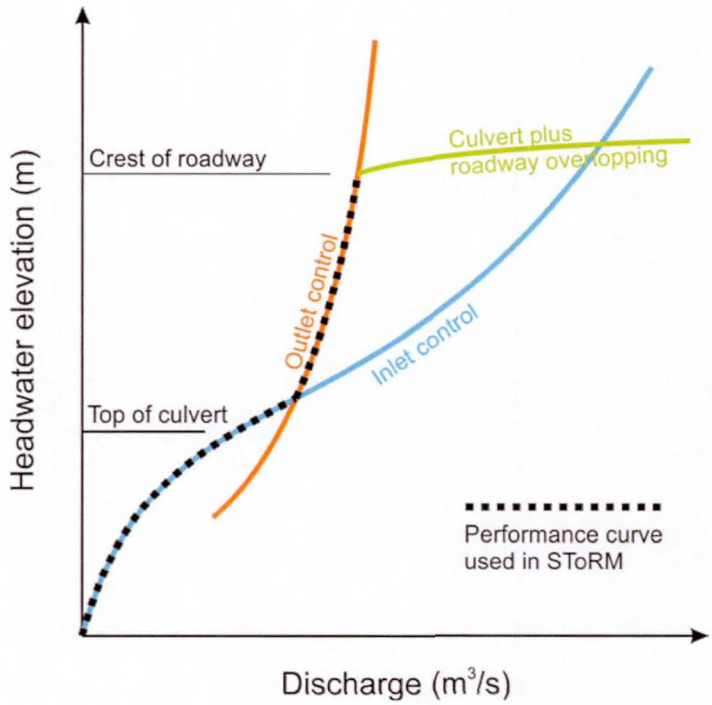


Figure 5.8

The performance curve for the culvert passing under 73rd St. is given in file CulvertShifted.txt. To import it follow these steps:

- Select **Calculation Condition | Setting...** from the main menu bar.
- Click on the **General Options** option in the left window (see Fig. 5.9).
- Select **Yes** in the pull-down menu **Activate culvert computations**.
- Locate the file with the culvert data by clicking on ... and selecting the appropriate file.
- Click on **Save and Close** to complete the operation.

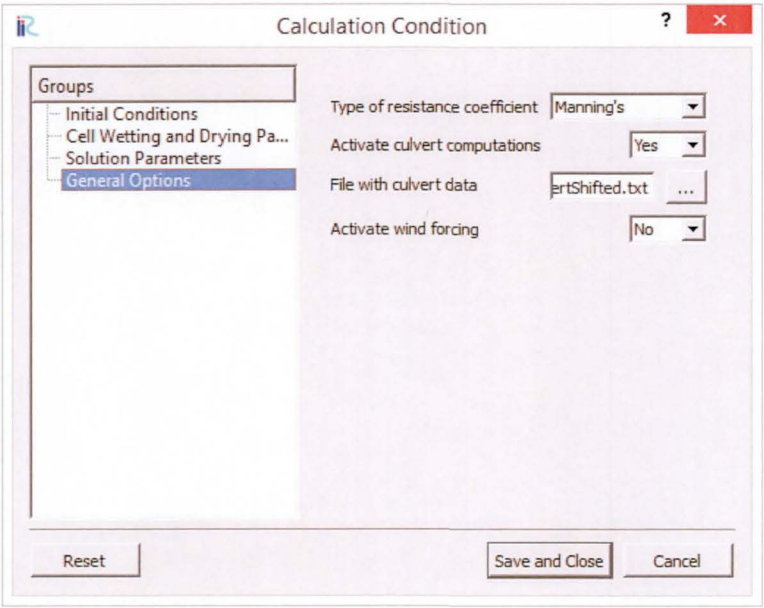
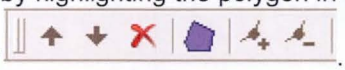


Figure 5.9

5.2. Boundary Roughness

Roughness is set by drawing polygons (sometimes called coverage polygons) over the areas of interest. Follow these steps to set the roughness:

- In the **Object Browser**, under **Geographic Data** right-click on **Roughness** and select **Add** and **Polygon** in the respective pop-up menus.
- Draw a polygon covering the entire computational grid. The polygon is entered by left clicking the mouse on the canvas to enter each vertex of the polygon, in sequence. Finish by double-clicking the last vertex of the polygon (or by pressing <Enter>).
- When asked, enter a value of the roughness of 0.045 (Manning's *n*).
- Map the roughness values to the computational grid by executing the command **Grid | Attributes Mapping | Execute** from the main menu.
- The final polygon may look like what is shown in Fig. 5.10.
- The vertices of the polygon can be edited later by highlighting the polygon in the **Object Browser** window and using the tools in toolbar  .
- Map the boundary polygon to the grid by selecting **Grid | Attributes Mapping | Execute...** from the main menu.

In SToRM, roughness can be represented by Manning's *n* values, by Chézy's *C* coefficients, or by drag coefficients c_f. The default is Manning's *n*, which is used in this tutorial, but it will be shown later how to select the others.

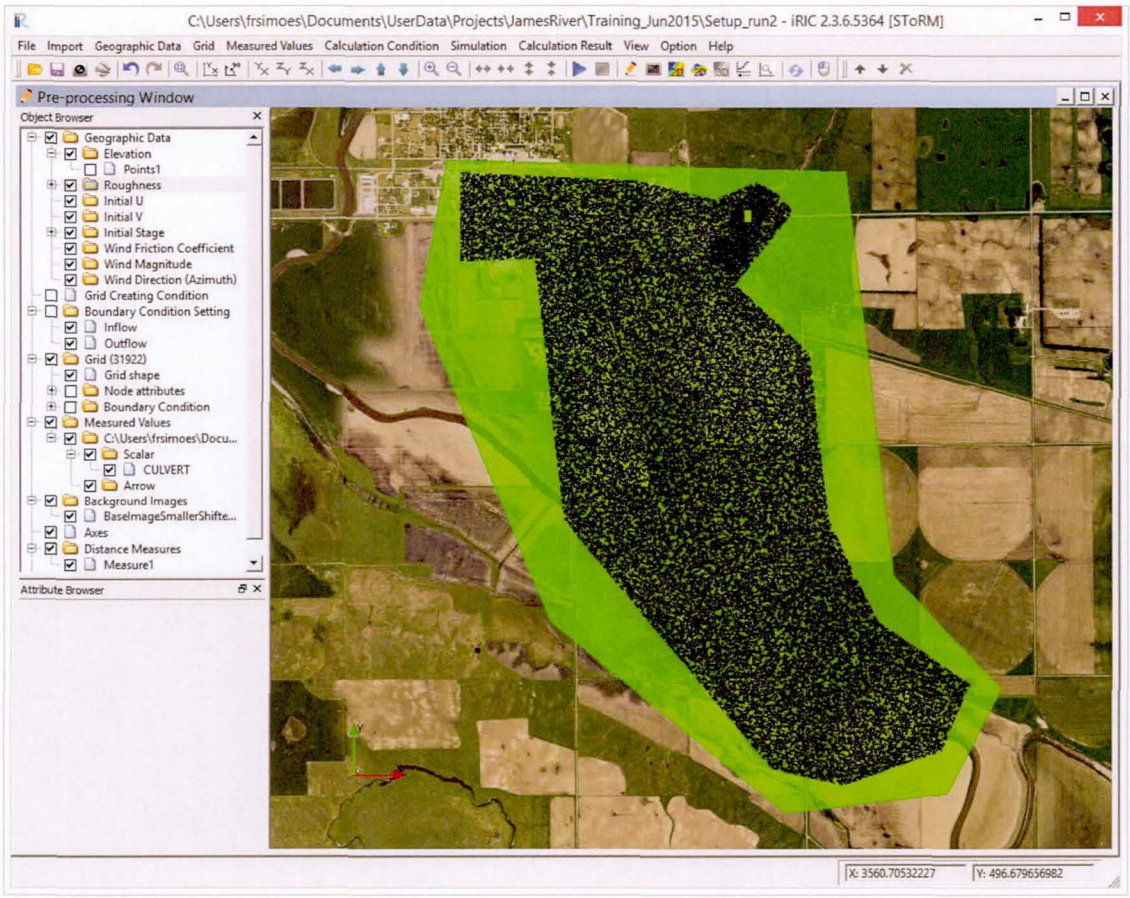


Figure 5.10

5.3. Initial Conditions

The term "initial conditions" reflects the state of the flow at the beginning of the simulation, i.e., at time $t = 0$ s. A simulation run starts with a known flow field, which is a known velocity and water depth throughout, and advances it in time (also called time marching or time stepping by the modeling community). SToRM can be run from a state of completely dry, partially wet and dry, or completely wet bed configurations. In this exercise, we use an initially wet inflow boundary, which helps model stability and allows larger time steps to be used in the calculation of the progression of the solution through time.

One of the ways that can be used to define initial conditions is by the use of coverage polygons, identical to what is done above for roughness data entry. A small region with prescribed water surface elevation can be created with the following steps:

- In the **Object Browser** window, right-click on **Initial Stage** and select **Add... | Polygon** from the ensuing pop-up menus.
- Click on the display canvas to define a polygon at the inflow boundary using Fig. 5.11 as a guideline. Left-click the mouse to create a vertex, double click or press <Enter> to finish the operation.
- When asked, enter a value of 412 m for the stage in the area of the polygon.

- Note that the vertices of the polygon can be edited later using the tools in toolbar .
- Map the boundary polygon to the grid by selecting **Grid | Attributes Mapping | Execute...** from the main menu.

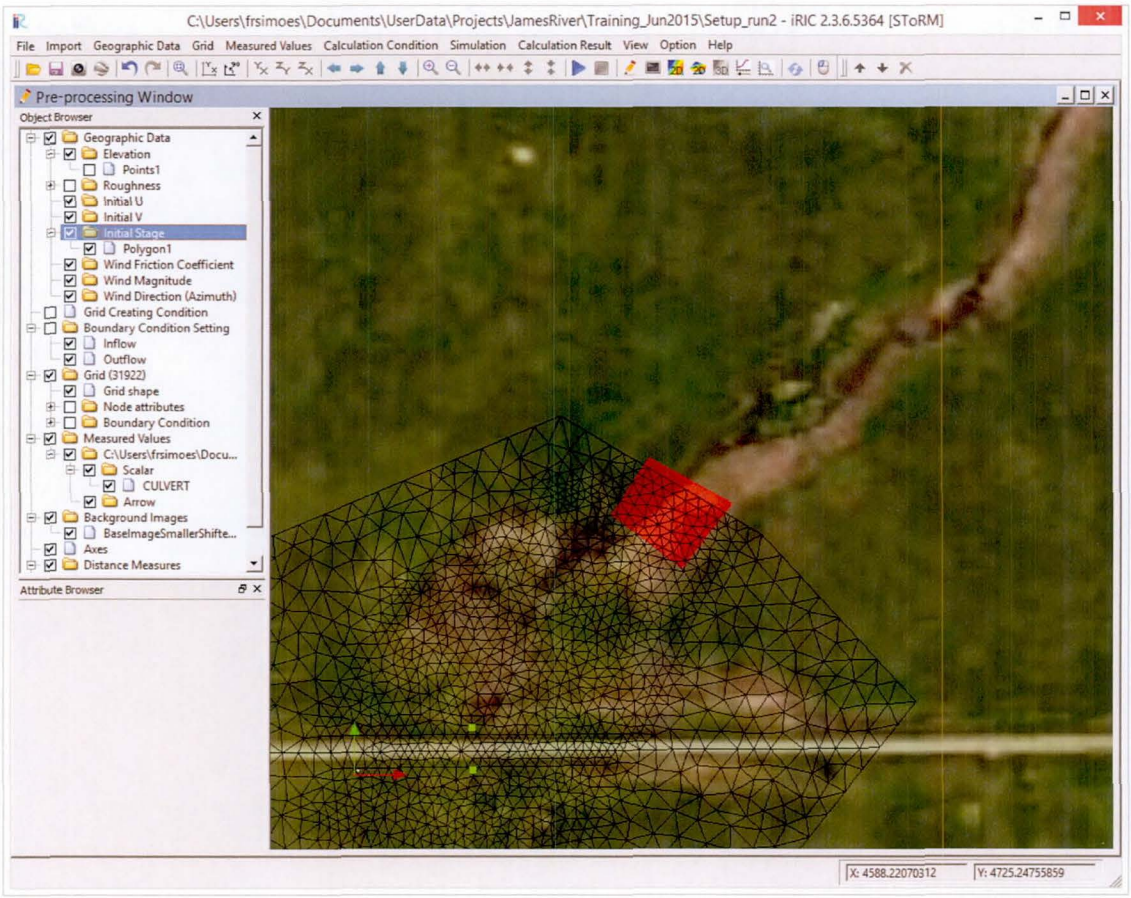


Figure 5.11

6. Running the Model

Like any other model, SToRM requires the definition of additional parameters. Some of these parameters are physical quantities, others are parameters used to set up the numerical methods employed to solve the equations. These must all be defined by the user in a set of entry screens that are going to be covered in this section.

To define the parameters needed for the run, there are a group of entry screens in the **Calculation Condition** window. This window is invoked from the **Calculation Conditions | Settings...** menu from the main pull-down menu bar. The different parameter entry screens are selected by clicking the appropriate text in the left-hand side panel. The first is titled **Initial Conditions** and is shown in Fig. 6.1, left. In this tutorial, the initial conditions were defined using coverage polygons in the previous section, therefore this entry screen must read **From coverage polygons** in the **Set-up initial conditions** pull-down menu.

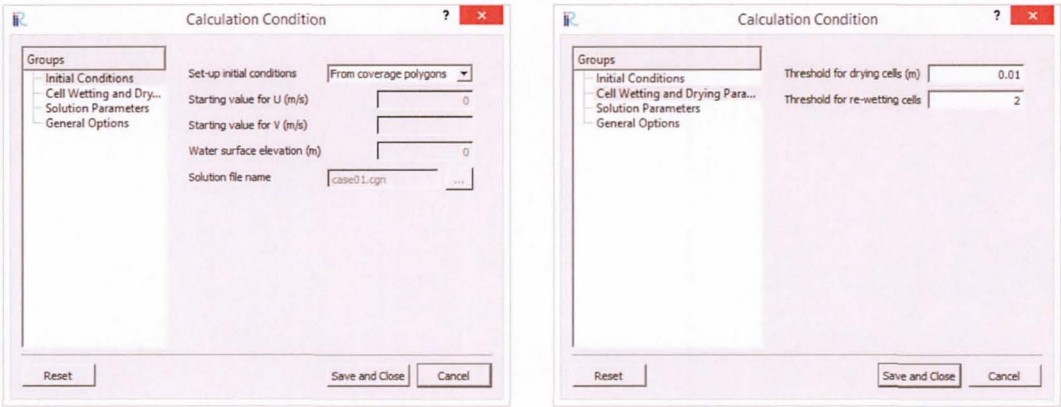


Figure 6.1

The **Wetting and Drying** entry screen shown in Fig. 6.1, right, is used to specify the parameters that control the behavior of the wetting and drying computation algorithm in SToRM. Parameter **Threshold for drying cells** is a value of the water depth (in meters) below which a wet cell becomes dry. In this example it is set to 1 cm. Parameter **Threshold for re-wetting cells** defines a value of the water depth above which a dry cell becomes wet. It is a dimensionless multiplier: this value multiplied by the **Threshold for drying cells** provides the desired value of the water depth. Here, the value of the parameter is 2, meaning that a dry cell is considered wet again when its water depth becomes higher than 2 cm (2 x 0.01). The **Threshold for re-wetting cells** parameter is important because it prevents excessive triggering of wetting and drying at cells at the river margins. This numerically-induced wetting and drying cycle can introduce instabilities in the solution and slow down convergence to steady state. Recommended values are of 2 or higher in the lowest range, and up to 10 or more, depending on the value used for the threshold for drying cells: low drying thresholds require higher re-wetting values.

The **Solution Parameters** entry screen (Fig. 6.2, left) is used to define the parameters used by the time stepping solution algorithms in SToRM. The **Time step size** defines the size of the time step used to advance the solution in time. The time units are seconds. The **Time stepping method** is a pull down menu that selects the time stepping method used in the calculations. There are three methods: **Order 1**, **Order 2**, and **Order 3**. The first method is the most computationally efficient and the last one the most accurate. The choice of time stepping method is a compromise: in a typical application, all methods should arrive to the same solution, but it may be necessary to use smaller time steps when using the **Order 1** method. Experimentation may be needed to choose between each, with the ultimate goal being to reach a solution in the shortest computer run time. Use the values shown in Fig. 6.2. In practice, the **Order 1** or **Order 2** methods should be used, because **Order 3** is computationally too onerous and the law of diminishing returns kicks in.

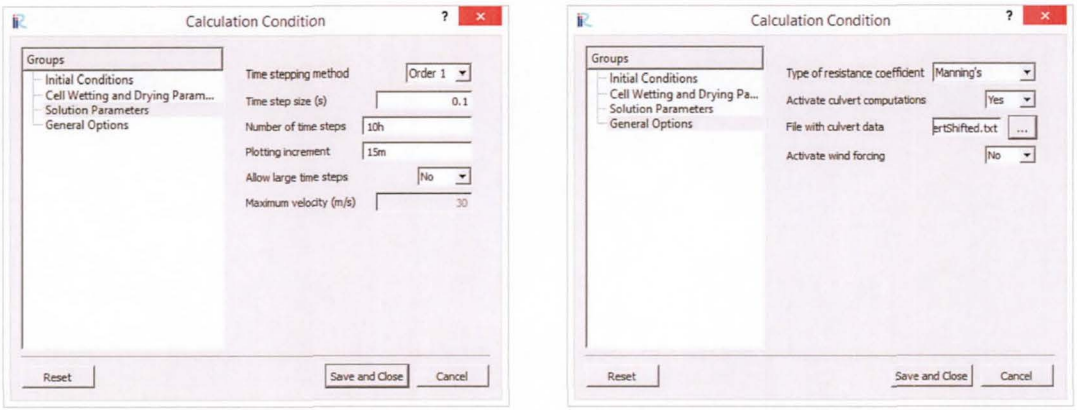


Figure 6.2

A run of 10 hours is sought, with intermediate output of the solution at every 15 minutes. The rules to enter the **Number of time steps** and the **Plotting increment** values are:

- A plain number is interpreted as the number of time steps.
- A number followed by a letter is interpreted as a time period. For example, typing 5s means a run of 5 seconds. The permissible units are s for seconds, m for minutes, h for hours, and d for days. Capital and lower case are both okay, but don't put a space between the number and the letter (1.5D is okay for a day and a half, but 1.5 D is not). Use only one time unit: for example 6.5h is okay for 6 and a half hours, but 6h30m is not.

Finally, the **General Options** entry screen (Fig. 6.2, right) allows to select the type of roughness coefficient from the **Type of resistance coefficient** pull-down menu, using culvert data, and wind forcing (which is not used in thin exercise).

After data preparation, STORM is run by clicking the button on the main window command bar. A model run can be stopped at any time by clicking on the button. The progress of the computation can be followed in a pop-up window (console). The first column contains the iteration number; the second column contains the corresponding time, in seconds; the third column contains the value of a quantity that indicates the relative change of the water depth in each time step, defined as

$$\sqrt{\frac{\sum\left(h_i^{t+\Delta t}-h_i^t\right)^2}{\sum\left(h_i^t\right)^2}}$$

where h_i^t is the water depth in triangle i and at time t, Δt is the time step size, and the summation is carried over all the wet triangles; the fourth column contains the inflow discharge of water (m^3/s); and the fifth column contains the discharge out of the computational domain (m^3/s).

Choosing the appropriate time step for the run is an important task. STORM uses explicit time stepping methods. In practical terms this means that, in order to keep the computations stable, there are constraints to the time step size. If not, the computations "blow up," which is a way to say that the variables become uncontrollably large or small— cause divisions by zero or the numbers become larger than the limits of computer representation — so that the water depth may become negative. The run stops prematurely and its results are meaningless. An indication of these events is the presence of 'Inf' or 'NaN' in the model output window during the run. Usually, in convection-dominated flows, stability is related to flow celerity, size of the time step, and triangle dimension by a dimensionless parameter:

$$C_r = \frac{\Delta t\left(U_m + \sqrt{gh}\right)}{L_T}$$

where Δt is the time step size, U_m is the magnitude of the flow velocity at the center of the triangle (= $(U^2 + V^2)^{1/2}$), h is the water depth at the same point, g is the acceleration due to gravity (9.8 m/s^2), and L_T is a grid-related length scale (for example, the triangle's perimeter). The dimensionless parameter C_r is called the Courant number and is fixed for a certain time stepping method, but usually different for different methods. The above relation translates into the flowing statements:

1. For a triangle of a certain size, larger flow velocity requires smaller time steps.
2. In the same situation, larger depths also require smaller time steps, but this constraint is less severe than the one for flow velocity.
3. For a certain flow field, larger time steps can be used for larger triangles. For smaller triangles the time step size must be reduced accordingly.

This process is succinctly schematized in Fig. 6.3, where the two constraining parameters, flow velocity and grid size, are shown by curves representing the stability of the numerical solution. Considering each parameter individually, the curves represent the maximum stable time step size—in other words, any time step size below that curve results in stable calculations. For example, the maximum allowed time step size increases as triangle size increases, but it decreases as flow velocity grows.

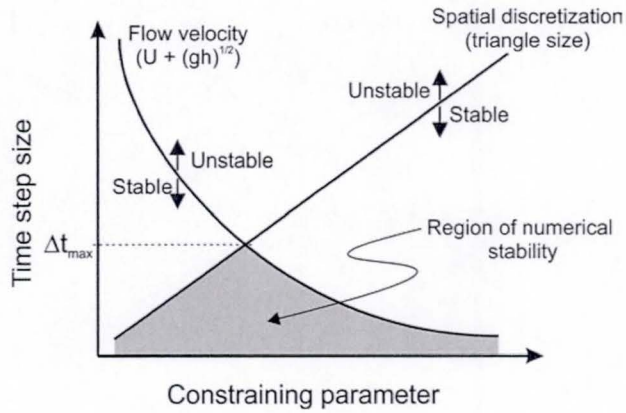


Figure 6.3

A stable time step must lie below both curves. In some cases the limiting factor is velocity magnitude, in others grid size. This means that, for each physical prototype, there is always a limit to the maximum value of the time step that produces a stable numerical solution, which is represented by Δt_{max} in Fig. 6.3. These are important facts to keep in mind when selecting the parameters that go into the **Solution Parameters** entry screen. Unfortunately, things are not that simple. For example, for shallow depths the friction terms become dominant, the flow is no longer convection-dominated, and new restrictions are imposed to the time step to keep the computations stable: small time steps are needed for very shallow depths. Therefore, a trial-and-error cycle is usually necessary to determine the largest time step that retains the stability of the computations. This is case dependent and varies with bed geometry and other boundary conditions.

7. Visualization of Results

Besides data preparation and model's parameter input, a process usually referred to as pre-processing, iRIC also provides means to view the model's numerical solution. This post-processing stage is done in its independent window, titled **Post-processing (2D)** and shown in Fig. 7.1 (water depth). The post-processing canvas of iRIC is invoked clicking on button 2D or by selecting **Calculation Result | Open new 2D Post-Processing Window** from the main menu. The **Post-processing (2D)** window works identically to the **Pre-processing Window**, with its own **Object Browser**, and is used to plot the solution variables.

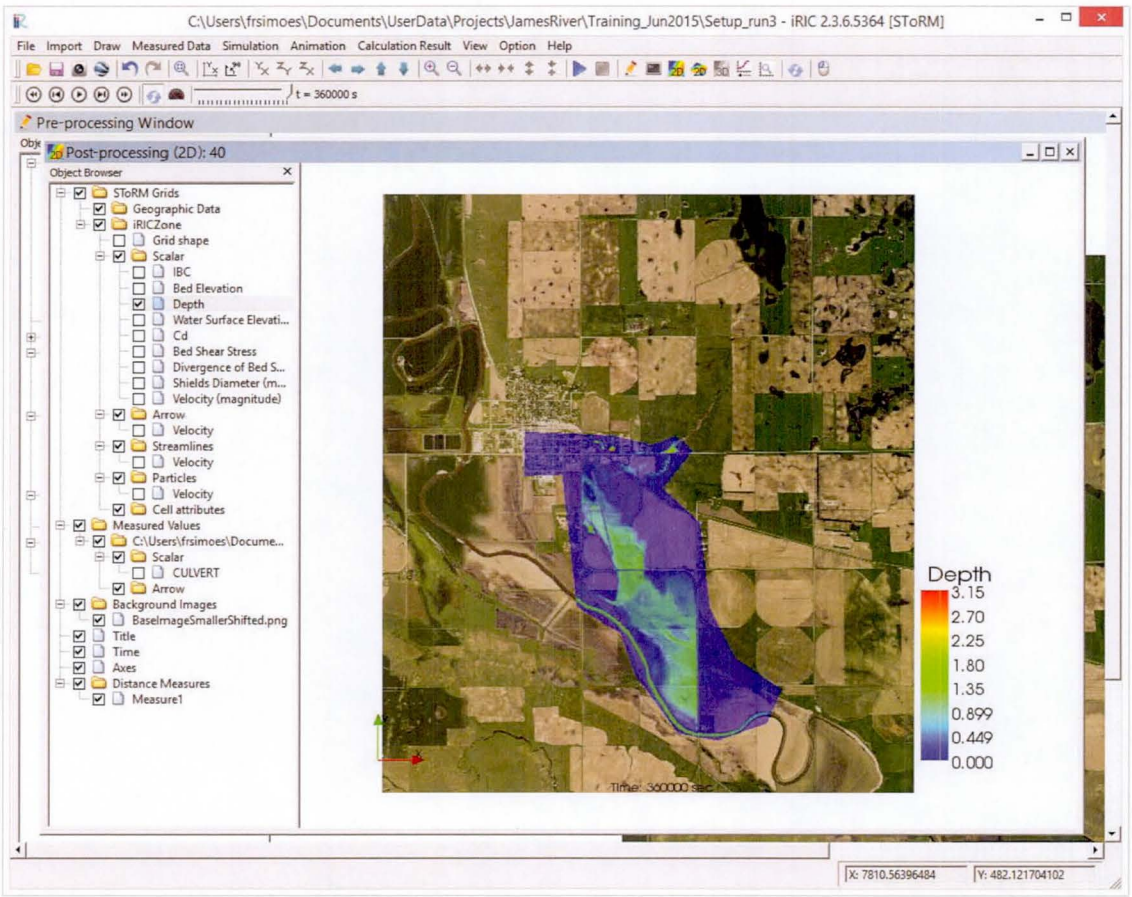


Figure 7.1

Fig. 7.1 shows water depth contours. Many other quantities can be displayed, including velocity vectors, streamlines, and particle paths. The visualization properties of each displayed quantity can be accessed through pop-up menus by right-clicking on the corresponding entity names in the **Object Browser** window:

- The lines and indices of the computational grid can be set from the pop-up menu from **SToRM Grids | iRICZone | Grid Shape** and selecting **Property...** from the pop-up menu.
- Visualization properties of all scalar quantities can be changed by right-clicking on **SToRM Grids | iRICZone | Scalar** and selecting **Property...** This allows creating color maps and change colorization of the displayed scalar field, as well as creating color legends (Fig. 7.2).

- Vector plots can be customized from **SToRM Grids | iRICZone | Arrow**. It is possible to change vector size, color, and sampling.

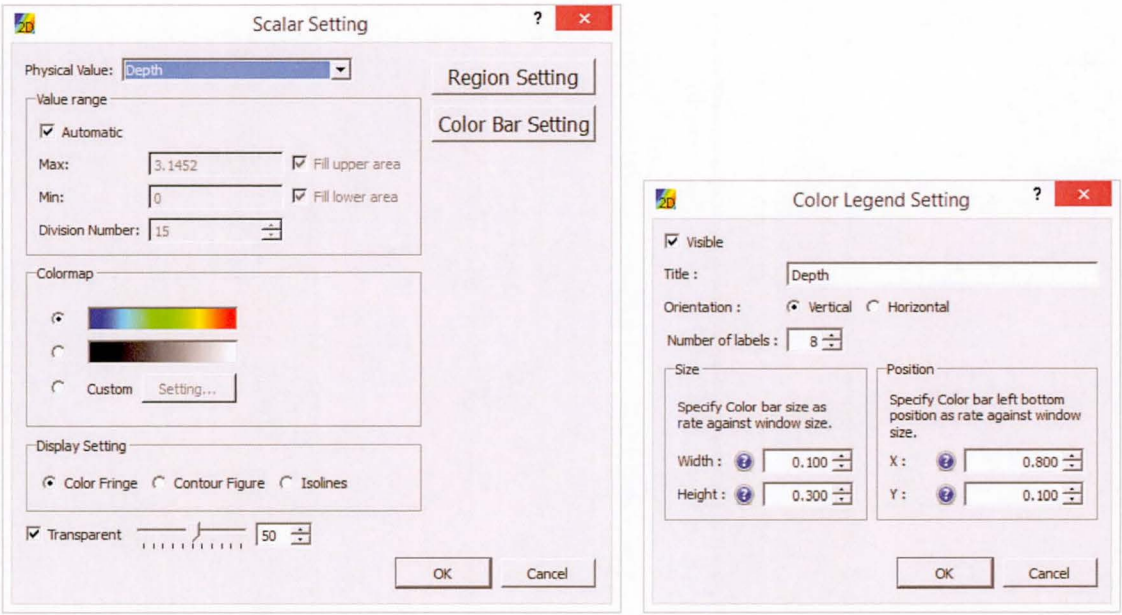


Figure 7.2

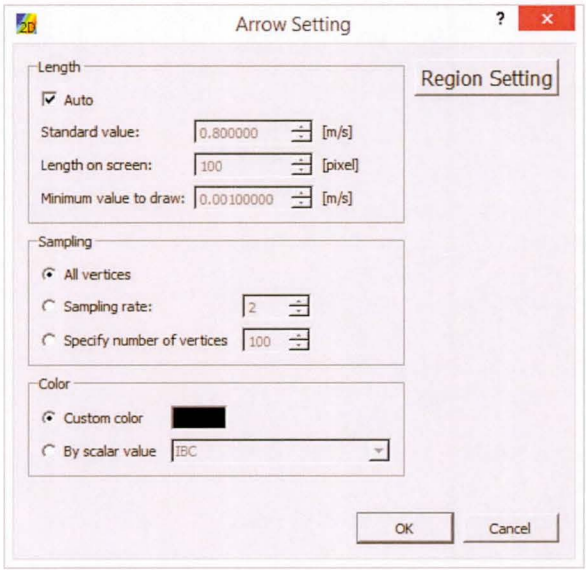


Figure 7.3

There is an important scalar quantity that is different and requires addressing. It is the **SToRM Gids | iRICZone | Scalar | IBC**. This quantity is used like a mask to separate wet from dry cells, and has a value of 1 in wet cells and 0 (zero) in dry cells. Its visualization is shown in Fig. 7.4. It is used to mask the other solution variables when it is desirable to plot their values in wet cells only.

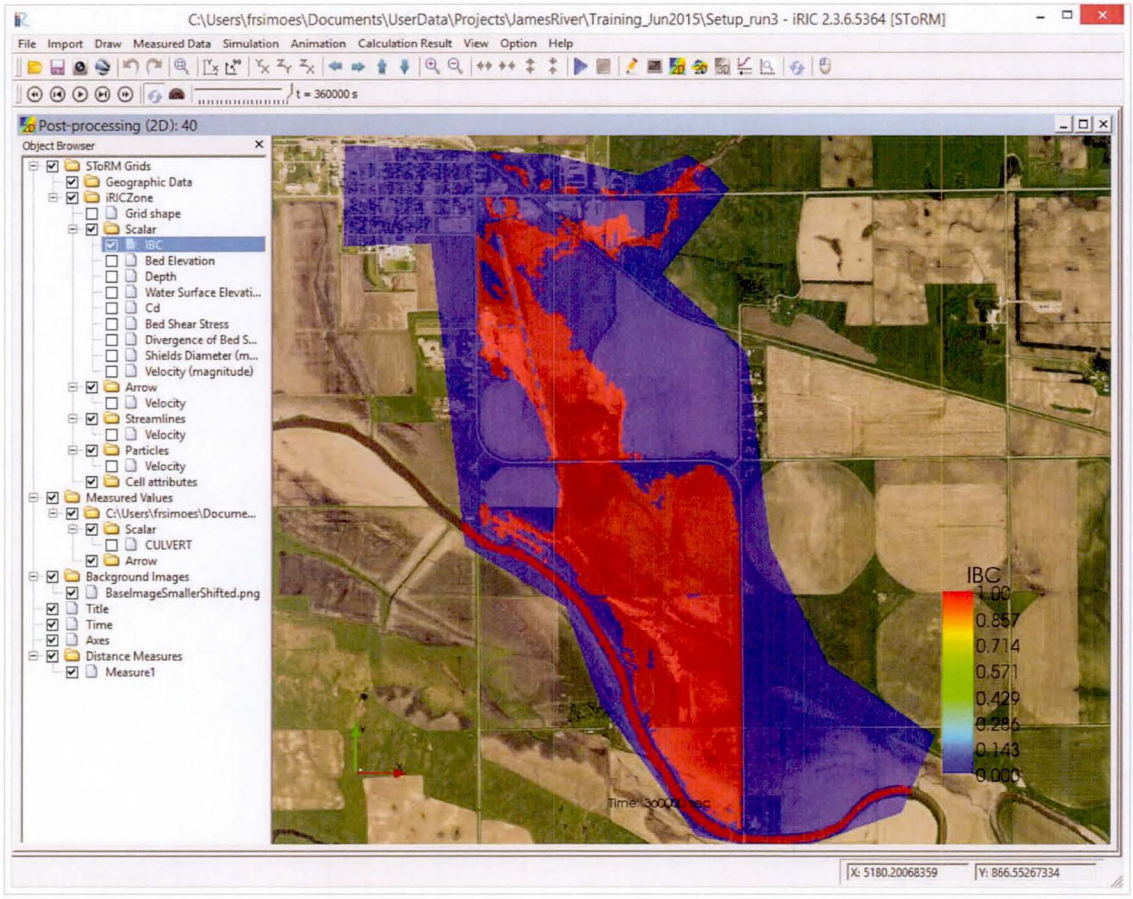


Figure 7.4

To show how to use **IBC** to mask a variable, let's use the water depth. Without masking, the water depth was plotted in Fig. 7.1. The display showed water depth in wet cells, but also in dry cells, where stage is not defined but plots with the same elevation as the bed (zero depth). To restrict the display of water surface elevation to wet cells only:

- Right click the mouse on **Scalar** (henceforth the string **SToRM Gids | iRICZone** will be omitted when there is no source of ambiguity) and select **Property** to bring up the **Scalar Setting** window.
- Click on the **Region Setting** button (top right) and select **Active Region**. Click OK when done.
- Click OK in the **Scalar Setting** window to complete the process.

The resulting plot is shown in Fig. 7.5, where the legend was removed to improve clarity. To remove the legend:

- Right click the mouse on **Scalar** and select **Property** to bring up the **Scalar Setting** window.
- Click on the **Color Bar Setting** button to bring up the **Color Legend Setting** window.
- Uncheck the **Visible** setting, in the top left corner of the window. Press OK when done.
- Click OK in the **Scalar Setting** window to complete the process.

Note that there are other attributes that can be changed in the **Color Legend Setting** window. They are self-explanatory and the user is encouraged to try different settings to see how they impact the look of the plotted variables.

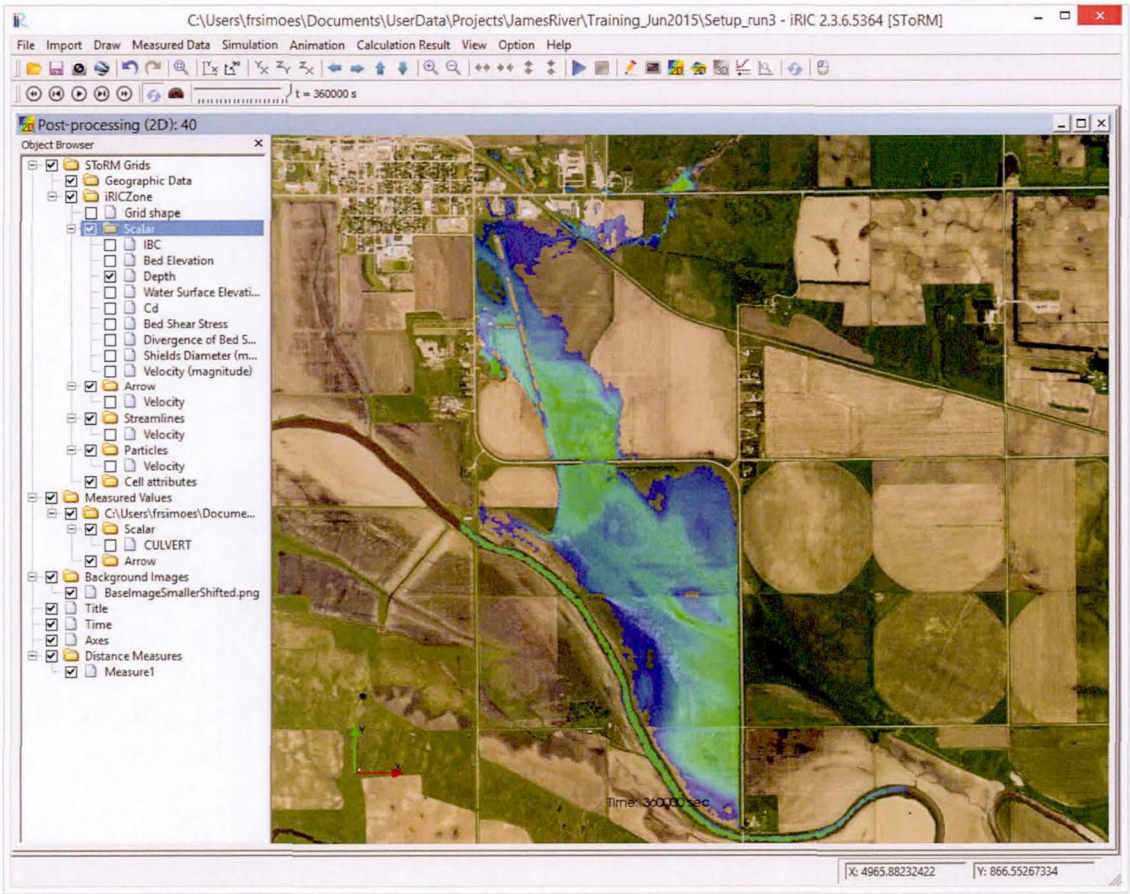


Figure 7.5

iRIC offers the possibility of analyzing the time-dependent data by plotting each output time step requested in the **Calculation Condition** window of Fig. 6.2 (left). Note the time tools toolbar of Fig 7.6, which appears in the previous figures. It indicates that the data viewed corresponds to a time of 360,000 seconds, or 10 hours, and is also printed in the main display canvas in Figs. 7.1 to 7.5. The solution corresponding to the other times steps can be accessed by moving the slider in Fig. 7.6 to the desired position. For example, the magnitude of the velocity for second 135,000 is shown in Fig. 7.7 (only wet cells are colored).

Figure 7.6

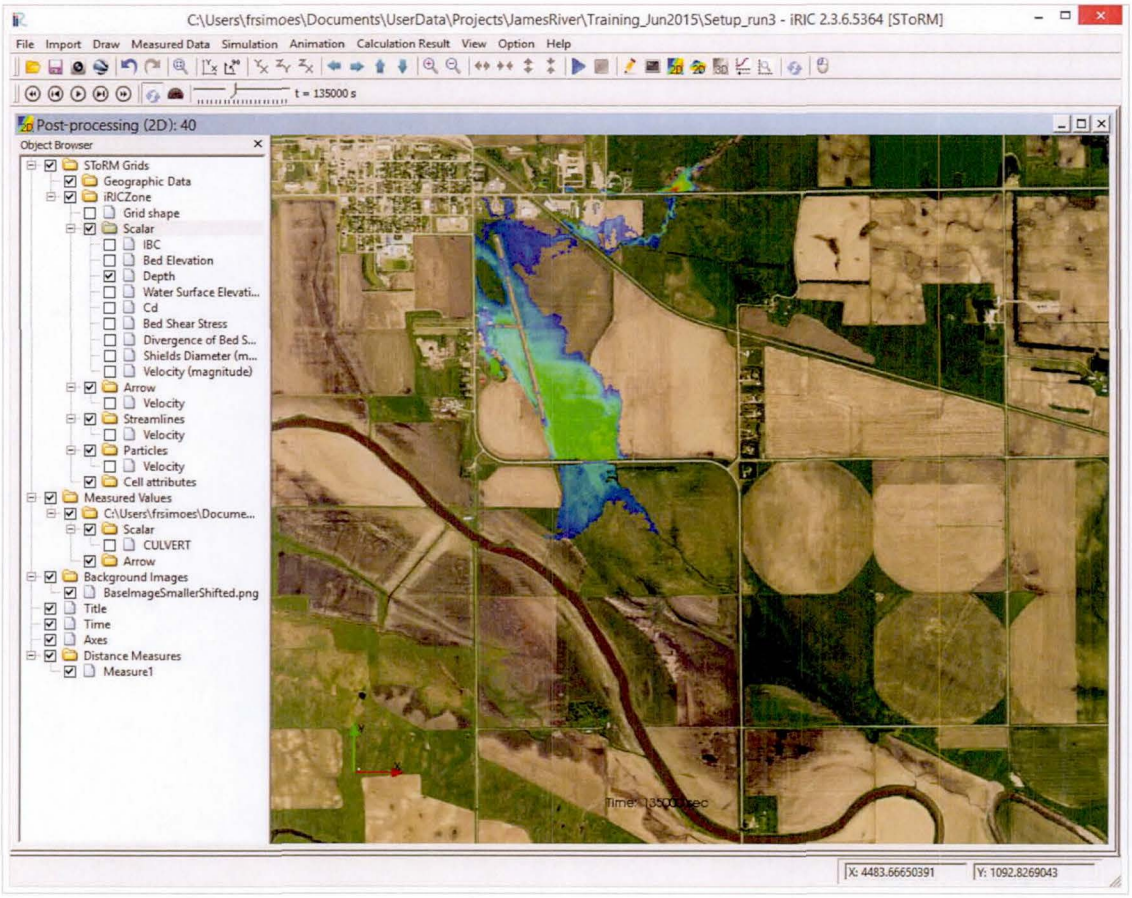


Figure 7.7

The buttons at the left of the toolbar in Fig. 7.6 are used to view the solution in different time steps and can be used to produce a timed animation of the flow solutions. They are easy and intuitive to use, and it is left to the reader to try them out. The buttons are:

- Go to first recorded solution.
- Step back in time by one solution.
- Start animation, starting on currently displayed solution.
- Stop ongoing animation.
- March solution forward in time by one.
- Go to last recorded solution.

The animation speed can be set using button to define the animation interval.