

iRIC Tutorials

Table of Contents

3.0 Getting Started	2
3.1 Exercises	2
3.1.1 Exercise 1	2
3.1.2 Exercise 2	6
3.1.3 Exercise 3	8
3.2 FaSTMECH Tutorials	14
3.2.1 FaSTMECH Tutorial 1 - Basic Flow Application	14
3.2.2 FaSTMECH Tutorial 2 - Variable Roughness and GIC Coverages.....	33
3.2.3 FaSTMECH Tutorial 3 - Introduction to vertical structure and secondary flows.....	41
3.2.4 FaSTMECH Tutorial 4 – 3D model in a natural channel	50
3.2.5 FaSTMECH Tutorial 5 – Evolution of Point Bars During Constant Discharge in Simple Meandering Channel....	55
3.2.6 FaSTMECH Tutorial 6 – Evolution of Point Bars during Time-Dependent Discharge	58
3.2.7 FaSTMECH Tutorial 7 – Habitat Analysis	63
3.2.8 FaSTMECH Tutorial 8 – Wilcock and Kenworthy Two Fraction Sediment-Transport Model	68
3.4 Morpho2D Tutorials	74
3.4.1 Morpho2D Tutorial 1 – An introduction to iRIC: flow modeling using Morpho2D solver; flow, bed evolution and vegetation.	74
3.5 Nays Tutorials.....	96
3.5.1 Nays Tutorial 1 – An introduction to iRIC: flow modeling using the Nays solver; simple meandering channel and morphodynamics.....	96
3.5.2 Nays Tutorial 2 – An introduction to iRIC: flow modeling using the Nays solver; grid generation and flood hydrograph.....	108

3.0 Getting Started

The best way to start using iRIC is to move through the set of exercises presented in Section 3.1, followed by the tutorials in Section 3.2, and by referring to individual topics in this user's guide along the way for further clarification. The exercises provide detail on the essential aspects of building, running, and visualizing results. The tutorials follow specific problems from beginning to end.

- [Exercises](#)
- [Tutorials](#)

3.1 Exercises

The following exercises are intended to give the user a brief introduction to iRIC before getting into more detail in the Tutorials. All the data sets necessary for the exercises can be found in the iRIC directory on the user's computer. The user is encouraged to follow the links or look at other appropriate sections within this user's guide for more detailed information.

1. [Exercise 1](#) - Introduction to importing data.
2. [Exercise 2](#) - Introduction to curvilinear orthogonal grids and pre-processing.
3. [Exercise 3](#) - Introduction to FaSTMECH model and post-processing

3.1.1 Exercise 1

This exercise provides familiarity with the process of importing measured data into iRIC. These data include topography, ancillary scalar and vector data, and images to place in the background of other data types. All the data and image files are in the iRIC Exercises\Exercise 1 directory.

Import topography

1. The raw (measured) 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 through the File Menu by selecting **File->Import->Topography**.
2. In the Select File to Import dialog, choose the embrk.tpo file in the iRIC Exercises\Exercise 1 folder. Enter "1" in the Filter Topography dialog. A console window will appear which provides an indication of progress for the Triangle application which, by default, creates a TIN of the topography file you are importing to iRIC (fig. 3.1.1A). To finish the triangulation simply press any key on the keyboard as indicated in the console window. Open the iRIC Exercises\Exercise 1 folder in your file browser and you will see that Triangle has added a few files (fig. 3.1.1B) including Elevation.1.poly, Elevation.1.node, Elevation.1.ele, and Elevation.1.poly. The first is the Triangle input file created by iRIC and the following three are the output files. See Section 1.4.6.1 in the User's Guide for more information on the formatting of the Triangle input and output files. In the Control Bar the topography can be made visible or not visible by selecting the open box to the left of the **Ancillary Data / Scatter Sets** branch of the Control Bar with the mouse. A check (fig. 3.1.1C) indicates the Scatter Set will be visible and an empty box indicates the Scatter Set is turned off. To add a legend, select the check box for the first **Data Legend / Data Legend**. Then right-click / **Data Legend** and select **Ancillary** in the pop-up menu.
3. On the Control Bar turn the **Ancillary Data / Scatter Sets** object on by activating the check box. On the graphics frame select the refresh button () to center the data in the graphics view, if necessary.
4. Explore the 2D Graphics Viewer – **Plane Viewer**
 - From the menu select the **View** menu. The **View->Plane Viewer** should be checked. If not, select it.
 - Using the left mouse button rotate the TransX, TransY and Dolly "wheels."
 - Select the viewing button () . Zoom in and out by left-clicking the mouse and dragging up and down. Pan the image by holding the Ctrl plus left-click and dragging.

5. View a TIN of the topography by turning on the **Ancillary Data / TINs** tree in the Control Bar.
 6. Explore the 3D Graphics Viewer – **Examiner Viewer**

- From the menu select **View->Examiner Viewer**.
- Using the left-mouse button rotate the RotX, RotY and Dolly "wheels."



- Select the viewing button (

7. Return to Plane Viewer. From the menu select **View->Plane Viewer**.
 8. Save project. From the menu select **File->Save**.

```
c:\Md_swms_iric_01\Generic\trunk\Triangle_Console.exe
 0 - 10 degrees: 227 | 90 - 100 degrees: 599
 10 - 20 degrees: 526 | 100 - 110 degrees: 513
 20 - 30 degrees: 1097 | 110 - 120 degrees: 187
 30 - 40 degrees: 1837 | 120 - 130 degrees: 104
 40 - 50 degrees: 2411 | 130 - 140 degrees: 60
 50 - 60 degrees: 2585 | 140 - 150 degrees: 39
 60 - 70 degrees: 2308 | 150 - 160 degrees: 39
 70 - 80 degrees: 2106 | 160 - 170 degrees: 48
 80 - 90 degrees: 1673 | 170 - 180 degrees: 42

Memory allocation statistics:

Maximum number of vertices: 2748
Maximum number of triangles: 5492
Maximum number of subsegments: 25
Approximate heap memory use (bytes): 308532

Algorithmic statistics:

Number of incircle tests: 20077
Number of 2D orientation tests: 29238

Press any key to exit...
```

Figure 3.1.1A Triangle output.

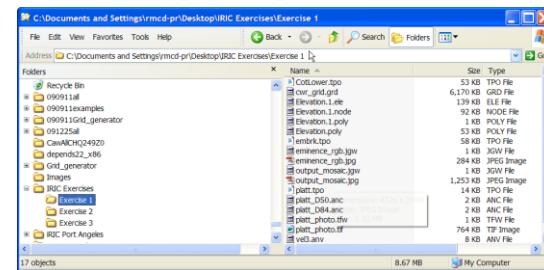


Figure 3.1.1B Exercise 1 directory showing Elevation_1.* files added by Triangle.

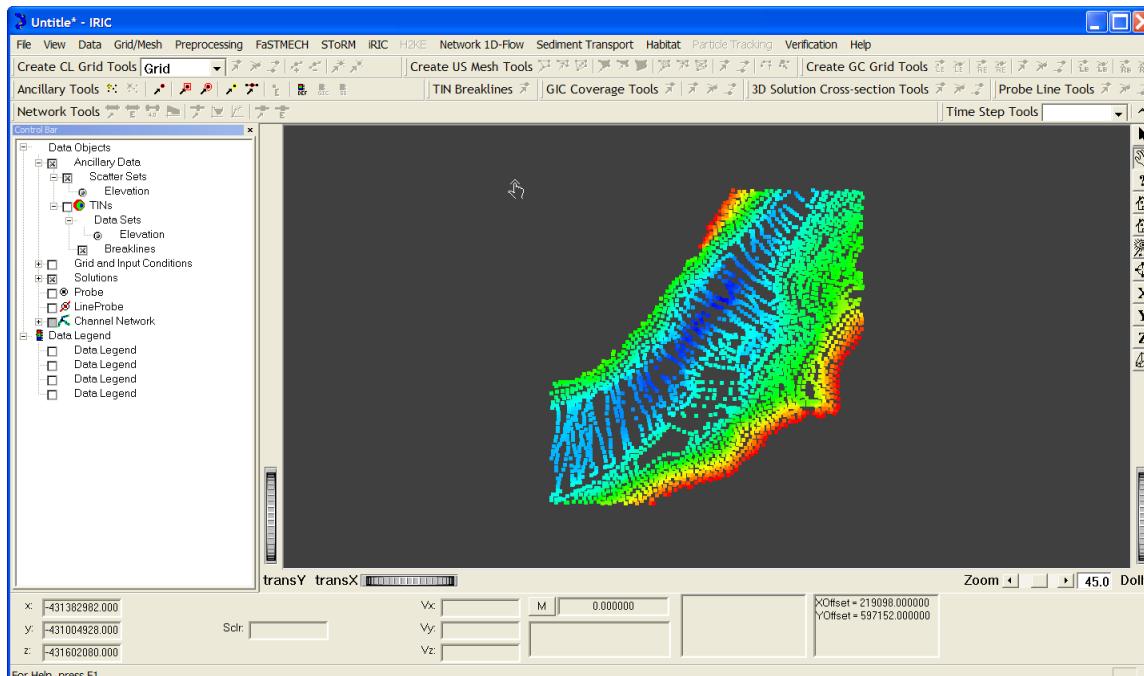


Figure 3.1.1C. View Topography.

Scatter Set Attributes

1. To change the graphic attributes of the scatter set locate the **Ancillary / Scatter Sets** branch of the Control Bar and double click on the **Scatter Sets**.
 - Turn TIN off.
 - Experiment with the attributes (See Section 2.4 Scatter Attributes in the User's Guide).

Import ancillary data.

1. Create a new project.
 - From the menu select **File->New**.
 - Import the platt.tpo topography file using the default Filter and Tolerance Values.
 - Turn on the **Ancillary Data / Scatter Sets** tree and refresh the graphics, if necessary.
2. Import an ancillary data set (for example, grain size) into the project.
 - From the menu, select **File->Import->Ancillary Data->Scalar** to open the Import Ancillary Data dialog.
 - Open the data file by clicking the browse button (), which will bring up the File Open dialog. Select platt_D50.anc. You will see the path to the platt_D50.anc file in the “File to Import” field.
 - Choose the **Grainsize_D50** data type by double-clicking **Sediment Data Types / Grainsize_D50** in the dialog’s data tree. This will add the Grainsize_D50 text to the “Data Type” field of the dialog.
 - Once both a “Data Type” and “File To Import” have been determined, the OK button should be enabled. Import the data by clicking the OK button.
 - Open the platt_D84 file as in the steps above. In the Ancillary Data dialog, notice that now the Grainsize_D50 is selected and cannot be added to the “Data Type” field. Note that a check is place in the checkbox only after the data type has been entered into the project.
3. Toggle between the different ancillary data sets by choosing the option button in the Control Bar of the data set that you want to view
4. Save the file.

Import Images

1. Create another new project.
 - From the menu, select **File->New**.
2. Import the Cotlower.tpo topography file. Remember to turn on the **Ancillary Data / Scatter Sets** and refresh the graphic, if necessary.
3. Import an image to place in the background of the data
 - Select **File->Import->Ancillary Data->Image** from the menu. In the File Open dialog select the output_mosaic.jpg file.
 - Turn on the **Image Sets** in the Control Bar to display the image.
4. Save File.

Import measured velocity data

1. Import the measured velocity data file.
 - Using the same project select **File->Import->Ancillary Data->Vector** from the menu.
 - Locate the vel3.anv file to open it.
 - Turn off the **Ancillary Data / Scatter Sets / Elevation**.
 - Turn on the **Ancillary Data / Vector Sets** to view the data.

Scatter Vector Attributes

1. To change the graphic attributes of the scatter set locate the **Ancillary Data / Vector Sets** branch of the Control bar and double click on the **Vector Sets**.
 - Experiment with the attributes (See Section 2.5 Scatter Vector Attributes in the User's Guide). For example, set the Vector Scale field to 30 and then select the Update Scale button.
 - Save File.

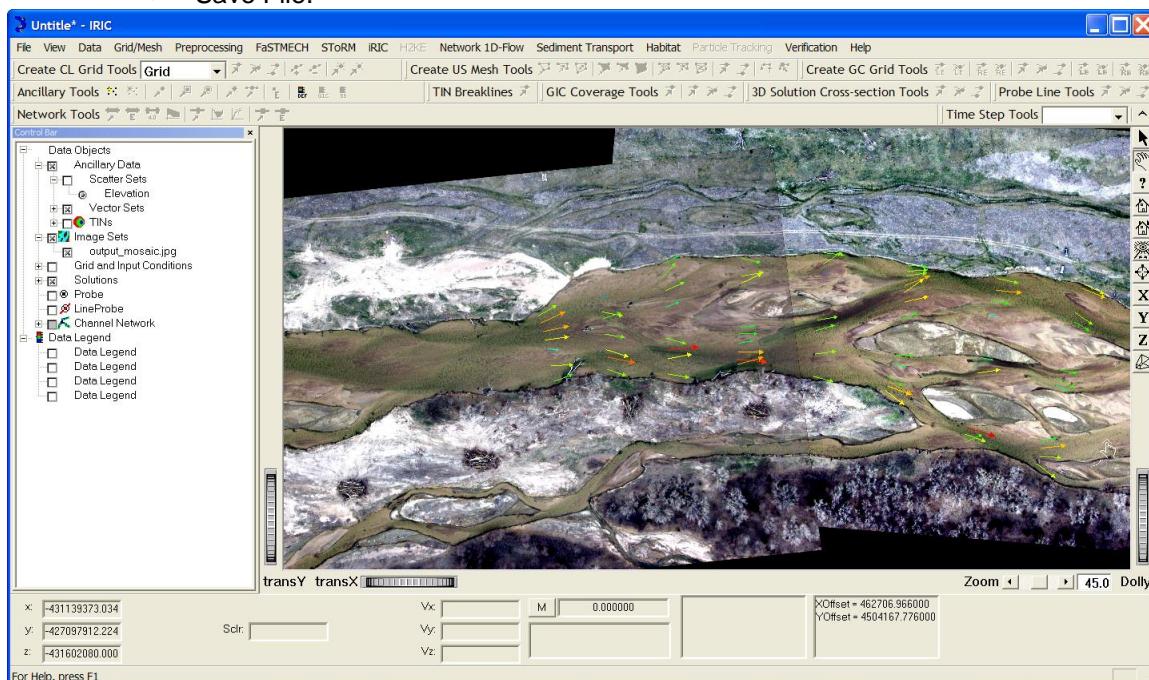


Figure 3.1.1D Measured velocity represented as vectors against a geo-referenced image.

3.1.2 Exercise 2

This exercise provides familiarity with the process of creating curvilinear orthogonal grids by using iRIC. This exercise assumes Exercise 1 has been completed.

Open Topography

1. Create a new project with **File->New**.
2. Import the blind.tpo topography file.
 - View the data in either the Scatter object, TIN object, or both. Remember, it may be necessary to hit the refresh button ().

Create Grid

The following steps are required to create a grid. First define a centerline following the channel from upstream to downstream and second define the grid by specifying the number of streamwise and cross-stream points and the width of the grid.

Centerline

- Turn on the **Grid and Input Conditions / Create Grid** object in the Control Bar. (i.e., Turn on the Grid and Input Condition and then Turn on Create Grid)
 - The Create Centerline/Grid toolbar is now accessible. Go to [Section 1.4.1 Create Centerline/Grid Toolbar](#) in the User's Guide and view the demo.
- Draw a centerline by selecting the Create Centerline tool (). The centerline must be drawn in the direction of flow. In this case, flow is from the upper left to lower right. Click the left mouse button to define centerline points and select Enter from the Keyboard when finished drawing.

Build the grid

- Select **Preprocessing->Set 2D Curvilinear Grid Parameters** from the menu. In the dialog, enter the grid dimensions and the grid width. Try 101 for the number of points in the streamwise, 51 points in the cross-stream points, 400 for the width and then press the Apply Button. The Apply button allows the changes to be seen without closing the dialog. Use the mouse to move the Tension slider and notice the change in the centerline (if there is little curvature in the centerline it could be subtle).
- Save the project.

Map Elevations

1. Map elevations to the grid by using the [Map w/TIN](#) method.
 - The Map w/TIN method works on the currently selected input condition and the currently selected Ancillary Data set.
 - Make sure that **Elevation** is the currently selected **Ancillary Data / Scatter Set**.
 - Select **Preprocessing->Set Current Input Condition->Map w/TIN** from the menu. Every time the elevation is mapped to the grid, a roughness input condition also is created by default. Turn the **Grid and Input Conditions / GIC Scalar Set** object on in the Control Bar to visualize the mapped elevations.
2. Map Elevations to the Grid with [Map w/Template](#) method.

- Select the **Preprocessing->Set Current Input Condition->Map w/Template** from the menu. The curvilinear template dialog will appear.
 - Select Elevation in the Ancillary Data Set drop down menu.
 - Choose the template dimensions (try a length of 100 and a width of 10). Leave the number of template expansions at the default value.
 - Select OK and when finished, view the new topography mapping.
 - Try different template dimensions and notice the difference in the mapping.
3. Save the project.

Edit Data to Improve Mapping

1. Add data points to the upstream **river right** bank to fix the embayment created with the template mapping.
 - Explore Section [1.4.2 Ancillary Toolbar](#) in the User's Guide to view the different options for editing ancillary data.
 - Add topography data such that the right bank is well defined in the mapped numerical grid. It helps to turn off **GIC Scalar Sets** to see where to add points.
 - Save File.

Repeat with New Data

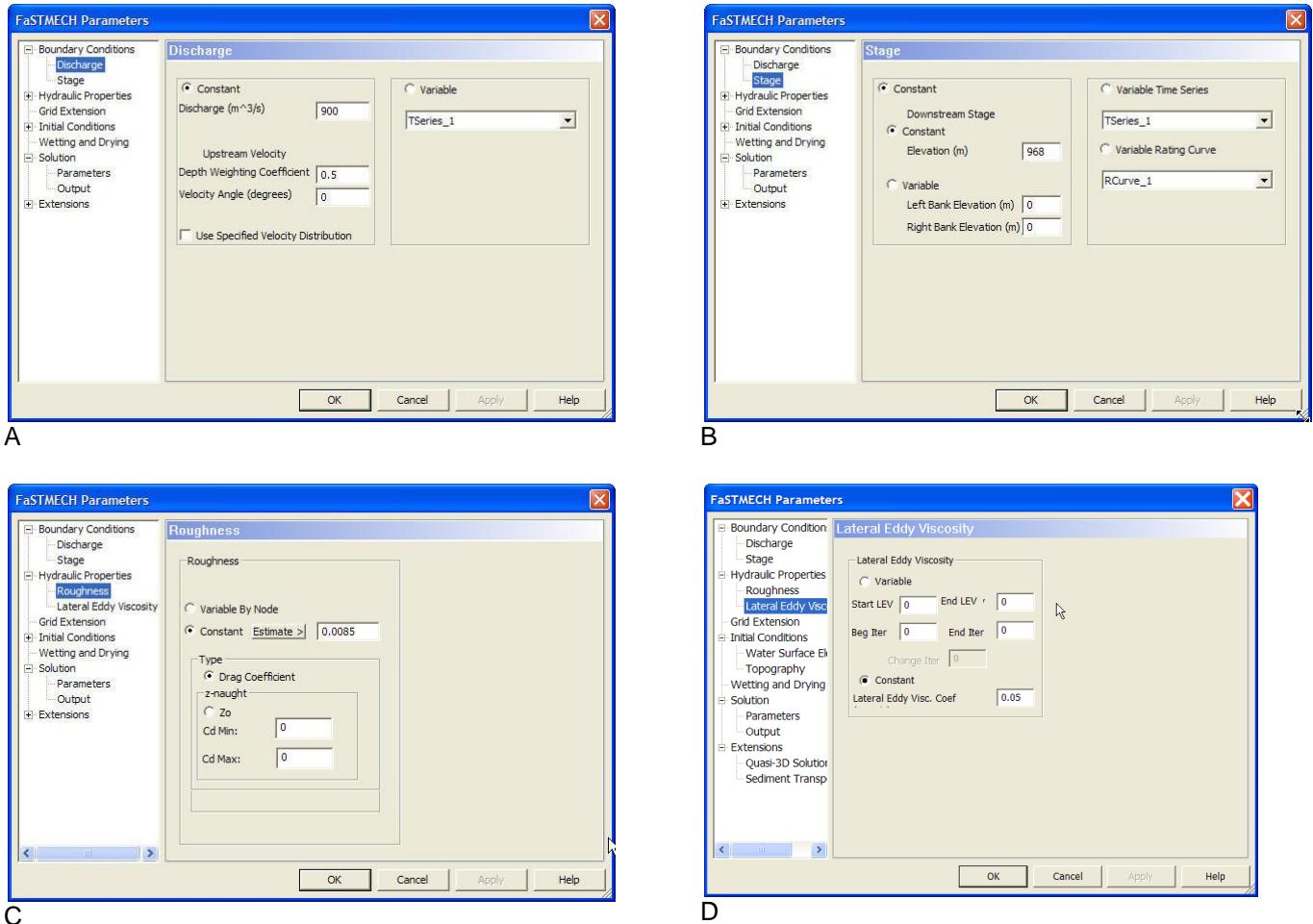
1. Repeat the previous steps with the embrk.tpo topography file, you can also import the eminence_rgb.jpg image as background..
2. Try a few other data sets.
 - bend2k41deep.tpo – Simulated/synthetic meandering river channel
 - Suzy.tpo and suzy_island_rgb.jpg – Channel with island on the Snake River, Idaho

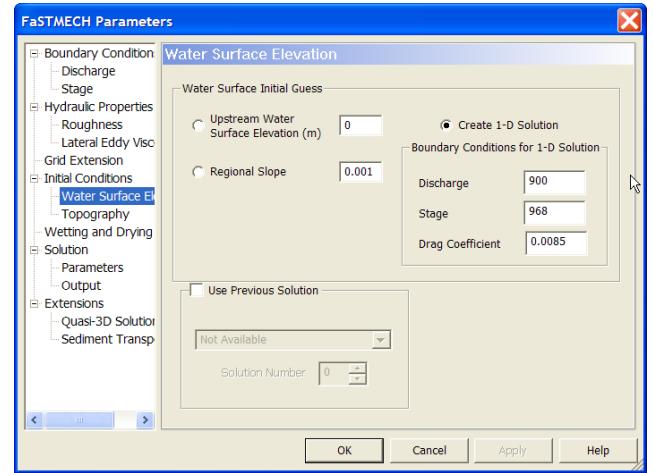
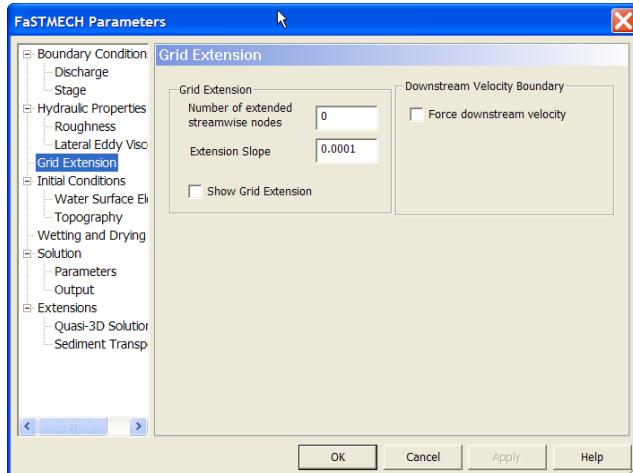
3.1.3 Exercise 3

Open existing Project

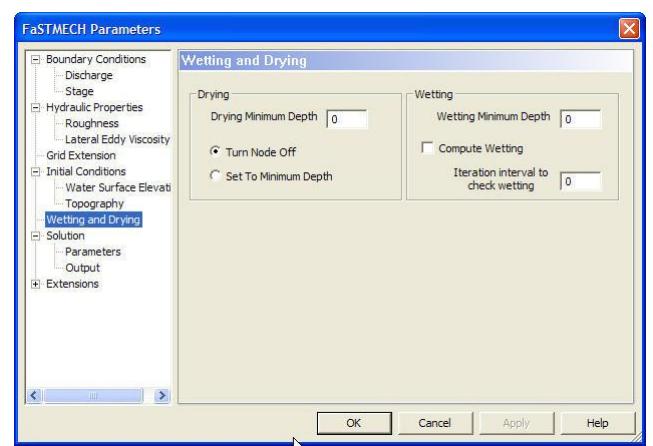
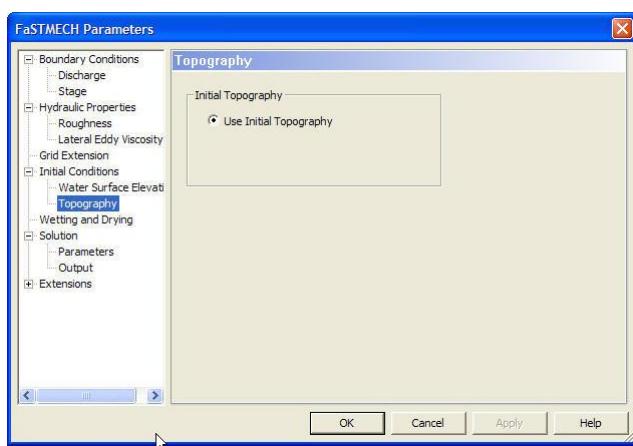
1. From the menu select **File -> Open** and then select the blind.riv project file in the Exercise 3 folder.
2. Create a simulation to run
 - From the menu, select **FaSTMECH->New Simulation**. Enter a name for the simulation (for example, b1). The interface will take care of the file extension. Note that all simulations have the extension .cgn. Notice that the b1.cgn simulation has been added to the **Solutions / Simulations** object in the Control Bar.
3. Create the model parameter input file.
 - From the menu, select **FaSTMECH->Edit Input File**. In the Generate Model Input File dialog enter the parameters shown in figure 3.1.3 A-J, but note that K and L, which parameterize the quasi-3d solution and the sediment transport solution are not required for this exercise.
4. From the menu, select **FaSTMECH->Run**.
5. Look at the Solutions | 2D Solution branch of the data tree in the Control Bar.

For more information of FaSTMECH model parameters see the subheading Create Input File in Section 1.5.5 FaSTMECH Menu.



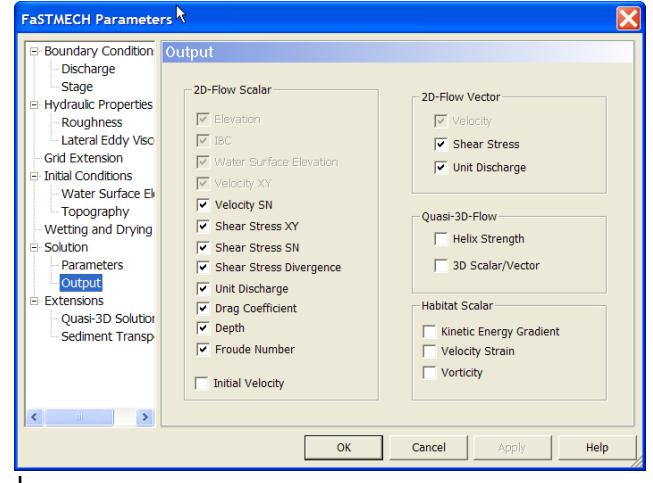
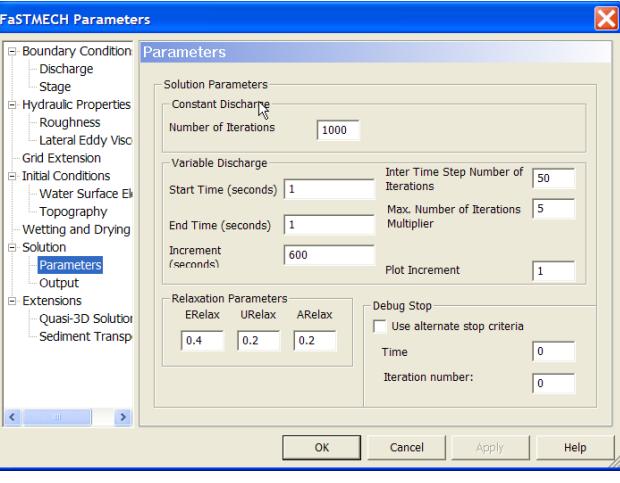


F Note the Create 1-D Solution option is selected



G

H



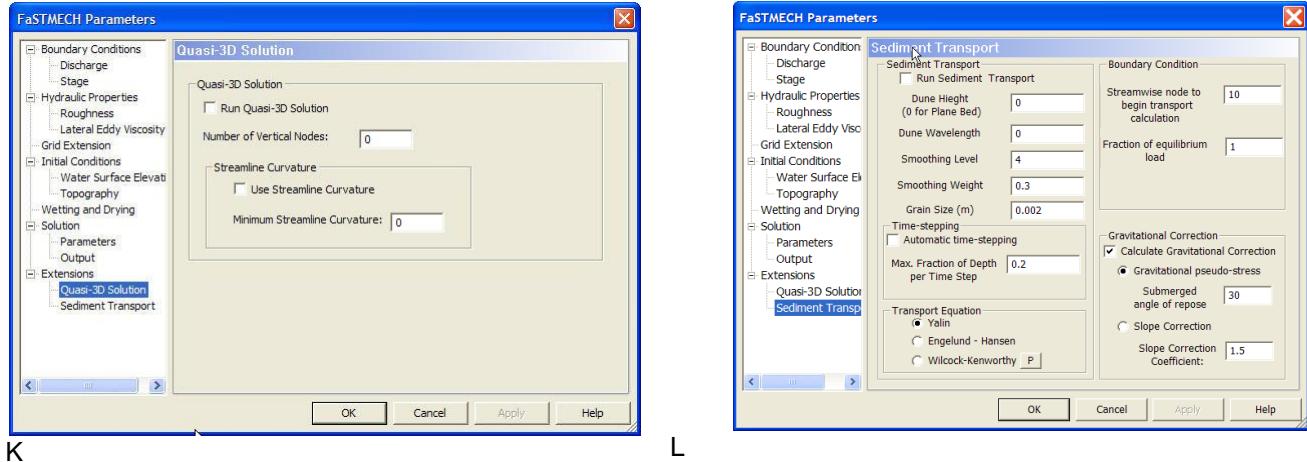


Figure 3.1.3a. Enter the following parameters into the FaSTMECH Parameters dialog. (A) Discharge, (B) Stage, (C) Roughness, (D) Lateral Eddy Viscosity, (E) Grid Extension, (F) Water Surface Elevation, (G) Topography, (H) Wetting and Drying, (I) Solution Properties, (J) Output, (K) Quasi-3D Solution, (L) Sediment-Transport Parameters.

Visualize Scalar results:

1. Turn on the **Solution / 2D Sol. Scalar Sets** object.
 - Toggle on the different Scalar sets. Notice that they are masked such that those nodes that are dry are transparent.
2. Change Scalar Attributes
 - Double-click on **Solution / 2D Sol. Scalar Sets** to bring up the Scalar Attributes Dialog (fig1 3.1.3b).
 - Experiment with attributes.

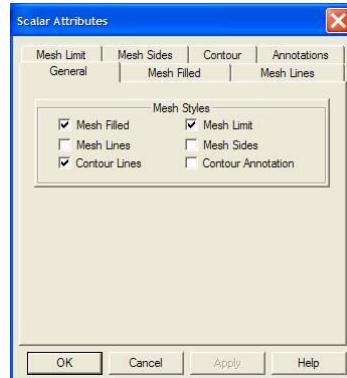


Figure 3.1.3b. The Scalar Attributes dialog. The General tab determines which attributes are on and off, and the other tabs contain parameters for each attribute.

3. Create a legend for the current **Solution / 2D Sol. Scalar Sets**.
 - Turn on the first **Data Legend / Data Legend** in the Control Bar by selecting the adjacent check box.
 - Right click **Data Legend / Data Legend** in the pop-up menu select **2D Sol. Scalar Sets**.
4. Experiment with the Data Mapping: Color Map attributes and Isovalue attributes.
 - Double-click on the **Legends / 2D Sol. Scalar Sets** to bring up the Data Mapping Dialog (fig. 3.1.3c).

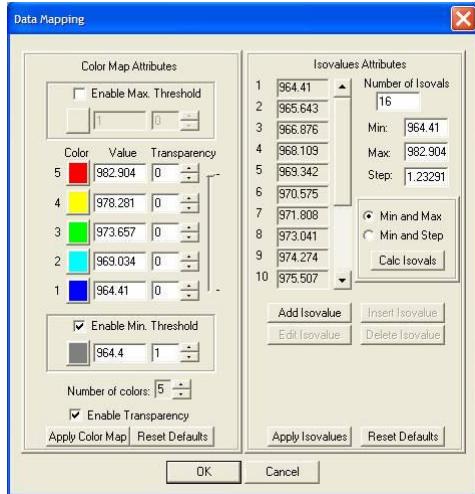


Figure 3.1.3c. The Data Mapping dialog controls the color map and contour interval of the currently selected data set of the legend, in this case the **Solution / 2D Sol. Scalar Set**.

For more information on Scalar Attributes, see [Section 2.1 Scalar Attributes](#). For more information on Data Mapping, see [Section 2.8 Legend Attributes](#).

Visualize Vector results:

1. For now, turn off the **Solution / 2D Sol. Scalar Sets** in the Control Bar. Turn on **Solution / 2D Solution Vector Sets**.
 - The default values for the vectors often need adjustment, so double-click on the **Solution / 2D Solution Vector Sets** to bring up the Vector Attributes dialog (fig. 3.1.3d).
 - Experiment with the vector attributes.

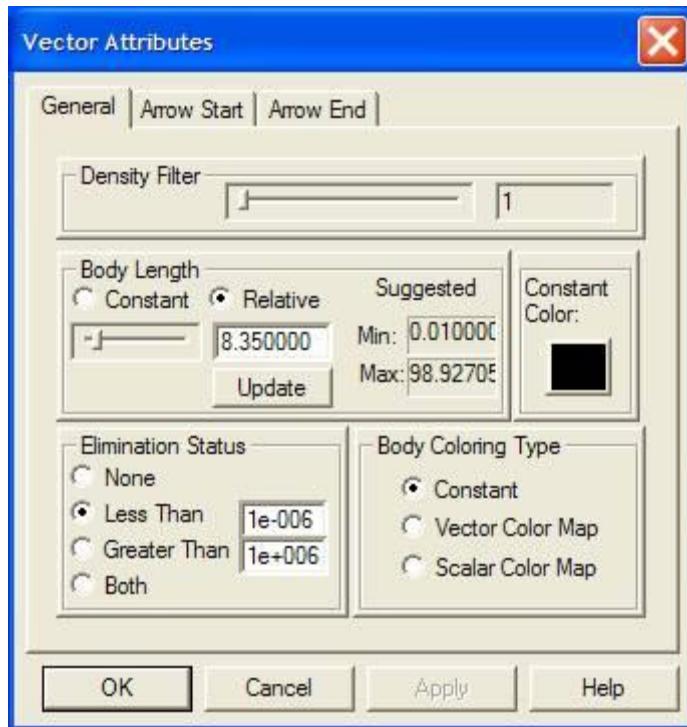


Figure 3.1.3d. The Vector Attributes dialog.

2. Turn off the **2D Sol. Scalar Sets** legend.
3. Turn on the second **Data Legend / Data Legend**.
4. Right-click **Data Legend / Data Legend** turned on above, and select **2D Sol. Vector Sets** in the pop-up menu.
5. Experiment with the Data Mapping: Color Map attributes and Isovalue attributes.
 - Double-click on **Data Legend / 2D Sol. Vector Sets** in the Control Bar to bring up the Data Mapping dialog.

For more information on Vector Attributes, see [Section 2.2 Vector Attributes](#).

Visualize Streamlines

1. Turn on **Solutions / 2D Solution / Streamlines**. The streamlines are drawn by using the currently selected **Solution / 2D Sol Vector Set**. There are a number of variations to the visualization of streamlines. The user can specify a line source with a specified number of streamlines and user-defined origin or a user defined number of random sources. The streamlines themselves can be visualized as static lines or as a particle-tracking animation. The section below introduces the static streamlines. The user is encouraged to experiment with the other types of representations.
 - Notice that there is now a graphic object called a **Jack Dragger** in the Graphic View that looks like a "jack" with some red dots on it. Click the selection tool () in the Graphics View. Clicking the mouse on the Jack Dragger Translation Plane and drag to the top of the grid as shown in figure 3.1.3e. Rotate the Jack Dragger by selecting one of its arms so that the red dots (the streamline starting points) are parallel to the upstream boundary of the grid.

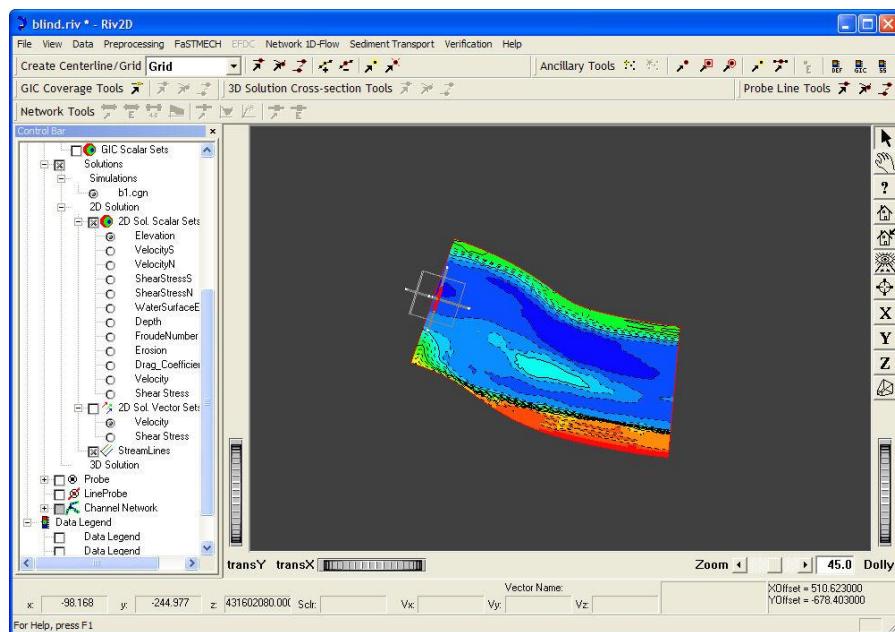


Figure 3.1.3e. The Jack Dragger can be used to locate the origin (red dots) of the streamlines.

2. Double-click on the **Solutions / 2D Solutions / Streamlines** to bring up the Streamlines Dialog and set the parameters as shown in figure 3.1.3f and figure 3.1.3g. The resulting streamlines can be moved to a new source by moving the Jack Dragger. Setting the streamline source attribute (fig 3.1.3f) to random will plot streamlines at random starting locations. See [Section 2.3 Streamline Attributes](#) for more information.



Figure 3.1.3f. The Streamline Attributes dialog. Here the Type is set to Streamlines in the General tab.



Figure 3.1.3g. The Solution Attributes tab. Here the Max. Lifetime and Max Length have both been set to 10000.

Probing the data

The Initial Conditions and the 2D Solution can be probed by using the Jack Dragger. The Probe Bar at the bottom of the iRIC GUI will display the spatial location, x, y, and z values, as well as the value of the currently selected scalar set and vector set. For more information on the Probe Bar see [Section 1.6 Probe Bar](#).

1. Turn off the **Solutions / 2D Solutions / Streamlines**.
2. Turn on **Data Objects / Probe** in the Control Bar.
3. Expand the **Probe** tree by selecting the "+" and then select the **Probe / Solution Sets** option.
4. Notice the that the Status Bar at the bottom of the application (fig. 3.1.3h) displays the current **2D Sol. Scalar Set** and its value and the current **2D Sol. Vector Set** and its component values.

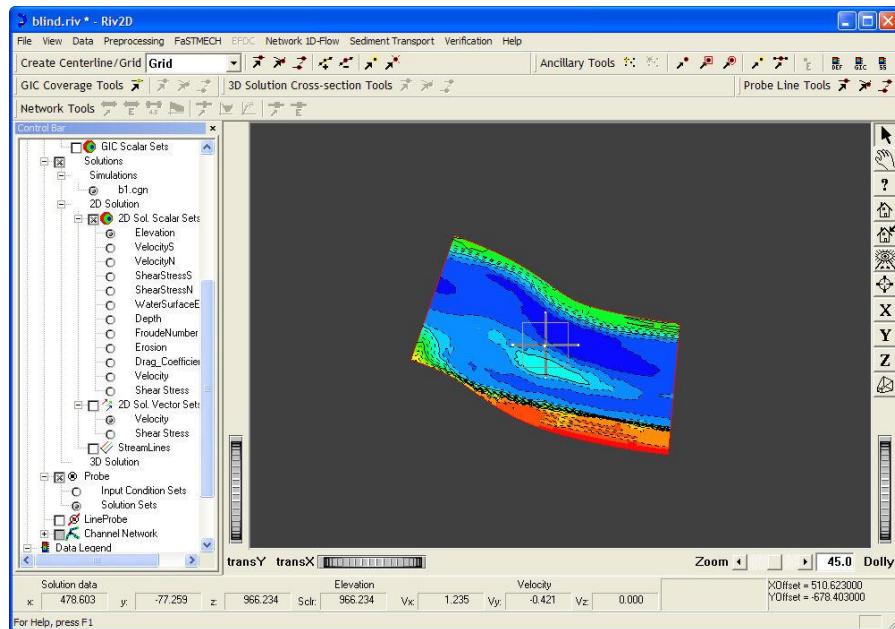


Figure 3.1.3h. The Probe Jack dragger on the 2D Sol. Scalar - Set Elevation and 2D Sol. Vector Set - Velocity.

3.2 FaSTMECH Tutorials

These tutorials lead the user through a set of typical problems from beginning to end. The tutorials assume the user has gone through the exercises in Section 3.1. These tutorials should provide enough guidance to lead the user through each problem; however, it may be helpful to use specific sections in this user's guide to elaborate on the use of a tool in the tutorial.

The tutorials include:

- Tutorial 1 – Basic Flow Application
- Tutorial 2 – Variable Roughness
- Tutorial 3 – Introduction to Vertical Structure and Secondary Flows
- Tutorial 4 – 3D Model in a Natural Channel
- Tutorial 5 – Evolution of Point Bars During Constant Discharge in Simple Meandering Channel
- Tutorial 6 – Evolution of Point Bars during Time-Dependent Discharge
- Tutorial 7 – Habitat Analysis
- Tutorial 8 – Wilcock and Kenworthy Two Fraction Sediment-Transport Model

3.2.1 FaSTMECH Tutorial 1 - Basic Flow Application

In this tutorial you will use the modeling system along with measured topography and water-surface elevations to model the flow through the reach of interest. The basic steps in this tutorial are to (1) Import the topography and water surface elevations into iRIC, (2) use the topography as a guide to build a numerical grid, (3) run the model and visualize results, and finally (4) compare the model predicted water-surface elevations to the measured values and iterate (3 and 4) as necessary until the best calibration is achieved. This tutorial assumes that the user has gone through Exercises 1 - 3 and has a basic understanding of iRIC. In synopsis, in this tutorial the student will complete the following tasks:

Tutorial 1 steps:

- Import topography
- Build numerical grid
- Define boundary conditions and model parameters
- Visualize results
- Visualize 3D results
- Verify results

Part A - Import Topography

- The raw (measured) 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 through the File Menu by selecting **File->Import->Topography**.
- In the Select File to Import dialog, choose the cigar.tpo file in the iRIC Tutorials\FaSTMECH\Tutorial 1 folder. Enter “1” in the Filter Topography dialog. A console window will appear which provides an indication of progress for the Triangle application which, by default, creates a TIN of the topography file you are importing to iRIC (fig. 3.2.1b). To finish the triangulation simply press any key on the keyboard as indicated in the console window. Open the Tutorial 1 folder in your file browser and you will see that Triangle has added a few files (fig. 3.2.1c) including Elevation.1.poly, Elevation.1.node, Elevation.1.ele, and Elevation.1.poly. The first is the Triangle input file created by iRIC and the following three are the output files. See Section 1.4.6.1 in the User's Guide for more information on the formatting of the Triangle input and output files. In the Control Bar the topography can be made visible or not visible by selecting the open box to the left of the **Ancillary**

Data / Scatter Sets branch of the Control Bar with the mouse. A check (fig. 3.2.1d) indicates the Scatter Set will be visible and an empty box indicates the Scatter Set is turned off. To add a legend, select the check box for the first **Data Legend / Data Legend**. Then right-click / **Data Legend** and select **Ancillary** in the pop-up menu.

- Save the Project (for example, Tutorial 1) by selecting **File ->Save** from the main menu
- A triangulated grid of the Ancillary Data also can be viewed. In the control bar select **Ancillary Data / TINs** to turn the triangulated grid on. By default, the raw elevation data is triangulated (fig. 3.2.1e). The **graphic attributes** of the TINs can be changed by double-clicking on the **Ancillary Data / TINs** branch in the Control Bar. For a complete description of the graphic attributes for the TIN or any gridded scalar, see [Section 2.1, Scalar Attributes](#).

```

Triangle aspect ratio histogram:
1.1547 - 1.5 :      3 | 15 - 25 :    826
1.5 - 2 :       4 | 25 - 50 : 1972
2 - 2.5 :     16 | 50 - 100 : 9892
2.5 - 3 :     16 | 100 - 300 : 3594
3 - 4 :      62 | 300 - 1000 : 629
4 - 6 :     178 | 1000 - 10000 : 465
6 - 10 :    336 | 10000 - 100000 : 82
10 - 15 :   362 | 100000 - 1000000 : 8
(Aspect ratio is longest edge divided by shortest altitude)

Smallest angle: 0.00016414 | Largest angle: 179.91

Angle histogram:
0 - 10 degrees: 22636 | 90 - 100 degrees: 9058
10 - 20 degrees: 1348 | 100 - 110 degrees: 565
20 - 30 degrees: 906 | 110 - 120 degrees: 306
30 - 40 degrees: 420 | 120 - 130 degrees: 256
40 - 50 degrees: 302 | 130 - 140 degrees: 291
50 - 60 degrees: 253 | 140 - 150 degrees: 436
60 - 70 degrees: 327 | 150 - 160 degrees: 887
70 - 80 degrees: 802 | 160 - 170 degrees: 1353
80 - 90 degrees: 10847 | 170 - 180 degrees: 4162

Memory allocation statistics:
Maximum number of vertices: 9333
Maximum number of triangles: 18404
Maximum number of subsegments: 19
Approximate heap memory use (bytes): 1036548

Algorithmic statistics:
Number of incircle tests: 119269
Number of 2D orientation tests: 134104

Press any key to exit...

```

Figure 3.2.1b.

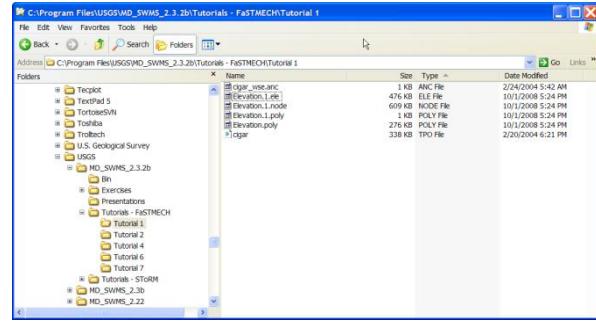


Figure 3.2.1c.

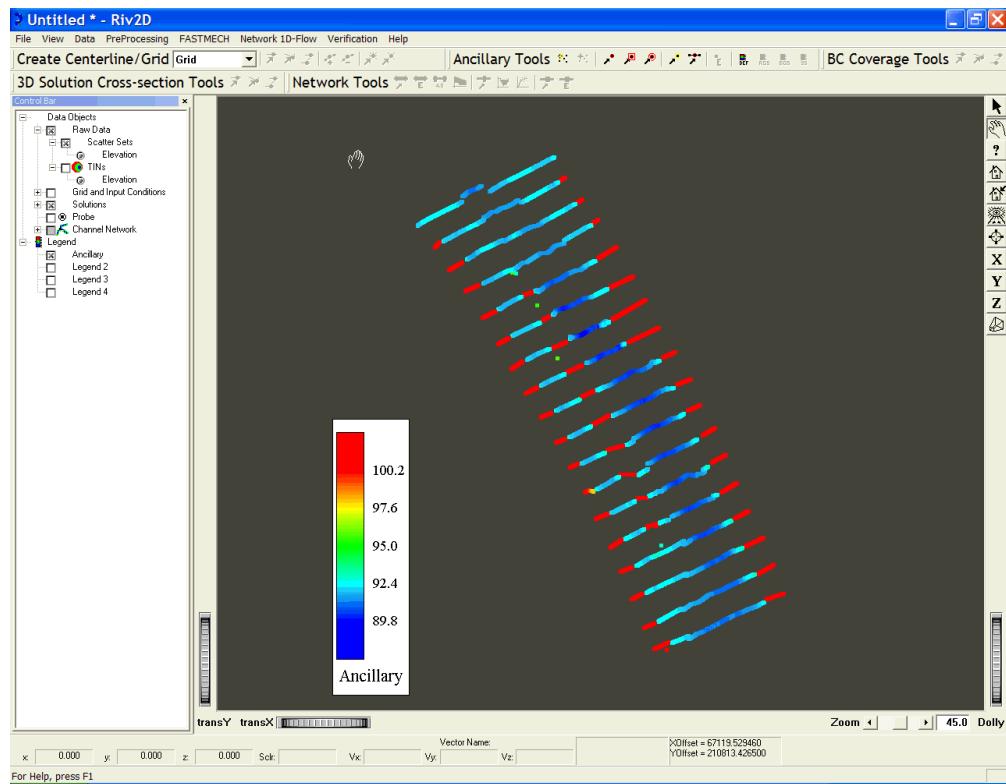


Figure 3.2.1d. View Topography

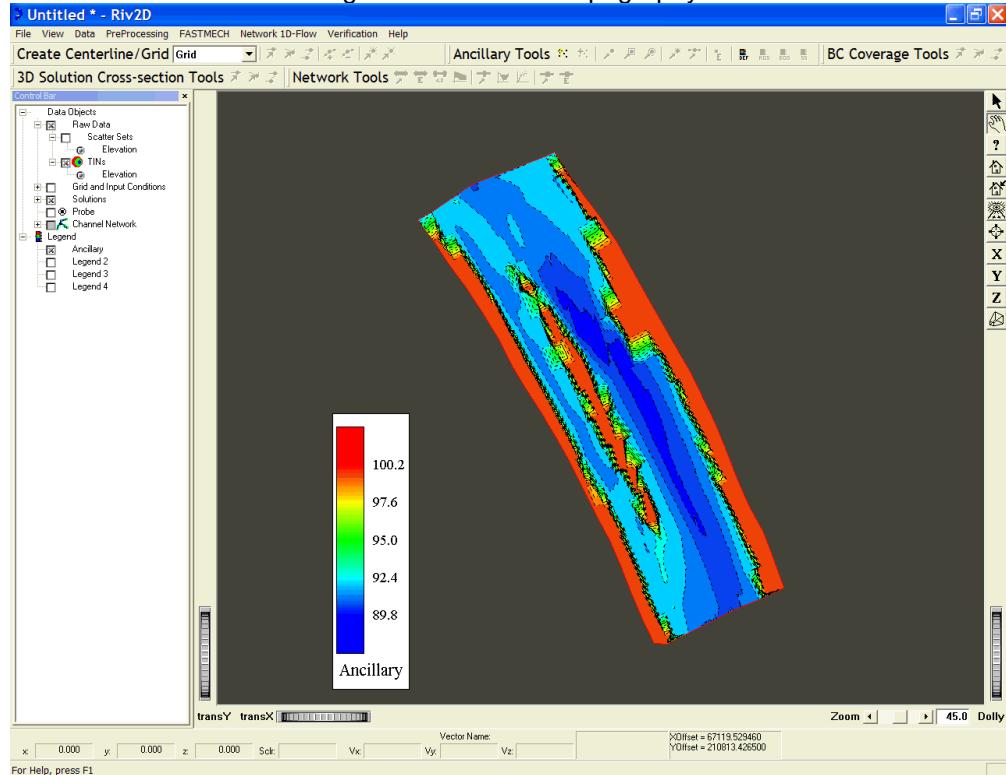


Figure 3.2.1e. View Topography as a Triangular Irregular Network Grid

Part B - Build Numerical Grid

Building a grid with the curvilinear orthogonal grid tools in iRIC is composed of three basic steps: defining the grid centerline, defining the width and density of points in the grid, and refining the curvature and location of the grid. Detailed guidelines on creating the centerline and refining the curvature of the centerline can be found in [Section 1.4.1 - Create Centerline/Grid in the MD_SWMS User's Guide](#). When defining the width of the grid, the goal is to minimize the amount of the grid outside the active channel, therefore maximizing the number of nodes in the grid contributing to the solution.

Create the spatial grid coordinates:

- In the Control Bar select the check box adjacent to **Grid and Input Conditions / Create Grid** to activate the **Create Centerline/Grid** toolbar.
- The numerical model uses a curvilinear orthogonal grid or "channel-fitted" grid (fig. 3.2.1f). Select the **Create Centerline/Grid** tool button  to interactively draw the channel centerline. To draw the centerline, click the mouse in the desired locations from upstream to downstream (fig. 3.2.1g). The upstream end of the channel in Figure 3.2.1d is at the bottom and the channel centerline should be drawn in the direction from lower-right to upper-left. When finished press enter on the keyboard.
- See Section 1.2.3 in the User's Guide for more information.

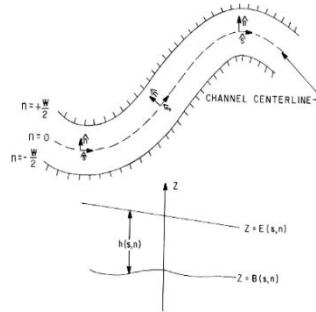


Figure 3.2.1f. Definition of the curvilinear orthogonal coordinate system (Nelson and others, 2003). Here, the grid is defined by the channel centerline. The coordinate system defaults to rectilinear when only two points define the centerline.

- From the Preprocessing menu, select **Preprocessing -> Set 2D Curvilinear Grid Parameters**. This will start the Curvilinear Grid Parameters Dialog. Set the Grid Width equal to 450 meters and define the number of points in the stream-wise and stream-normal dimension to give corresponding increments of about 10 meters. The dialog displays the distance between nodes in the stream-wise direction along the center line and the stream-normal direction, which is constant everywhere on the grid. Using the Apply button on the dialog will dynamically change the view of the grid and allow the user to find the desired spacing of nodes in the stream-wise and stream-normal directions.
- See Section 1.2.3 in the User's Guide for more information.
- The **Create Centerline/Grid toolbar** contains a number of tools to fine-tune your grid location. Use the Move Centerline button  to fine-tune the location of the grid. Use the Move Control Point button  to fine-tune the grid curvature and set the upstream and downstream boundaries normal to the general flow direction. Work with these tools until the results are similar to Figure 3.2.1h.
- See Section 1.2.3 in the User's Guide for more information.

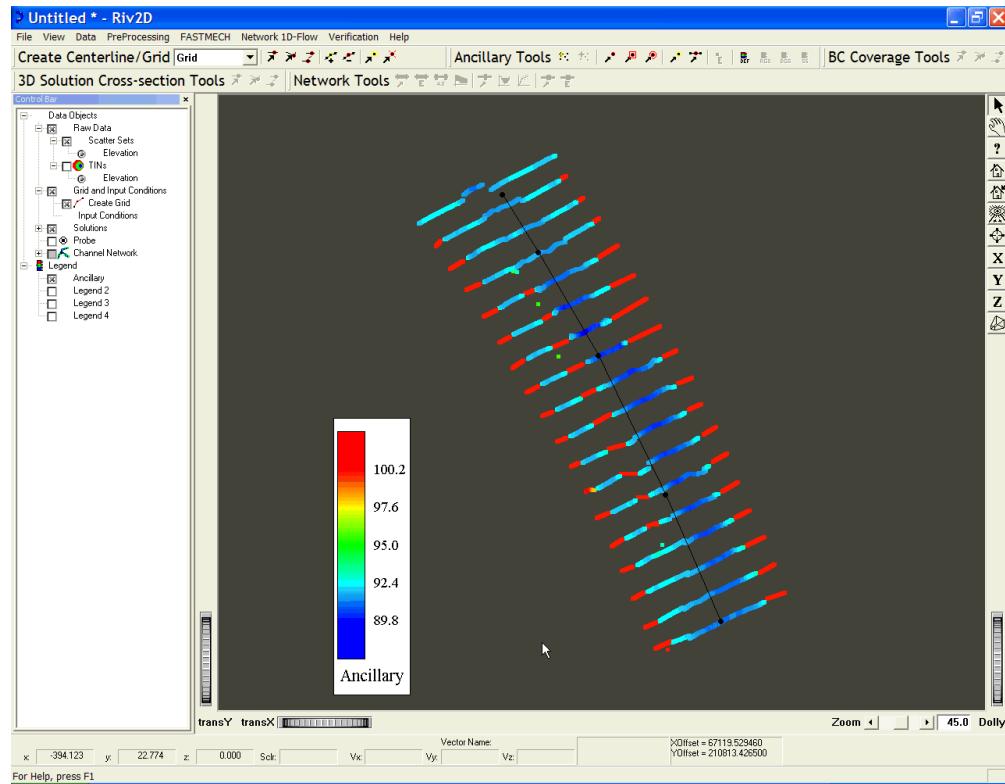


Figure 3.2.1g. Location to draw centerline. Flow is from bottom to top so start at the lower end of the channel and select three points ending near the top of the channel.

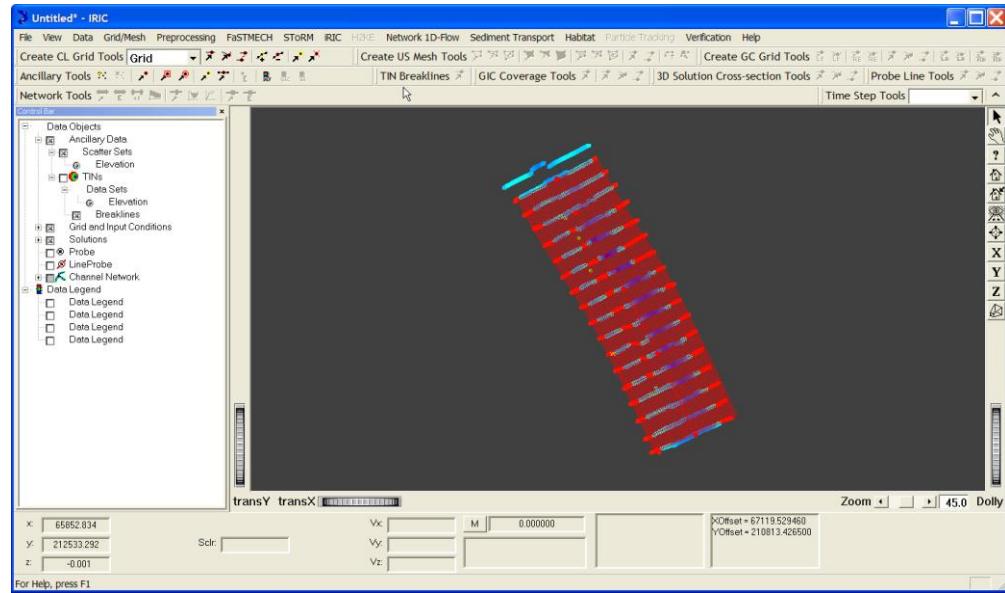
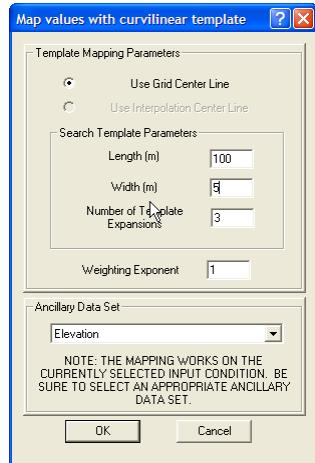


Figure 3.2.1h. Working with the *Create Centerline Grid* tool the grid can be located to cover both left and right banks adequately and the curvature of the grid can be adjusted by moving the middle centerline point so that the upstream and downstream boundaries are nearly perpendicular to the flow. Note that both the *Grid and Input Conditions* and *Create Grid* branches are turned on.

Map the measured elevations to the grid:

- In the Preprocessing menu select **Preprocessing -> Set Current Input Condition -> Map w/Template**. This will bring up the Map Values with Curvilinear Template Dialog as shown in figure 3.2.1f. The [template mapping algorithm](#) is similar to a nearest-neighbor search where the search is conducted in a bin or template with a specified length and width that follows the local curvature of the grid. In this case, use a value of 100 for the length and 5 for the width. Be sure to select Elevation as the Ancillary Data Set. By default, when the elevation is not mapped to the grid, Elevation is selected automatically.



- To view the resulting change, select **Grid and Input Conditions / Input Conditions / GIC Scalar Sets** in the Control Bar and then select Elevation in Input Conditions (fig. 3.2.1j). Add a legend by turning one of the four **Data Legends** on and then right-clicking on the corresponding **Data Legend** branch and selecting **Input Conditions** in the resulting pop-up dialog box.

- Save the Project.

Figure 3.2.1i. Enter the parameters for mapping the measured elevation to the grid by using the curvilinear template.

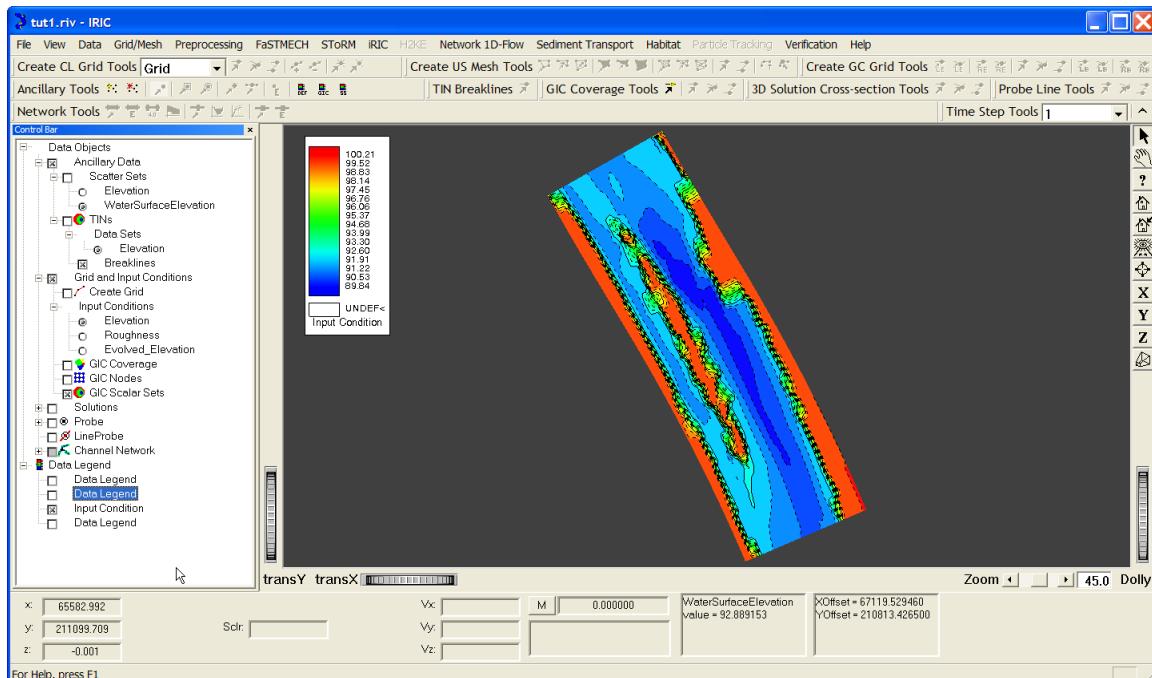


Figure 3.2.1j. The resulting computational grid with the measured elevations mapped to the grid.

Part C - Define Boundary Conditions and Model Parameters

Having created a numerical grid and mapped the measured topography to the grid, the next step is to run the FaSTMECH model.

Create new simulation:

- Create a new simulation file. In the main menu select **FaSTMECH -> New Simulation**. This opens the Save As dialog prompting for a name of the simulation file. Enter "cigar1" and select return.
- In the Control Bar under the Simulations branch of the tree view, note the cigar1.cgn file. The .cgn extension is the default extension for 2D modeling I/O files.
- Results will be verified with measured water-surface elevations. To import the measured water-surface elevations from the main menu select **File -> Import -> Ancillary Data -> Scalar** to open the Import Ancillary File dialog (fig. 3.2.1k). Select the WaterSurfaceElevation Data file type in the Hydrologic Data Type Tree by double-clicking it. The Data Type attribute list is defined as WaterSurfaceElevation. Then browse for the water-surface elevation file by selecting the browse button  and then selecting the file "cigar_wse.anc" in the File Open dialog. Press OK to import the file.

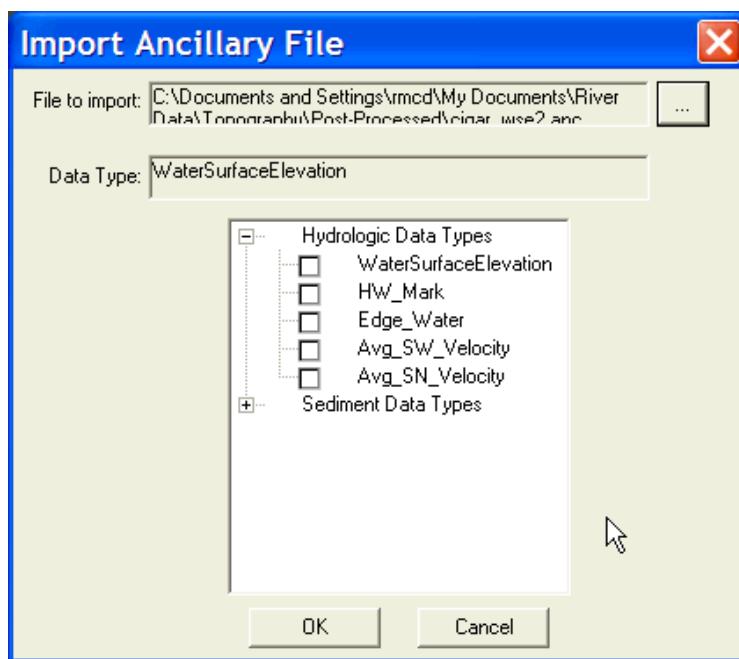
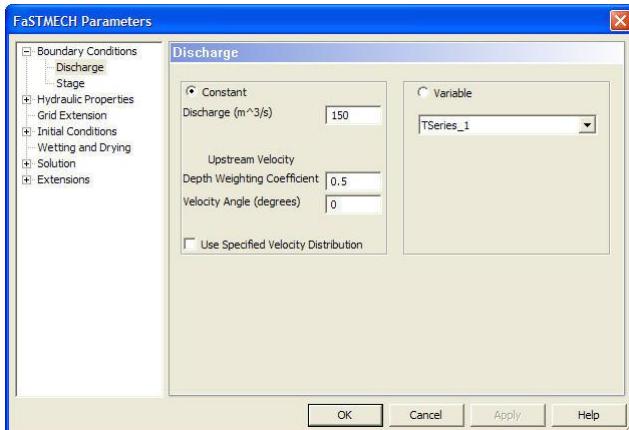


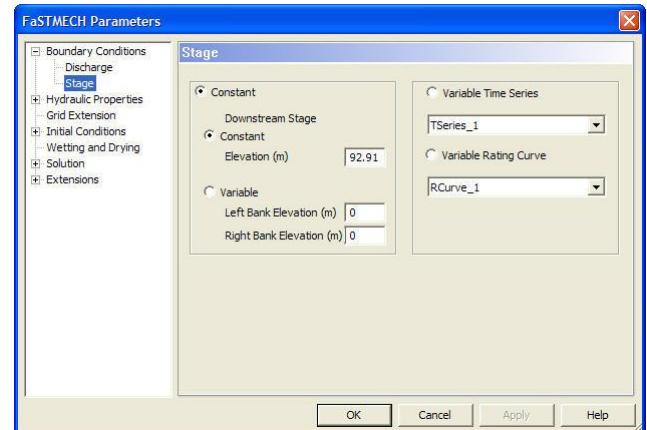
Figure 3.2.1k. The Import Ancillary File dialog with the Data Type and File to import attributes set.

Define the Boundary Conditions:

- Once a simulation file has been created, you can define the boundary conditions and model parameters. Select **FaSTMECH -> Edit Input File** from the main menu and enter the values as shown below in figure 3.2.1l.



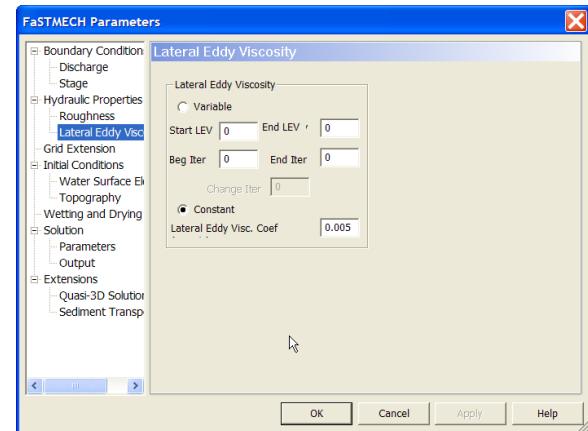
A



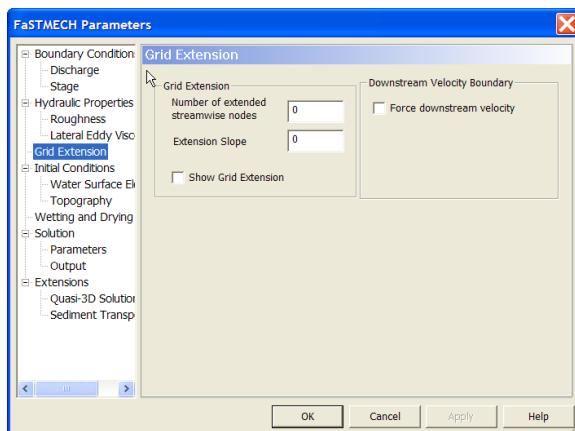
B



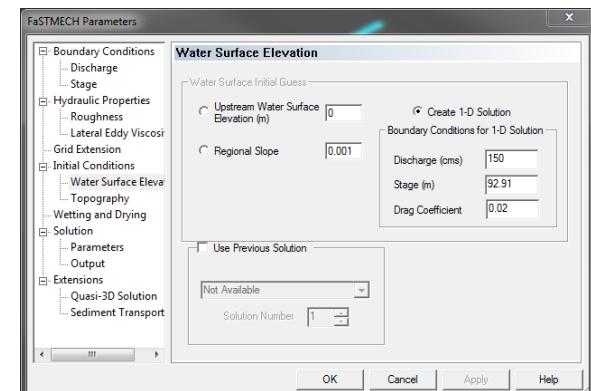
C



D



E



F Note the **Create 1-D Solution** option is selected

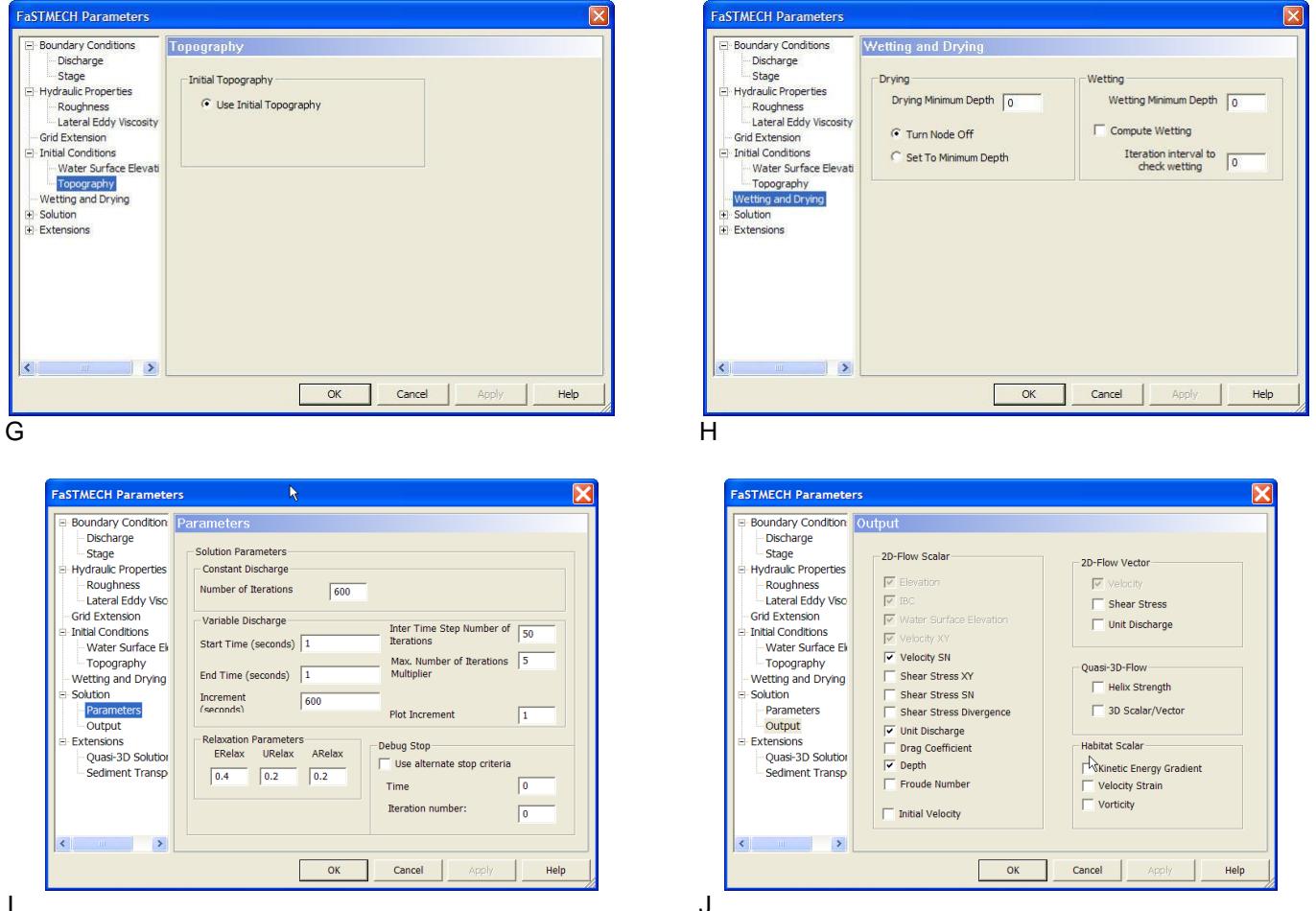


Figure 3.2.1I. Enter the following parameters into the FaSTMECH Parameters dialog. (A) Discharge, (B) Stage, (C) Roughness, (D) Lateral Eddy Viscosity, (E) Grid Extension, (F) Water Surface Elevation, (G) Topography, (H) Wetting and Drying, (I) Solution Properties, (J) Solution Output.

- Select OK in the Dialog and then from the menu select **FASTMECH -> RUN** from the main menu. Upon completion of a successful model run, inspect the convergence plot by selecting **FASTMECH -> View Convergence**. The Convergence plot shows the percent deviation from the normalized discharge at each "cross-section" in the grid. In other words, the predicted discharge at a cross-section divided by the specified discharge multiplied by 100. Acceptable values are generally thought to be +/- 3 percent. Figure 3.2.1m shows that the predicted discharge at each cross-section is approximately +/- 0.4 percent.

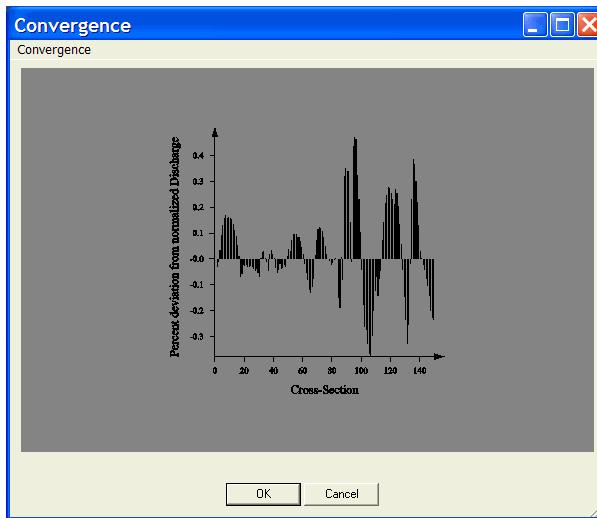


Figure 3.2.1m. A plot of the convergence of the simulated discharge to the specified discharge at each cross-section of the model grid.

- The convergence history or the RMS change of the cross-sectional mass conservation can be viewed by selecting **Convergence -> Convergence History** from the menu in the Convergence dialog (fig. 3.2.1n).

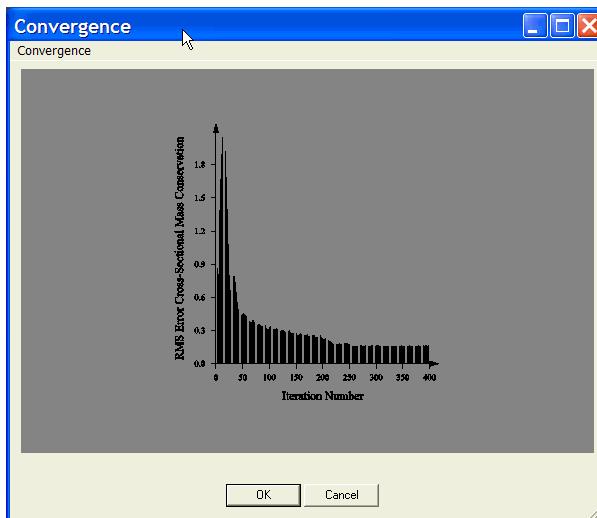


Figure 3.2.1n. A plot of the model's solution convergence history represented as the RMS error in the simulated discharge to the specified discharge at each cross-section of the model grid at each iteration.

Part D - Visualize Results

iRIC provides a complete suite of tools for visualizing model results. In the **Solutions / 2D Solutions** branch in the Control Bar there are three representations of the 2D Solution. Scalar results represent the magnitude of a value through contour plots, the vector results represent magnitude and direction, and the streamline results represent integrated motion through static streamlines or particle animations. Each of these visualization types are described below.

Scalar Results

The resulting scalar data sets available by default with a 2D model run are shown in the **2D Solution / 2D Sol. Scalar Sets** branch in the Control Bar. An example of the predicted water-surface elevation is shown in figure 3.2.1o. Note that dry nodes are masked when showing a solution scalar set. Attributes of scalar sets can be set by double-clicking the **2D Sol. Scalar Sets** branch in the Control Bar.

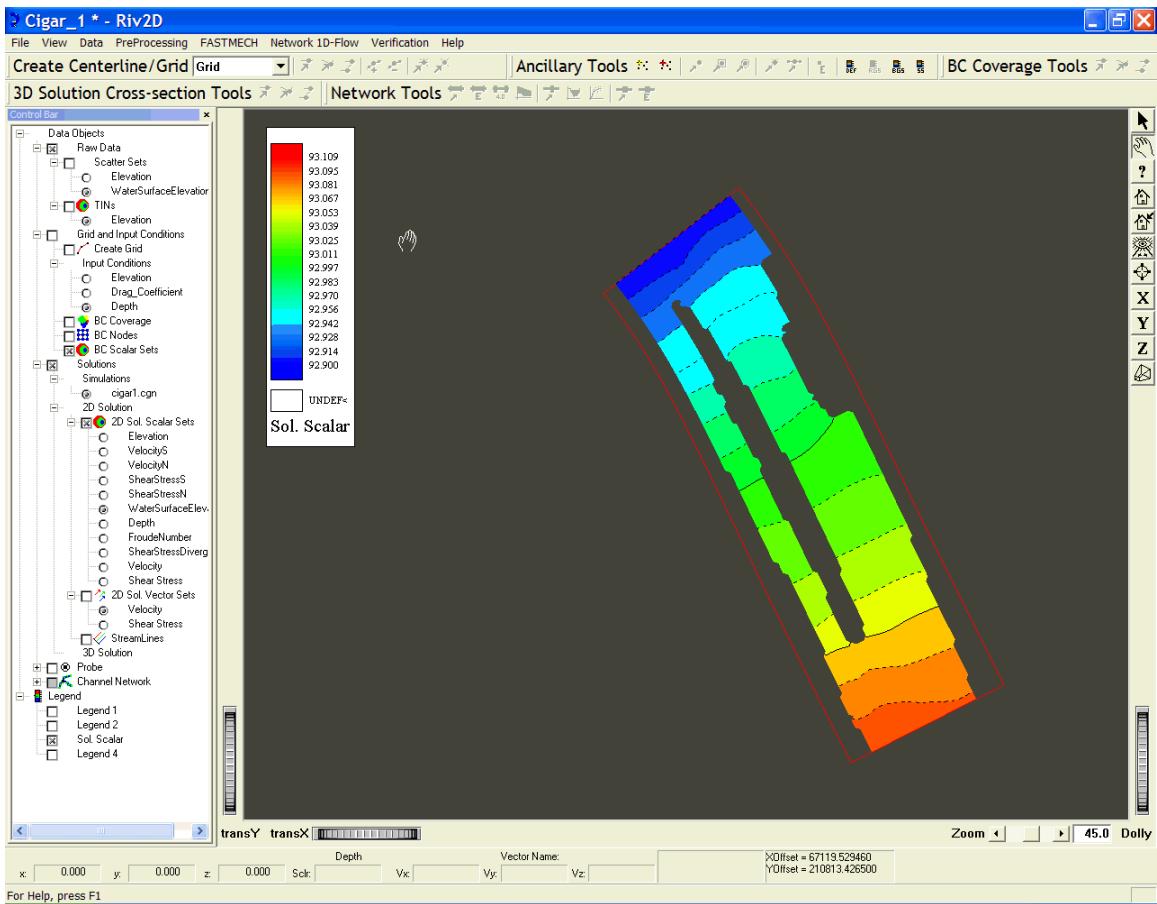


Figure 3.2.1o A plot of the water-surface elevation in the modeled reach created by choosing WaterSurfaceElevation with the 2D Solution Scalar Set selected under the Solutions.

Vector Results

The resulting vector data sets available by default with a 2D model run are shown in the **2D Solution / 2D Sol. Vector Sets** subheading. An example of the predicted water-velocity vector field for a portion of the reach at the head of the island is shown in figure 3.2.1p. Note that those nodes that are dry are masked when showing a solution vector set. The graphical [attributes](#) of vector sets can be set by double-clicking the **2D Sol. Vector Sets** branch in the Control Bar.

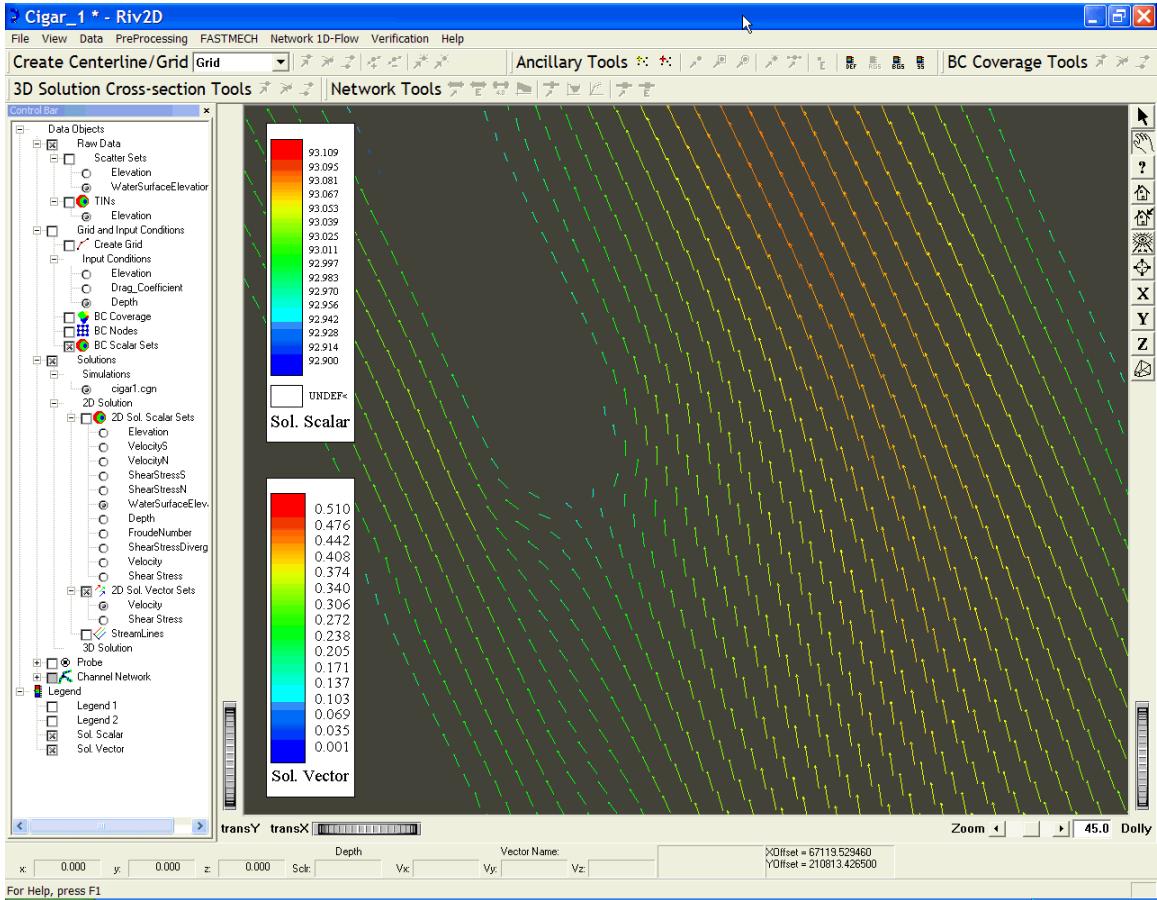


Figure 3.2.1p. A plot of velocity in the modeled reach represented as vectors. Vectors are color coded and scaled according to their magnitude.

Streamline and Particle Tracking Results

The resulting vector data sets can be viewed as a set of streamlines or particle-tracking animations. Note that these representations are of the advective component of the currently selected vector field only. An example of the streamlines for the velocity field shown in figure 3.2.1p is displayed here in figure 3.2.1q and 3.2.1r. The [attributes](#) of the streamlines and particle-tracking can be set by double-clicking the **2D Solution / Streamlines** branch in the control bar.

Examples of both the streamline and particle-tracking visualizations are shown in figures 3.1.2n and 3.1.2o.

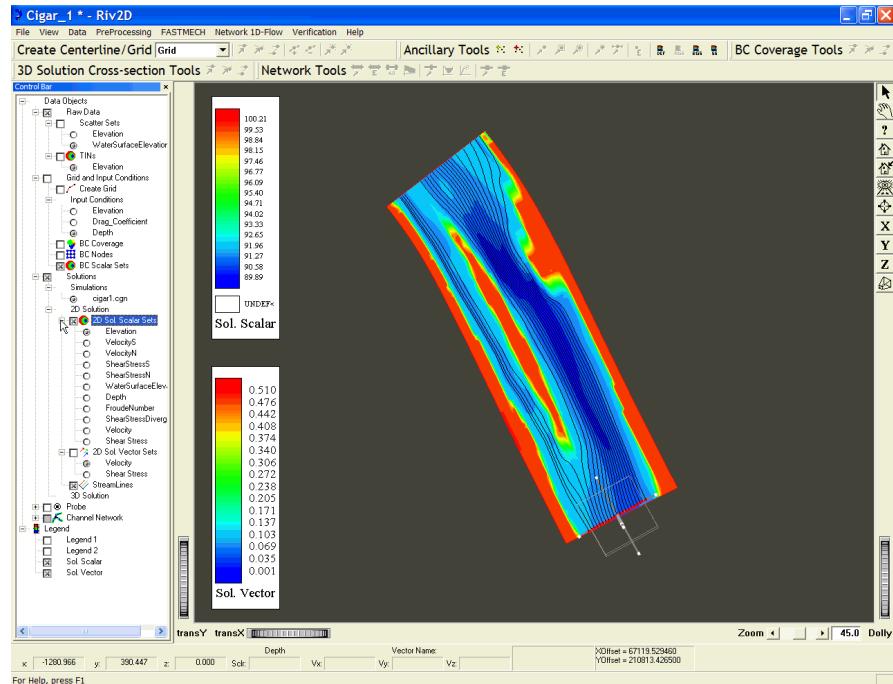


Figure 3.2.1q. The velocity solution, the currently selected 2D Vector Set, is represented as static streamlines using the Jack Dragger as the source. The streamlines use a constant color.

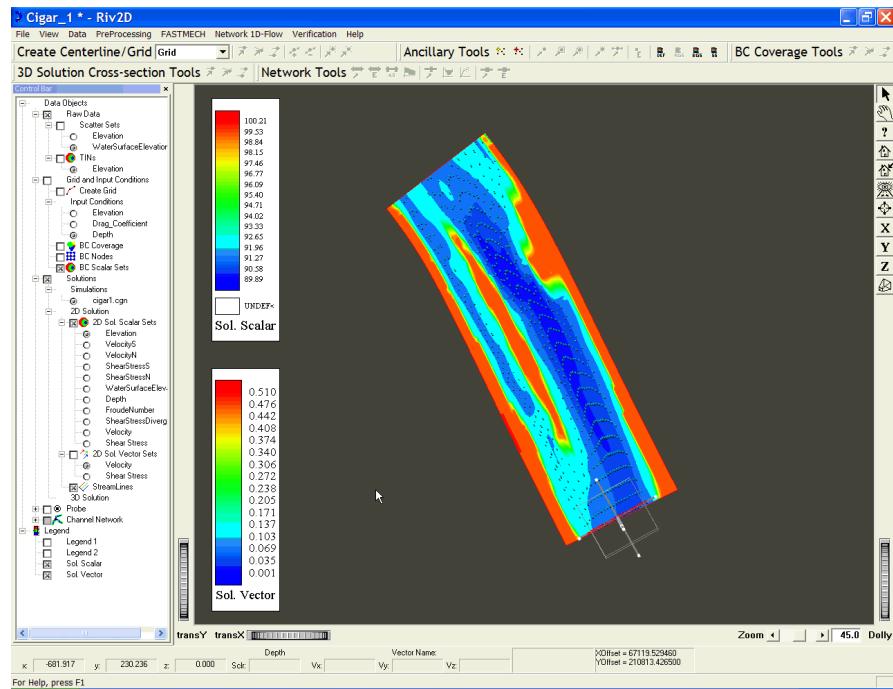


Figure 3.2.1r. The velocity solution represented as a single frame of particle-tracking animation. Here the Streamline Type attribute is set to stream-motion, and the Randomize attribute is not selected so that the particle at each source point starts at the same time. The cross-stream shear in the main (river right) channel flow becomes evident as the particles move downstream from the source.

Part E - Visualize 3D Results

The following steps will show how to take advantage of the ability to visualize the two-dimensional modeling results in three-dimensions.

Topography:

The Boundary Conditions topography scalar set is used to display the topography. Other layers can be added on top. The exercise below will show the user how to layer the scalar, vector and streamline solutions on top of the topography.

- First make sure to be in the **Examiner Viewer** by selecting from the main menu **View -> Examiner Viewer**. Rotate the topography to the desired orientation.
- Turn on the **Grid and Input Conditions** and set the input condition to **Elevation** and turn on the **GIC Scalar Sets** (fig. 3.2.1s).

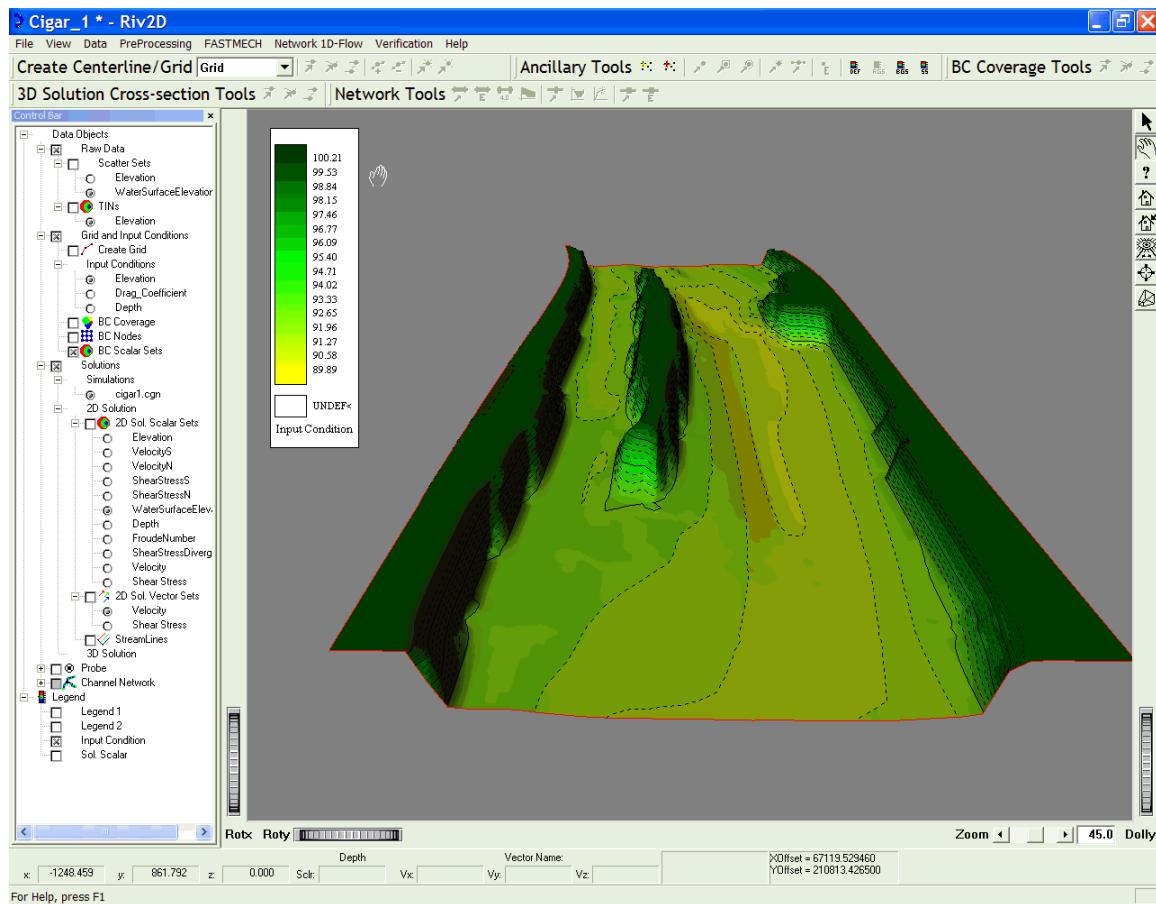


Figure 3.2.1s. Topography input condition that will be used as a background to overlay the model solution.

- Set the vertical scale by selecting **View -> Set Vertical Scale**. In the dialog that follows enter 5.0 for the scale.
- Change the color map to something other than the default rainbow color map, as follows. In the Control Bar, make sure that the **Grid and Input Conditions** branch is on, and then select the check box of the **GIC Scalar Sets** branch and make sure that **Input Condition** is set to **Elevation** by selecting the radio button. Set one of the legend tabs on and right-click the label and select **Input Condition** in the pop-up

menu. Double-clicking on this legend label will bring up the Data Mapping dialog. Change the Color Mapping Attributes so that there are three colors from dark green at the highest value, to green, to yellow at the lowest (fig. 3.2.1t). To make the changes, select the Apply Color Map button.

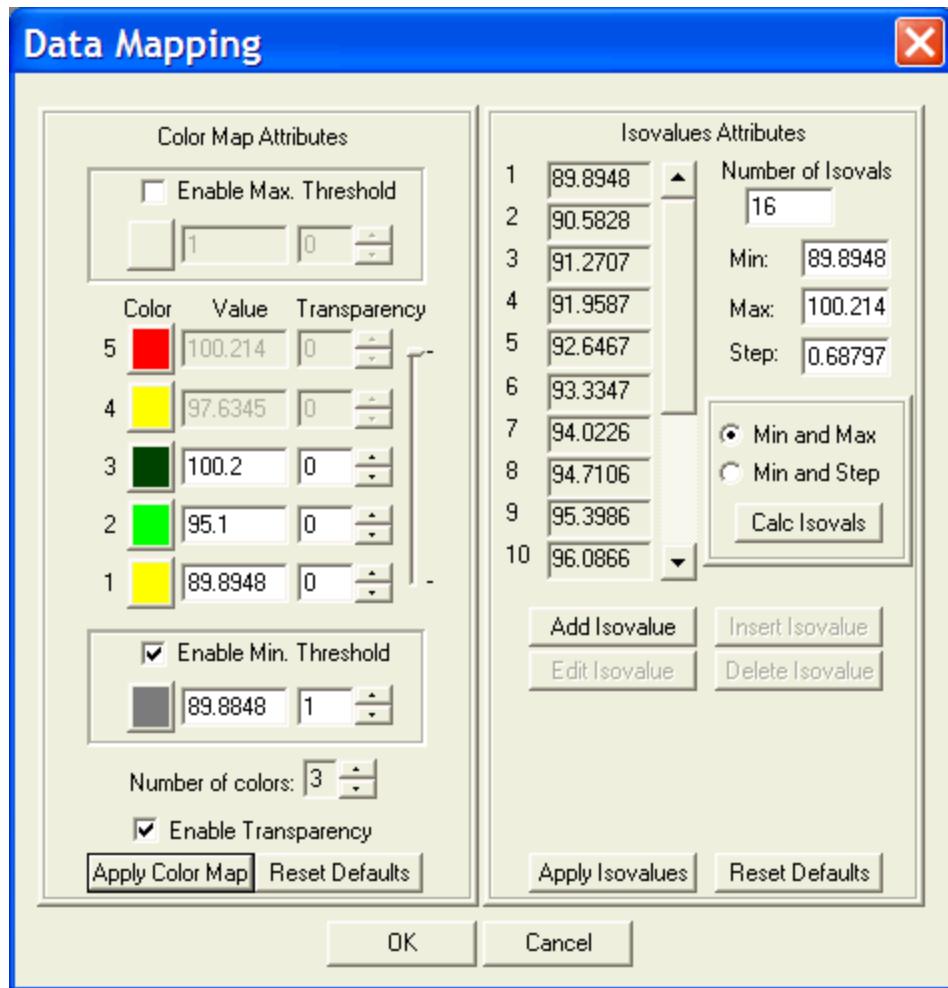


Figure 3.2.1t The colors and contour levels of the topography in figure 3.2.1s were set with the attributes shown in the Data Mapping Dialog here.

- Change the direction of the incoming light by selecting **View -> Light Editor** in the main menu. Move the arrow on the ball to change the direction of light.

Solution Scalar:

The water-surface elevation scalar solution, or any other scalar or vector solution, can be visualized as an overlay to the topography by setting the Solution Z-Index to that of the predicted water-surface elevation. In three-dimensions, the vertical coordinate of the scalar or vector solution can be represented at the topography or the water-surface elevation.

- In the control bar select **2D Solutions / 2D Sol. Scalar Sets / WaterSurfaceElevation**. Make sure the **Solutions** and **2D Sol. Scalar Sets** are both set to on (fig. 3.2.1u).
- To set the solution scalar elevation from the menu select **View -> Solution Z Index -> WaterSurfaceElevation**. This will set the water-surface elevation scalar set at its true elevation.

- As in the topography section above, we will change the Data Mapping Attributes. Select one of the **Legends** and right-click the legend label and select **2D Sol. Scalar** in the pop-up menu. Double-clicking the **2D Sol. Scalar Sets** label will bring up the Data Mapping Dialog. Select three colors from dark blue at the highest value to light blue the lowest.

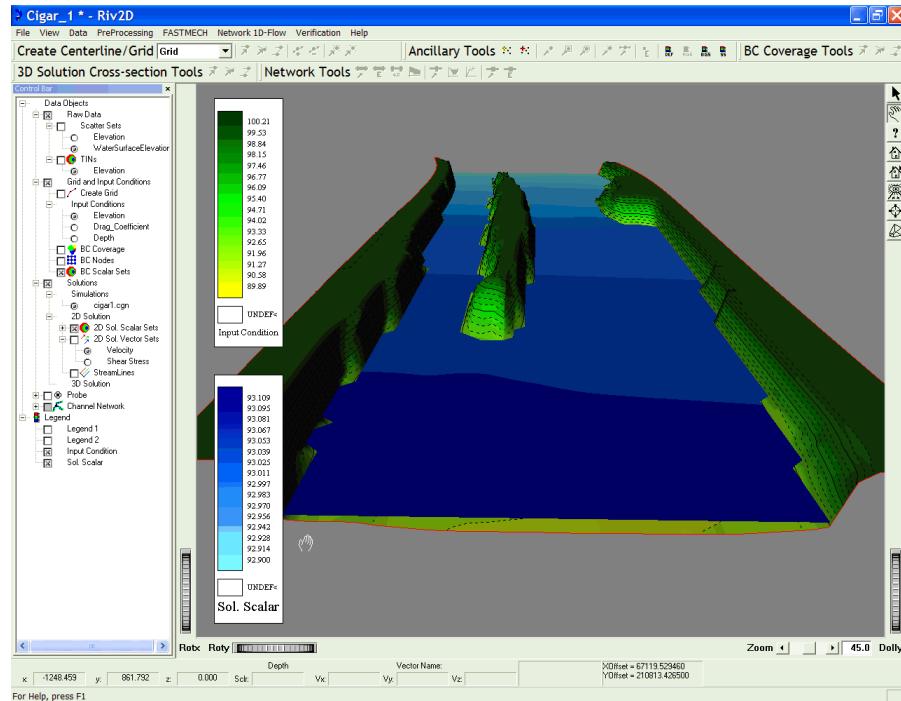


Figure 3.2.1u. Here we have overlain the topography with the model solution for the water-surface elevation

Vector Solution:

Once the solution elevation is set to be equal to that of the water-surface elevation, vectors can be drawn on the water-surface elevation solution.

- In the control bar turn on the **2D Solutions / 2D Sol. Vector Sets / Velocity** (fig. 3.2.1v).
- Access the vector attributes by double-clicking the **2D Sol. Vector Sets** label. If needed, change the scale of the vectors or change the Arrow End from the default Chevron to a Triangle.
- Make any desired changes to the color map and add a **2D Sol. Vector** legend.

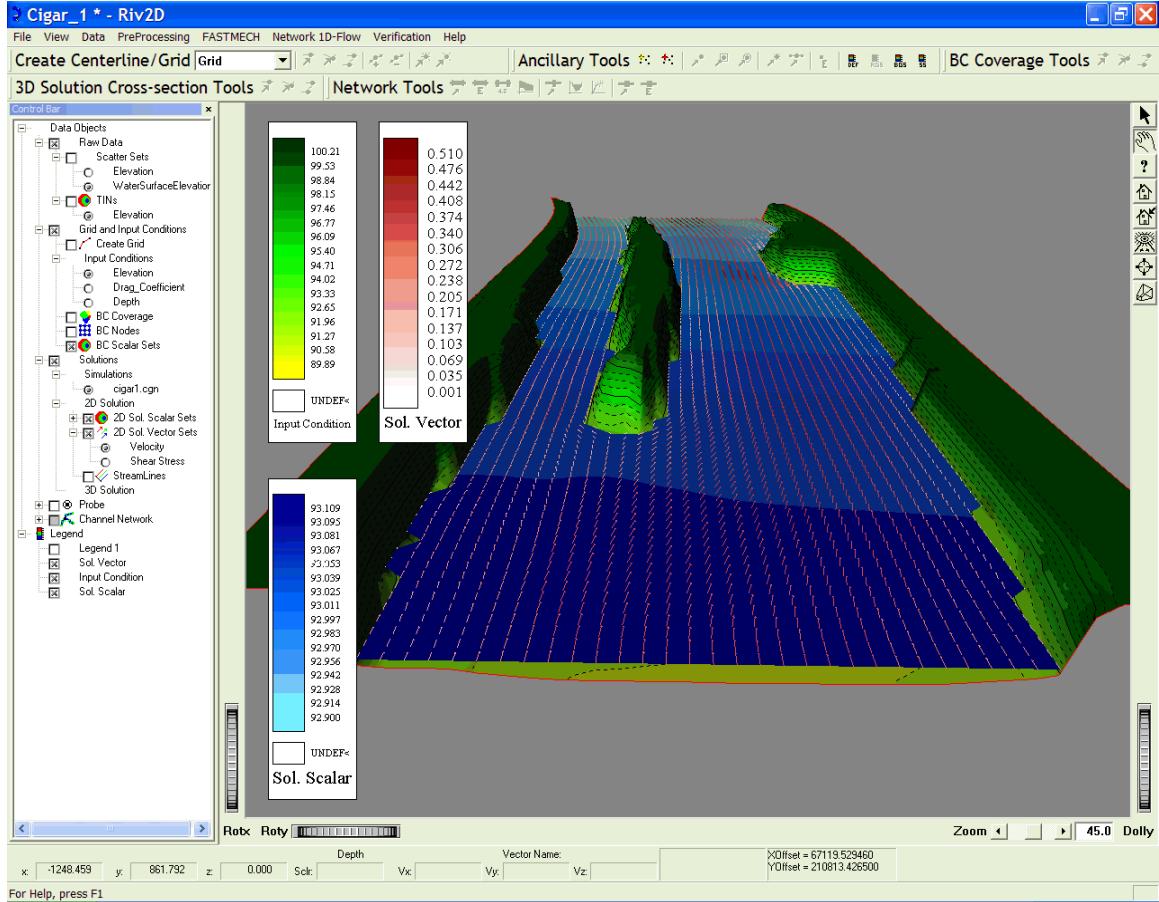


Figure 3.2.1v. An example of velocity vectors added to the water-surface elevation solution. All solutions are represented at the water-surface elevation.

Streamline Solution:

It is possible to set the streamlines to be near the water-surface elevation by dragging the source up to the water-surface elevation.

- In the control bar select the **2D Solutions / Streamlines** (fig. 3.2.1w).
- Move the streamline dragger to the desired location. To see how to manipulate the streamline dragger, see this [movie](#) demonstration.
- Access the **Streamline Attributes** by double-clicking on the **2D Solutions / Streamlines** text in the control bar. In the General Attributes tab set the following parameters:
 1. Set the width for the dragger sources. Visualize the change of the source point locations while dragging the Dragger Source Width slider bar. Set the width to approximately 180 meters, or the width of the wetted area.
 2. Set the Dragger #Pts to 20. Be sure to hit the Apply Button.
 3. Set the Type to Streamlines.
- Select the Solution Attributes tab and set the Max Length to 10000 and Max Lifetime to 10000.
- Select the Stream Motion tab and set the Length Factor to 50, the Time Step to 10, and uncheck the Randomize Start option.

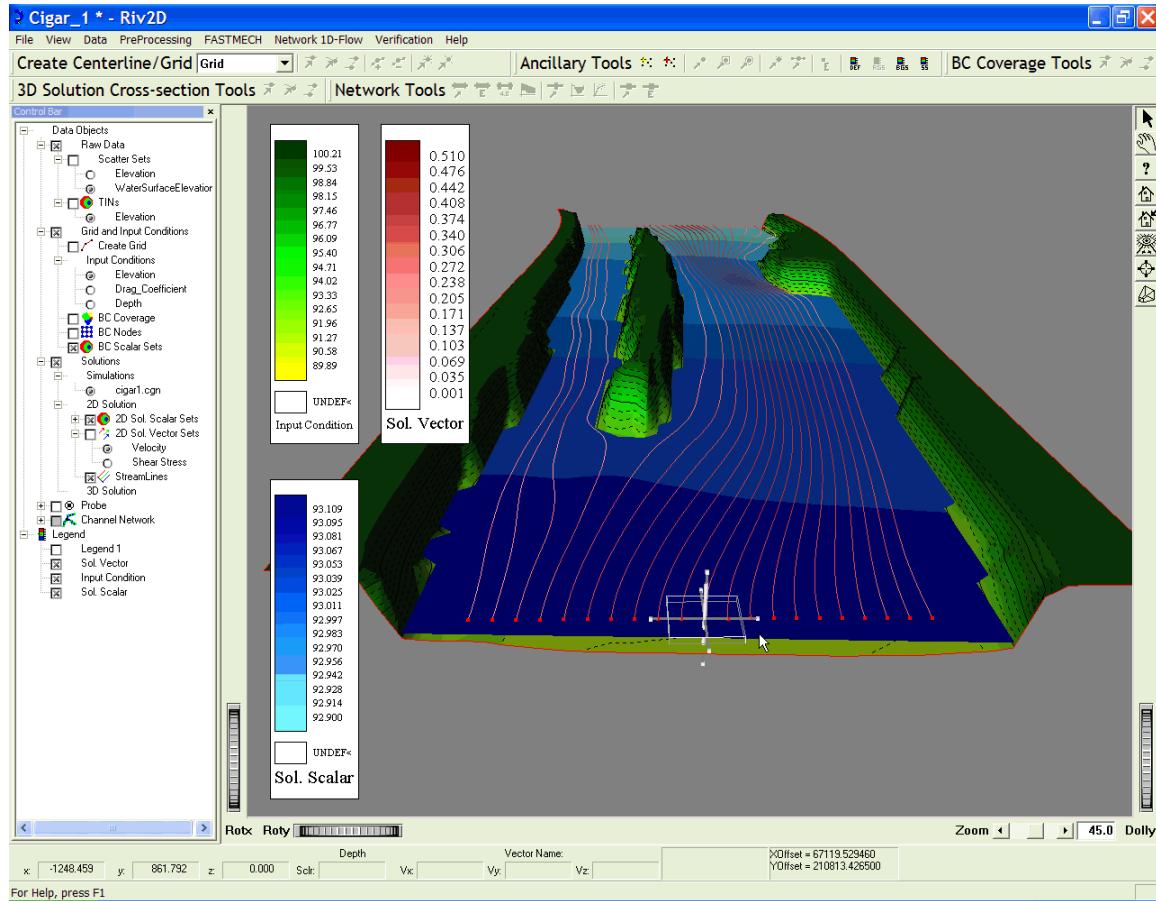


Figure 3.2.1w. The elevation of the streamline source is set to the water-surface elevation by interactively moving the source dragger to the location viewed here.

Part F - Verify Results

iRIC has a verification tool that allows visualization of model predictions compared to measured values in a number of different formats for both water-surface elevation and velocity. In this example, a set of measured water-surface elevations are supplied. With a successful model run, select **Verification -> WS Verification** in the main menu. This will bring up by default a plot of the predicted versus observed water-surface elevations along with the mean water-surface elevation as a function of streamwise distance. The different verification visualizations are shown below in figures 3.2.1 x-aa. Each of these can be accessed through the Water Surface menu in the Verification dialog. For more information on the verification tool, see [Section 1.5.8 Verification Menu](#).

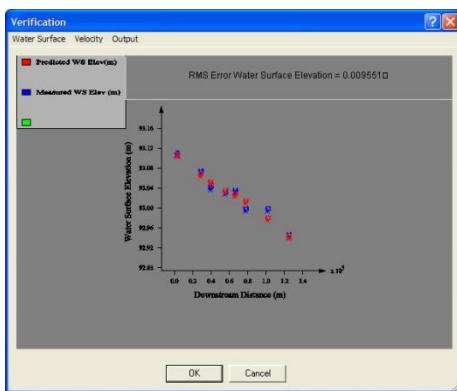


Figure 3.2.1x. Predicted and observed water-surface elevation with downstream distance. This representation often can be useful when calibrating the roughness. The water-surface slope is, in part, set by the roughness, and viewing the slope of the predicted water-surface elevation versus the observed can guide the user to a good value for the roughness. For example, if the roughness is set too high, the predicted water-surface slope will be greater than the observed.

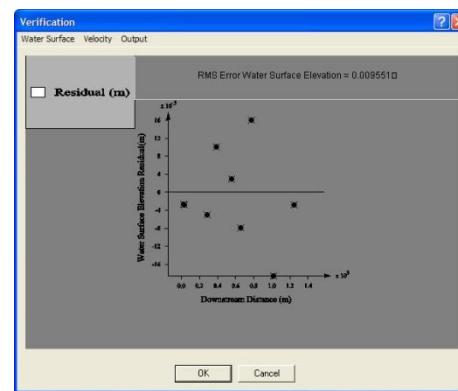


Figure 3.2.1y. Residual (predicted minus observed) of the predicted water-surface elevation with distance.

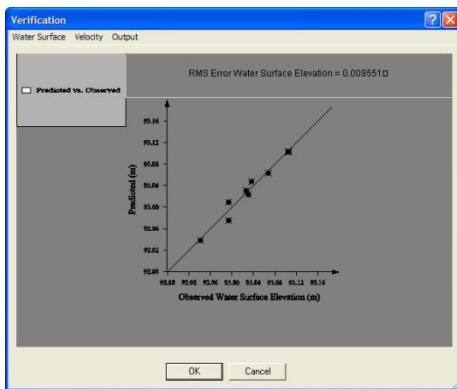


Figure 3.2.1z. Predicted vs observed water surface-elevation.

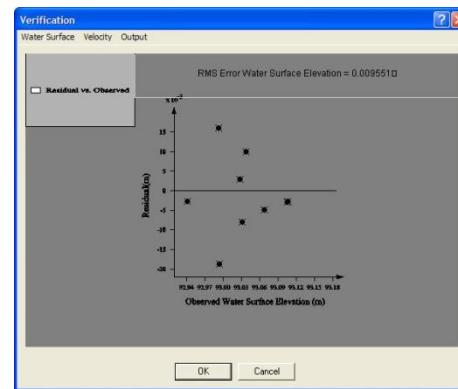


Figure 3.2.1aa. Residual (predicted minus observed) of the predicted water-surface elevation vs the observed water-surface

elevation.

3.2.2 FaSTMECH Tutorial 2 - Variable Roughness and GIC Coverages

The modeling system can be used along with the measured grain sizes to interactively map roughness to the computational grid. There are three basic steps in this tutorial: (1) to run and calibrate the model to the measured water-surface elevation by using a constant drag coefficient, (2) to use the measured grain sizes (mapped to the grid) to create a variable Z0 roughness formulation on the model grid that reflects the grain-size distribution on the bed of the river, and (3) to recalibrate the model to the measured water-surface elevations using the new roughness distribution. Note that this improves the calibration of the modeled water-surface elevation profile relative to the measured one. This tutorial assumes that the user has gone through Tutorial 1 and is familiar with the basic operation of iRIC. In the tutorial, the student will complete the following tasks.

Tutorial 2 steps:

- Import topography and grid.
- Run simulation and calibrate model with constant roughness.
- Create grain-size input condition.
- Map roughness to the grid.
- Calibrate model with variable roughness.

Part A - Import Topography and Grid

Import Topography and Ancillary Data

Import the topography file Trinity.tpo from the iRIC Tutorials\FaSTMECH\Tutorial 2 folder in the iRIC directory. In addition, import (1) the measured water-surface elevation file WSE_4000cfs.anc and (2) the measured mean grain size D50_040326.anc.

Import Existing Model Grid

From the menu select **File -> Import -> Curvilinear Grid -> XY**. In the File Open dialog select Tutorial 2.riv. This will import the node points of the grid but without any of the topography or ancillary data mapped to the grid. Importing the grid in this way can be useful in keeping the model grid consistent from one iRIC project to another. In the Control Bar turn on the **Grid and Input Conditions** and the **Grid and Input Conditions / Create Grid** data objects (fig. 3.2.2a). Note the Centerline for the Curvilinear Grid is defined by points that are approximately one channel width apart and follow as closely as possible to the mean curvature of the channel.

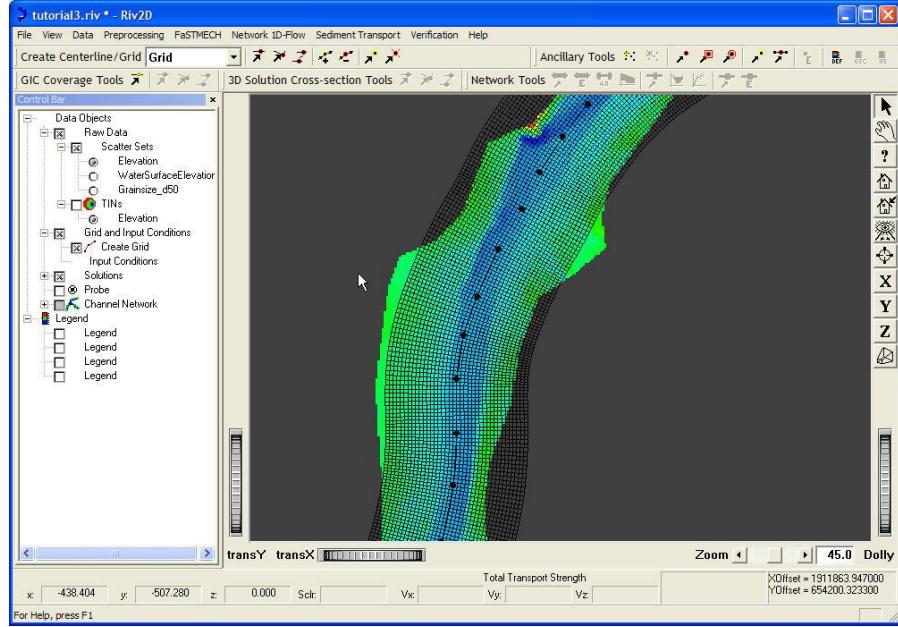


Figure 3.2.2a. The measured topography and curvilinear grid.

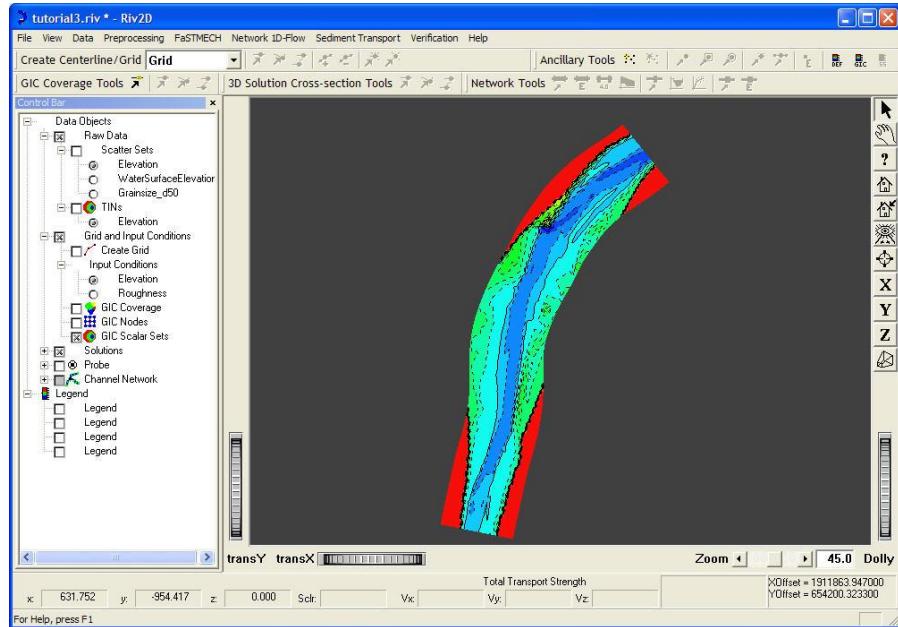


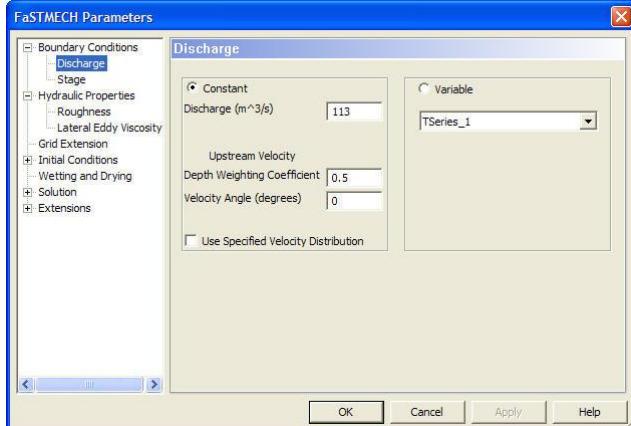
Figure 3.2.2b. Elevation mapped to the grid by using the TIN method.

Map Elevations to the Grid

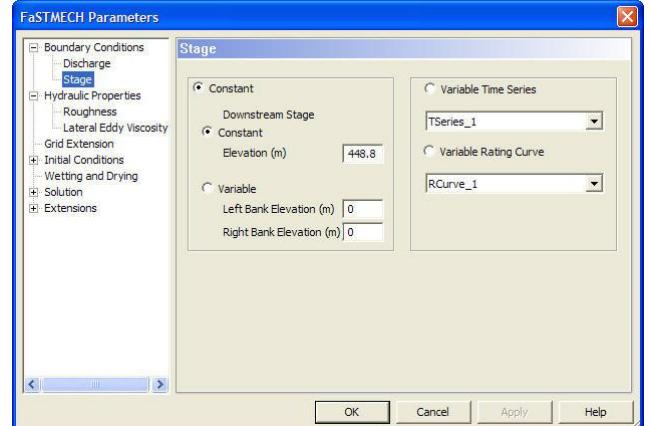
The measured elevations used in this example are generally evenly distributed and provide a nice example of using the Map w/TIN method to map the elevation. From the menu select **Preprocessing -> Set Current Input Condition -> Map w/Tin** (fig. 3.2.2b). When using the Map w/TIN method, all points on the grid that are outside of the TIN boundary are given the maximum elevation in the TIN.

Part B - Run Simulation and Calibrate by Using Constant Roughness

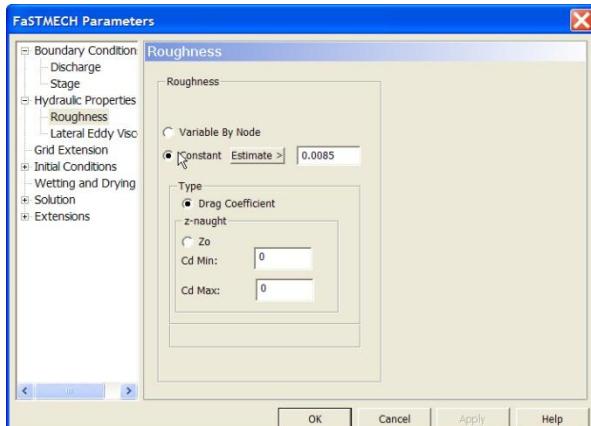
From the menu select **FaSTMECH->New Simulation**. In the Save As Dialog type in a name for the simulation (for example, sim1). From the menu select **FaSTMECH->Edit Input File** and enter in the values shown in figure 3.2.2c. Select OK in the Dialog and then from the menu select **FaSTMECH -> RUN** from the main. Upon completion of the model run, the Depth is added to the **Grid and Input Conditions / Input Conditions**. The water-surface elevation verification is shown in figures 3.2.2d and 3.2.2e. Note that the upstream end of the reach is poorly predicted when using a constant value for the roughness.



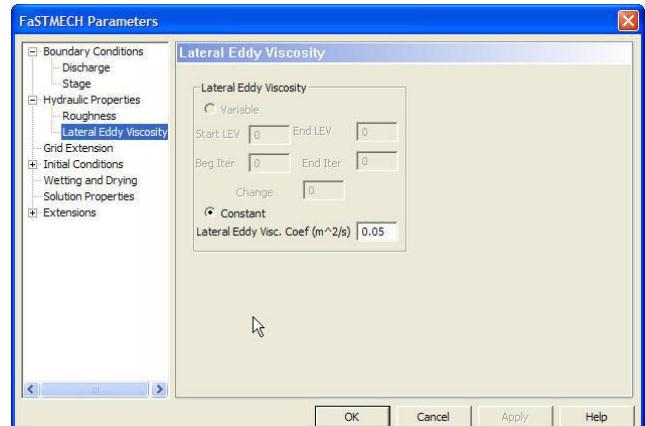
A



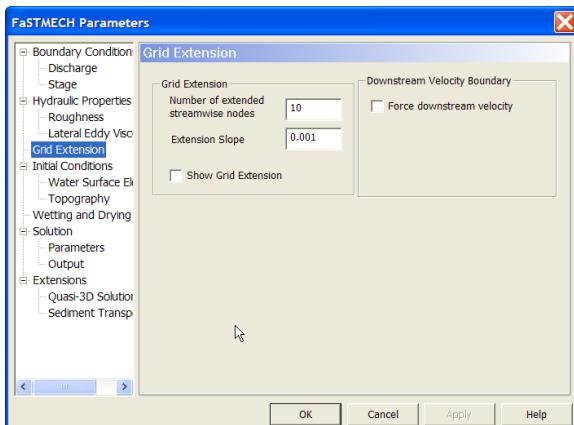
B



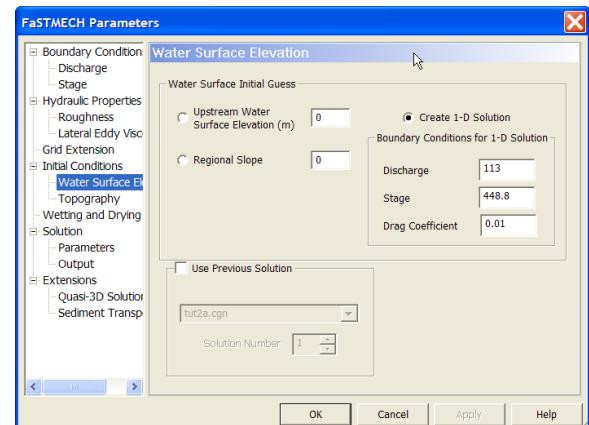
C



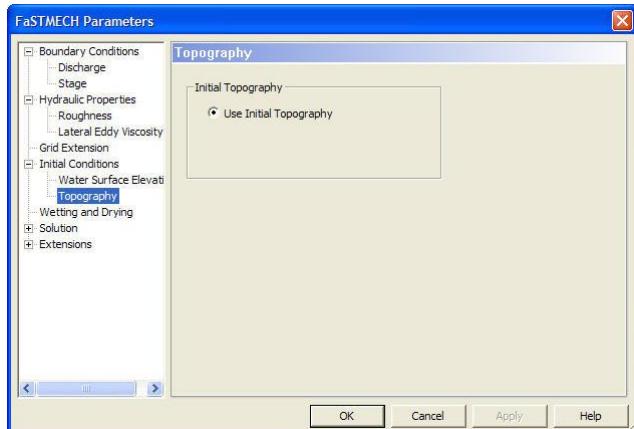
D



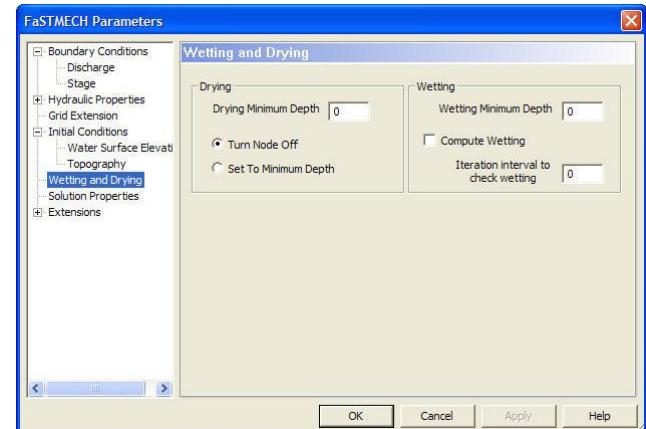
E



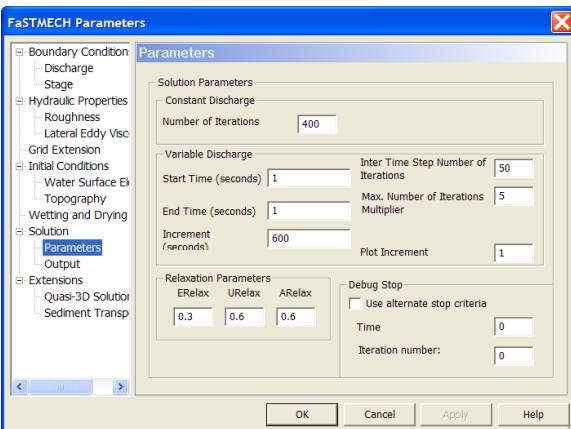
F Note the Create 1-D Solution option is selected



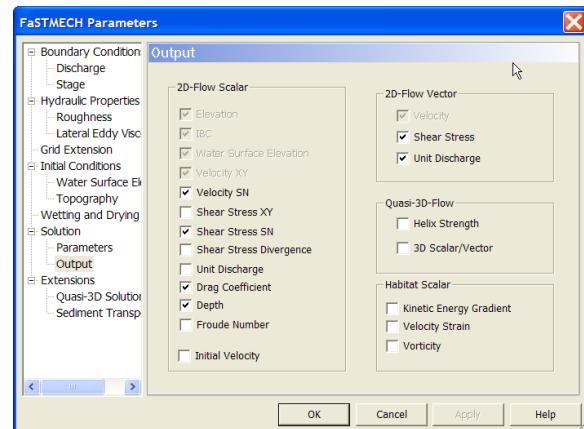
G



H



I



J

Figure 3.2.2c. Enter the following parameters into the FaSTMECH Parameters dialog. (A) Discharge, (B) Stage, (C) Roughness, (D) Lateral Eddy Viscosity, (E) Grid Extension, (F) Water Surface Elevation, (G) Topography, (H) Wetting and Drying, (I) Solution Properties, (J) Solution Output.

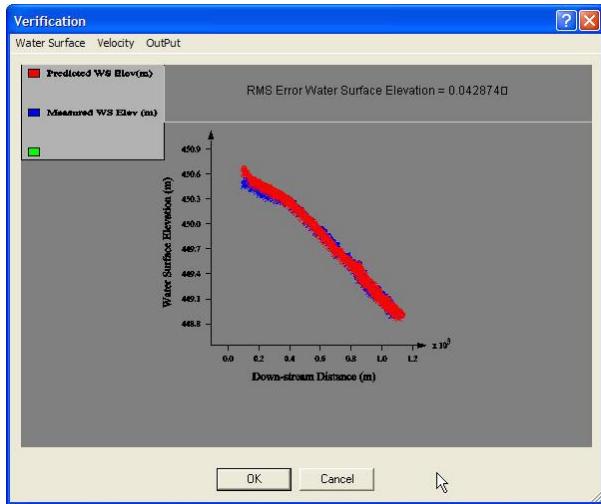


Figure 3.2.2d. Measured and predicted water-surface elevations as a function of distance along the grid centerline. Note that the predicted water-surface elevations are high at the upstream end.

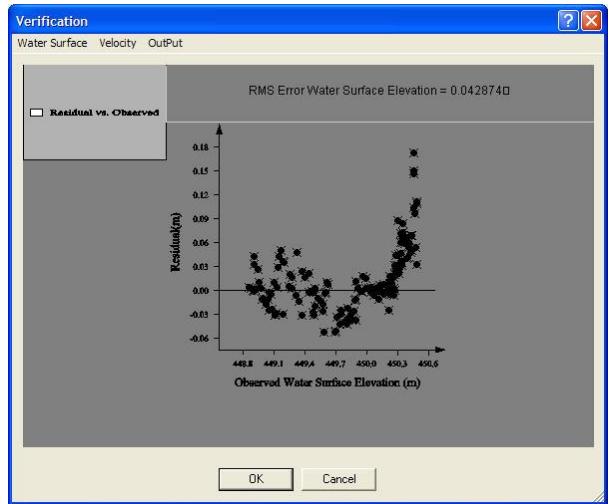


Figure 3.2.2e. The residual of the predicted water-surface elevation.

Part C - Create Grain-Size Input Condition

In Part B, it was noted that the predicted water-surface elevation poorly matched the measured water-surface elevation near the upstream end of the modeled reach. The water-surface elevation was steeper than the measured water-surface elevation suggesting that a constant drag coefficient predicted reasonably well the lower section of the modeled reach, but its value was too small for the upper section. While a constant drag coefficient can be a good approximation in some reaches (for example, in Tutorial 1) it may be a poor approximation in others. For the reach of river modeled here, there is an extensive grainsize data set that we will use to parameterize the drag coefficient.

Create a Grain-Size Input Condition

- In the control locate the **Grid and Input Conditions / Input Conditions** branch of the data tree and right-click on the **Input Conditions** data object and in the pop-up menu select **Create Input Condition -> Grainsize_d50**.
- In the Set Grainsize_d50 Default dialog leave the value as is and select OK.

Map measured grain-size data to the grid

Map the measured grain-size data to the Grainsize_D50 input condition. Two coverages also will be imported to account for the area of the grid not covered in the measured data.

- Select the **Grid and Input Conditions** branch of the data tree in the Control Bar so that it is on (for example, Grid and Input Conditions).
- Make sure the **Grainsize_d50** is the currently selected input condition. From the menu select **Preprocessing -> Import Current Coverage**. From the Import Coverage Data dialog select Grainsize_Cov.anc. View the Coverages by turning on the **Grid and Input Conditions / GIC Coverage** data object (fig. 3.2.2f). We can map the measured grain-size data by using the Map w/TIN method. Ensure that the Grainsize_D50 is the currently selected input condition and that Grainsize_D50 is currently selected in Ancillary Data | Grainsize_D50. This is important because the Map w/TIN method maps the currently selected Ancillary Data set. From the menu select **Preprocessing -> Set Current Input Condition -> Map**

w/TIN (fig. 3.2.2g). To see the values of the imported coverages, right click **GIC Coverages / Right Bank D50** and from the pop-up menu select **Properties**.

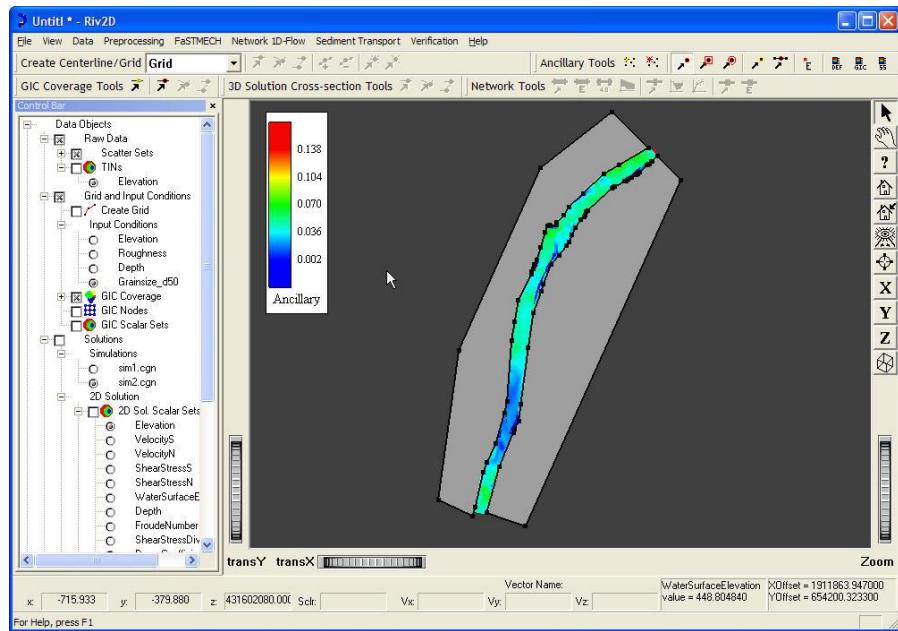


Figure 3.2.2f. The two grain-size coverage polygons and the measured grain size data are shown. The coverage polygons are drawn as close as possible to the limits of the measured data.

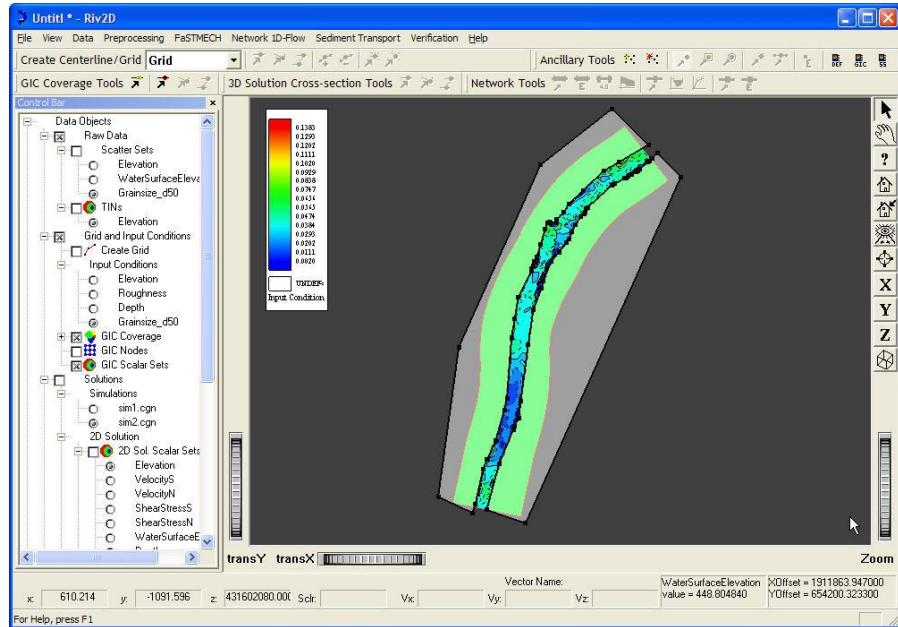


Figure 3.2.2g. The grain size coverage polygons and the Grainsize_D50 input condition as mapped with the TIN.

Part D - Map Roughness to the Grid

Map Roughness to the Grid Using Grain size

Using the grain-size input condition created in Part C and the predicted depth from the simulation with a constant drag coefficient in part B, create a new variable roughness input condition.

- From the menu select **Preprocessing -> Set Roughness Input Condition -> Map w/Grainsize**. Set the parameters for the Create Roughness - Grain size dialog as in figure 3.2.2h below. This takes the grain size distribution and the depth solution from the first simulation (using a constant drag coefficient) and calculates a variable roughness, in the form of a drag coefficient, at each node in the grid. Next, you will use this variable drag coefficient to re-calibrate the model predictions.

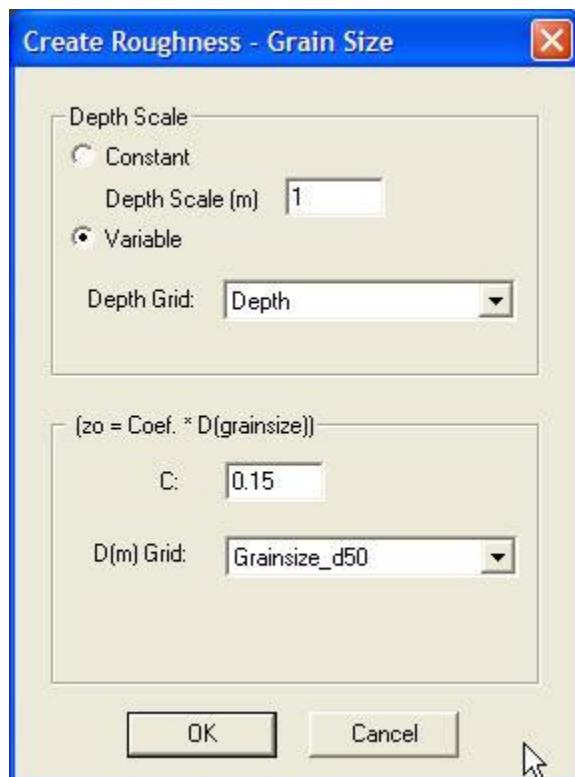


Figure 3.2.2h. Map roughness by using the input conditions of Grainsize_D50 and the Depth solution from the previous run.

Part E - Calibrate Model with Variable Roughness

Create a New Simulation and Calibrate Model to the Variable Drag Coefficient

- Create a new simulation file and call it sim2.
- Create the input file as in figure 3.2.2c with the single exception of the **Hydraulic Properties / Roughness** as shown in figure 3.2.2i and then run the model.
- The results of the water-surface elevation verification are shown in figures 3.2.2j and 3.2.2k. Overall, the fit is much better than the previous simulation (fig. 3.2.2d) when using a constant drag coefficient especially in the upper section of the reach. The overall RMS error in the prediction is 0.02 when using the variable roughness and 0.4 when using the constant roughness.

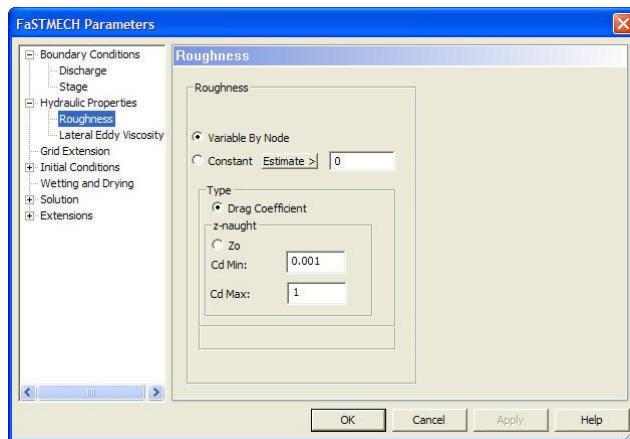


Figure3.2.2i. Input file parameters for the sim2 simulation.

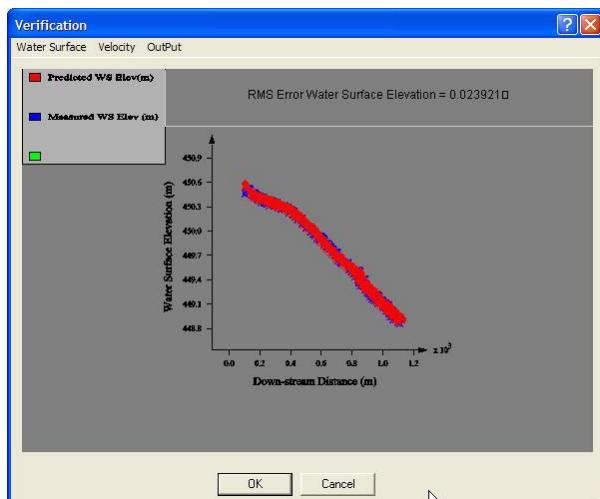


Figure 3.2.2j. Measured and predicted water-surface elevations as a function of distance along the grid centerline.

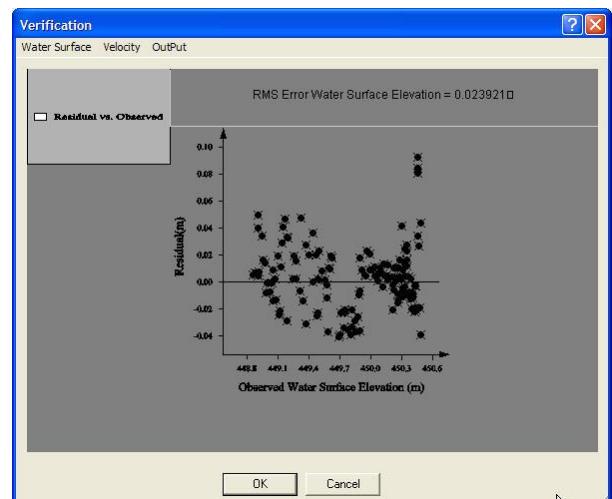


Figure 3.2.2k. The residual of the predicted water-surface elevation.

3.2.3 FaSTMECH Tutorial 3 - Introduction to vertical structure and secondary flows

In order to introduce three-dimensional flow modeling, this tutorial leads the student through the steps of construction, execution and visualization for a 3D model of two highly idealized channels. To construct these idealized channels, the iRIC interface incorporates a simple “Channel Builder” that build topography and model grids with just a few simple parameters. Although these channels are simpler than all natural channels, they allow the student to explore and understand certain important physical effects that arise in 3-D flows, notably the simple structure of shear flows and the generation of secondary flows. This tutorial assumes that the user has gone through Tutorials 1 and 2 and is familiar with the basic operation of iRIC.

Tutorial 3 steps:

- Construct a meandering channel with Channel Builder
- Run simulation and calibrate model with constant roughness.
- Visualize two- and three-d solution excluding streamline curvature
- Rerun simulation including streamline curvature
- Repeat steps above for a meandering channel with point bars
- Repeat steps above for a straight channel with alternate bars

Part A - Construct a Meandering Channel with Channel Builder

On the MD-SWMS menu, select **File -> New**, then select **Preprocessing -> Channel Builder**. The Channel Builder is a tool that constructs simple, idealized channels. Fill in the dialog as shown in Figure 3.2.3a.

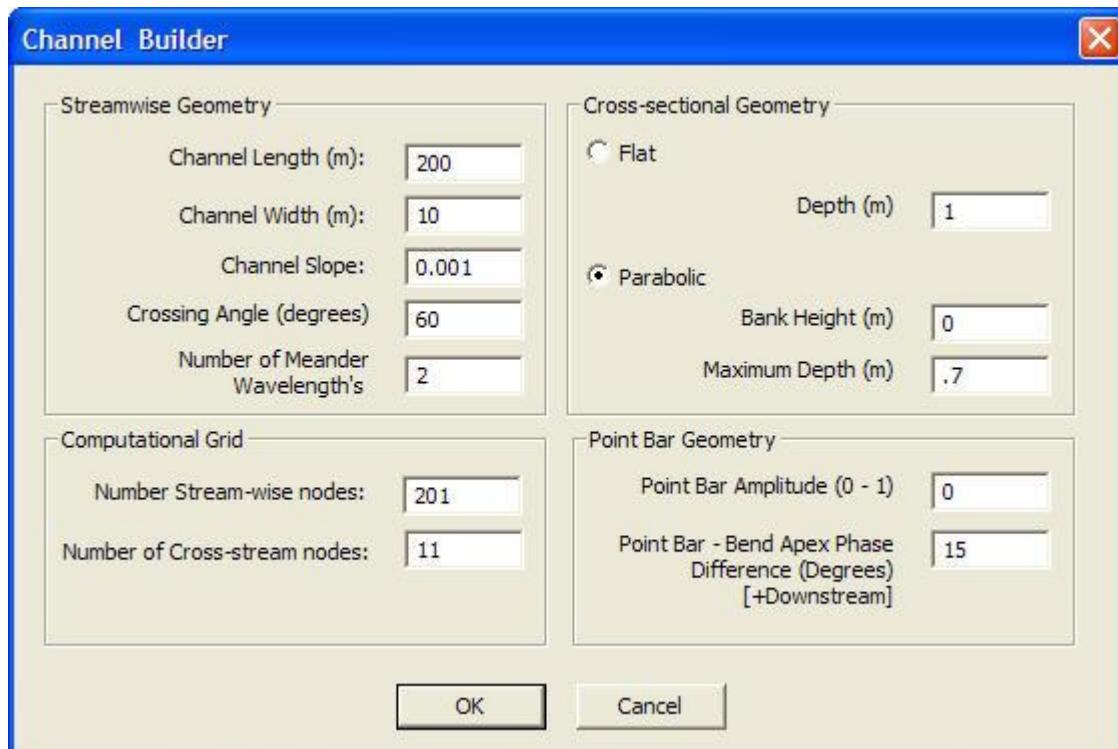
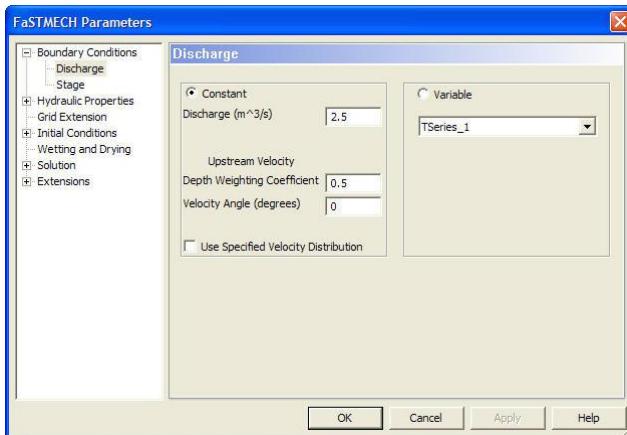


Figure 3.2.3a. Channel Builder dialog

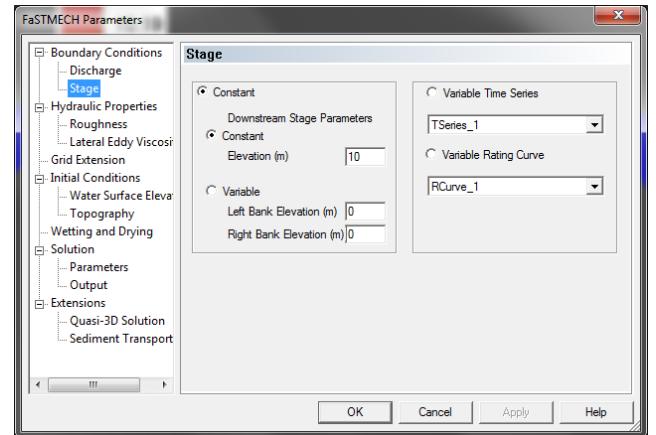
Visualize the resulting topography by selecting **Grid and Input Conditions / GIC Scalar Sets**. If you wish, you can visualize the grid by selecting **Create Grid**. Use the rotation, translation, and zoom tools to look at the channel; you may also want to look at it in the 3d viewer by selecting **View -> Examiner Viewer** on the iRIC menu. Remember that you can change the vertical scale and the light angle on the View menu in order to improve the visualization of the channel form. Once you are comfortable with these tools and understand this simple channel form, select **File -> Save** and save the .riv file under the tutorial 3 folder (renaming it however you like), then move on to the next step.

Part B – Run Simulation and Calibrate Model with Constant Roughness

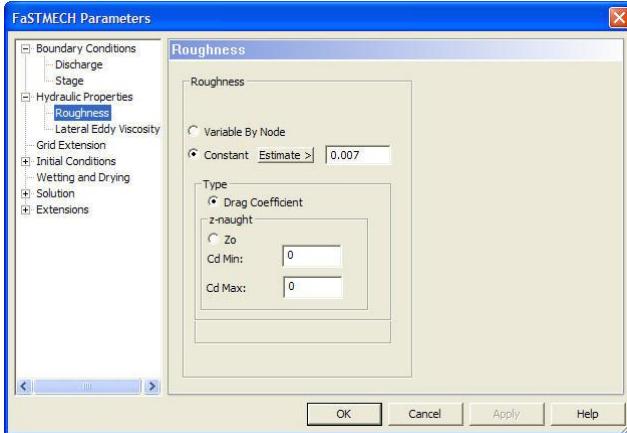
The next step is to run a flow simulation in the channel created above and iteratively correct the drag coefficient until the flow is close to reach-scale spatial uniformity over the 4 meander bends. Start by selecting **FaSTMECH -> New Simulation** and create a unique name for the simulation (remember, each .riv file can have a number of simulations associated with it). Next, select **FaSTMECH -> Edit Input File** and fill in the input dialogs as shown in Figure 3.2.3b. Note that streamline curvature effects are turned off in this simulation.



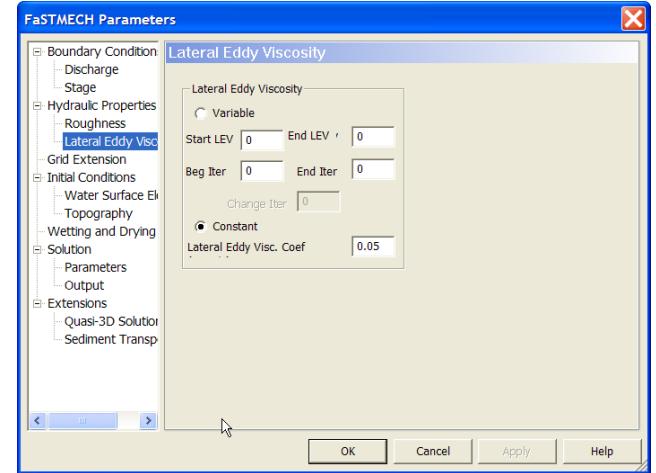
A



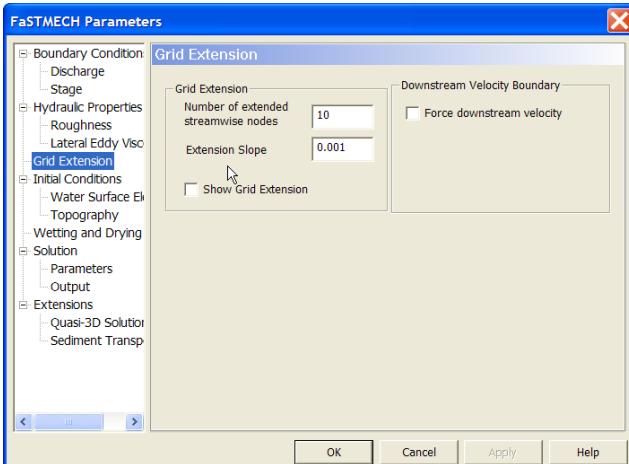
B



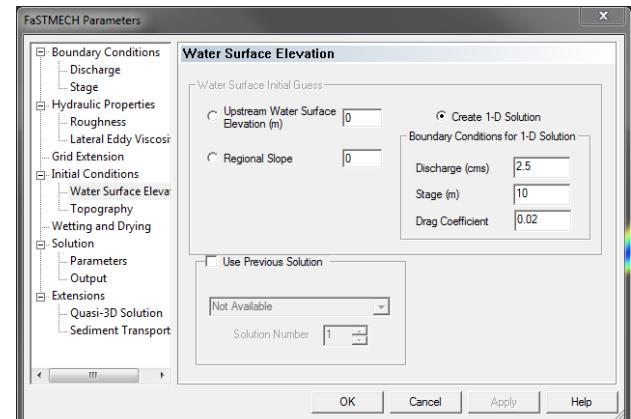
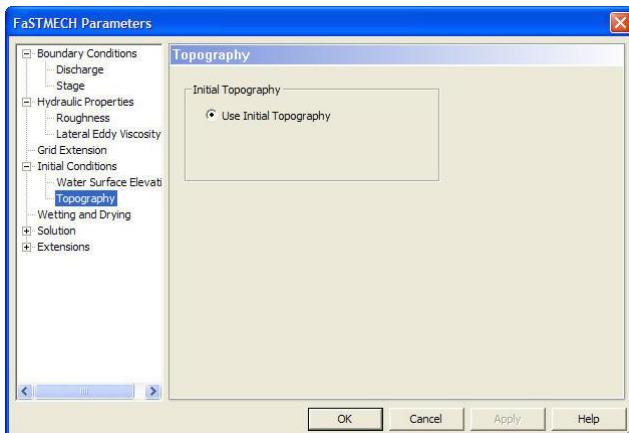
C



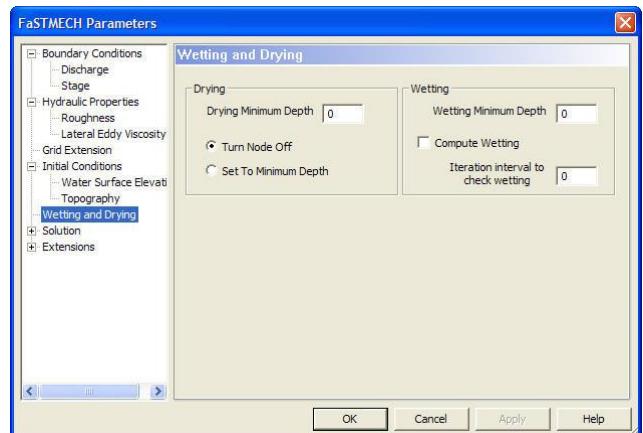
D



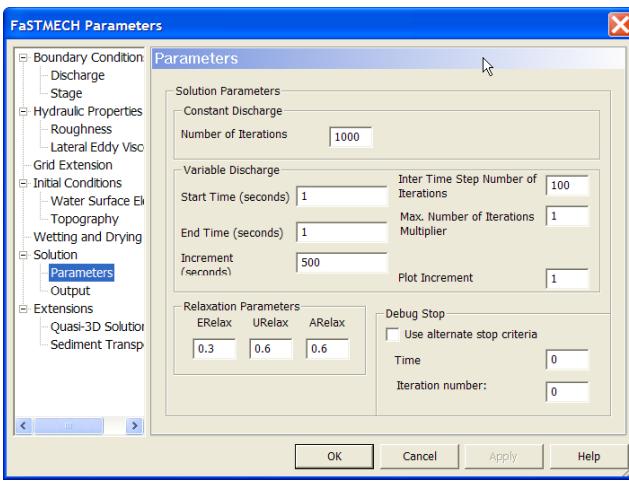
E

F Note the Create 1-D Solution option is selected

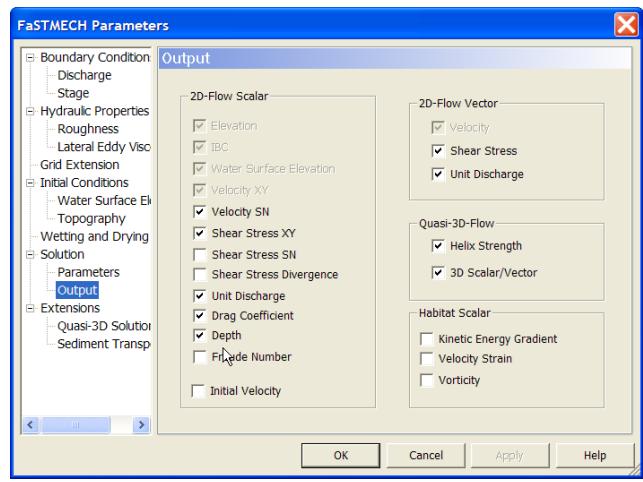
G



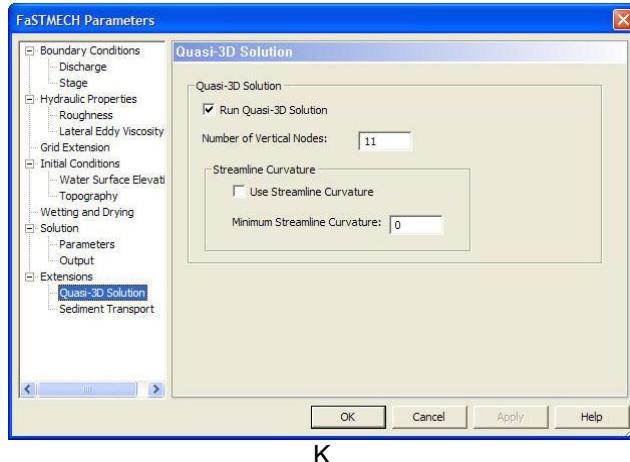
H



I



J



K

Figure 3.2.3b. Enter the following parameters into the FaSTMECH Parameters dialog. (A) Discharge, (B) Stage, (C) Roughness, (D) Lateral Eddy Viscosity, (E) Grid Extension, (F) Water-Surface Elevation, (G) Topography, (H) Wetting and Drying, (I) Solution Properties, (J) Solution Output, (K) Quasi-3D Solution

Once you've filled in the input file dialogs in accordance with figure 3.2.3b, select **FaSTMECH -> Run** on the iRIC menu. After the model runs (should take only a few seconds), select the 2D viewer and turn on the **2D solution** on the control bar. Using what you've learned about visualizing results, view the water-surface elevation and depth scalars. Note that for the chosen roughness value, the depth increases through the reach. This indicates that the flow is not uniform on a reach scale, i.e., the water surface is not parallel to the bed, on the average. In situations where surface elevation or other measurements are unavailable, observations of long-reach water-surface elevations may be suitable for calibrating roughness. In this case, the choice of discharge, lower boundary condition and roughness results in the flow going faster at the upstream end than the downstream one. In some natural situations, this can be a real effect, but in this idealized case, we want flow to be uniform on a reach scale; this means the drag coefficient must be higher than that given. The next step in the tutorial is to iteratively correct the value of the drag coefficient to achieve reach uniformity. Before doing this, view the helix strength scalar and use the probe or a suitable legend to determine the maximum/minimum helix strength in the bends. The helix strength 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. These flows are caused only by channel curvature for the case investigated. Don't be fooled by locally high values downstream of bends or bars associated with local areas of flow separation, in this case we are interested in the helix strength of the channel thalweg.

Now iteratively change the drag coefficient by selecting **FaSTMECH -> Edit Input File**, then selecting Hydraulic Properties | Roughness (fig. 3.2.3b (C)); change the drag coefficient here and under the Initial Conditions | Water Surface Elevation (fig. 3.2.3b (G)) and rerun the model simulation (it is not necessary to save these simulations under different names; you can save the final simulation when you have the appropriate value of the drag coefficient figured out). For each value of the drag coefficient you run a simulation, check the solution depth values to see if you have a reach-scale uniform condition and also look at the helix strength in the bends to see how it changes with roughness.

Part C – Visualize Two- and Three-D Solution Excluding Streamline Curvature

Once you have a calibrated value of drag coefficient you are happy with, examine the solution in detail using the plane viewer. Look at the final water-surface elevation, velocity vectors, bed stress vectors and so forth. Make a good plot of the helix strength with a legend and export it to your desktop for future comparison to other runs. Make a plot of the 3D and 2D vector fields as show in Figure 3.2.3c. A view of the 3D cross-section like that in Figure 3.2.3d can be viewed by selecting **View->3D Solution Cross Section**. Using your observations and various plots, try to answer the following questions:

1. How does the strength of secondary flow change with increasing roughness?
2. What is the phasing and direction of secondary flow relative to the channel curvature?
3. How do the cross-stream water surface slopes compare to the streamwise values?
4. What is the direction of the bed stress relative to the velocity vectors?
5. Why does the velocity tend to be higher on the inner bend side of the thalweg?

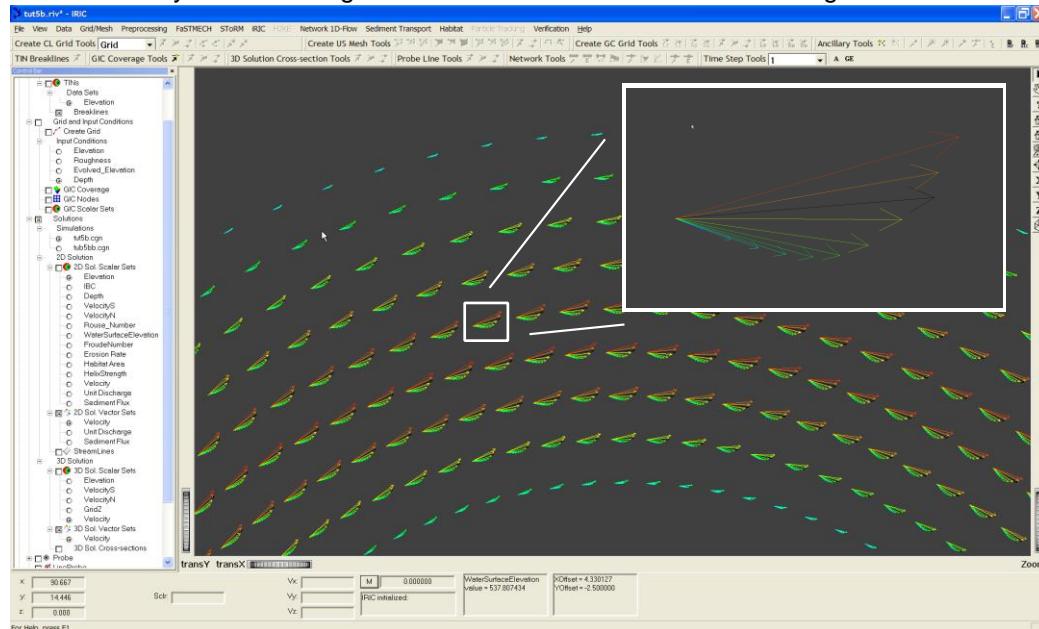


Figure 3.2.3c. 2D-velocity vectors shown in black and 3D velocity vectors with color mapping. An inset showing the detail at a single node is also shown. To make the figure turn on the 2D Sol. Vector Set and the 3D Sol. Vector Set in the Solutions branch of the Control Bar. Also make sure the viewer type button on the right hand side of the graphics viewer is set to Cartesian (looks like a square rather than a parallelogram). Zoom into one bend of the channel. Set the both the 2D and 3D Sol. Vector Attributes so that the body length scale is set to 1. Set the 2D Sol Vector Set color to constant and set the constant color to black. The vectors illustrate the secondary flow with outward flow at the surface and inward flow at the bed.

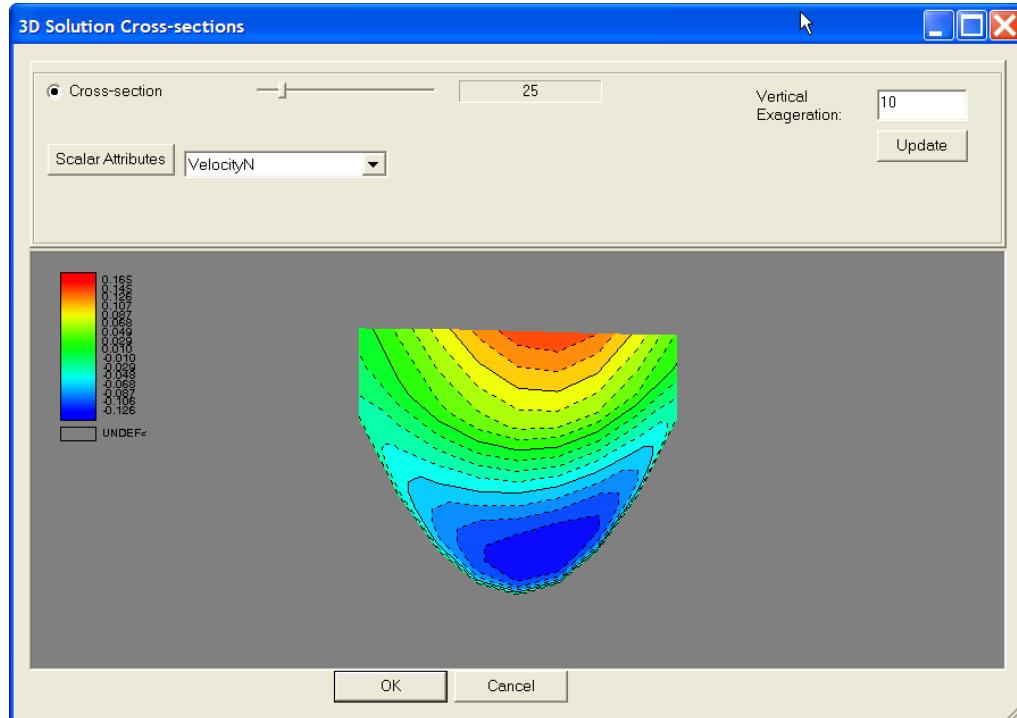


Figure 3.2.3c. The 3D Cross-Section Viewer. The Scalar value is set to VelocityN which is the cross-stream component of velocity in the curvilinear grid coordinate system. The cross-section number is 25, which is the apex of the first bend. The coordinate system is right handed with positive (outward) flow at the surface and negative (inward) flow at the bed.

Part D – Rerun Simulation Including Streamline Curvature

In the runs above, the curvature of the flow that gave rise to secondary flows was characterized only by the curvature of the channel. Although this is a good approximation in strongly curved channels like the one used above, it is not always appropriate. Furthermore, even for the case of simple curvature, the curvature of the actual flow streamlines will give slightly different results than that found from the channel curvature alone. In order to investigate this, save your existing model run then select **FaSTMECH -> New Simulation**. Give the simulation a new name, different from the simulation you just completed above. Note that you are still working on the same project (.riv) file, you are simply adding another simulation. Select **FaSTMECH -> Edit Input File** and enter the same values as shown in figure 3.2.3b with the exception of the **Extensions / Quasi 3D Solution** input. Edit the **Quasi 3D Solution** dialog to turn on streamline curvature, setting the minimum radius of curvature to 2 meters. Select **FaSTMECH -> Run** to run the model. Visualize the solution results, noting that you can toggle back and forth between this and the earlier solution excluding streamline curvature on the control bar. Compare the helix strength plots, paying particular attention to the magnitude and locus of the secondary flow near the apex of each of the bends. Try to answer the following questions.

1. How does the pattern of secondary flow strength change?
2. How does the maximum value of helix strength change?
3. Can you explain why these changes occur in terms of the flow in the bend?
4. Why do the first and third bends look different?
5. How might the inclusion of streamline curvature change the location of the point bar?

Part E – Repeat Steps A-D for a Meandering Channel with Point Bars

After completing steps A-D, students should have a good understanding of what vertical structure and secondary flows look like in simple channel bends. To extend this understanding to a slightly more realistic case, in this section a point bar will be added to the same bend used above, and the effect of that addition will be explored.

Save your existing model run, then select **File -> New**, then select **Preprocessing -> Channel Builder**. Fill in the dialog as shown in Figure 3.2.3a with the exception of the value for the point bar amplitude, which you should set to 0.5. This will add a simple low-amplitude point bar to the curved channel used above. As above, look over the new topography and then prepare a input file using the values given in Figure 3.2.3b with the exception of the drag coefficient, which you should set to the value you determined in B above. Run the model and look at the solutions. Is the reach-scale flow uniform? If not, adjust the drag coefficient as in B, above, until it is approximately uniform. Again, investigate the flow solution using the graphics tools. Save your results and then rerun the model with the streamline curvature turned on. Again, compare the solutions with and without streamline curvature effects (some example plots are shown in Figures 3.2.3e and 3.2.3f). Try to answer the following questions:

1. How does the presence of low-amplitude point bars affect the solution?
2. How does the helix strength change when streamline curvature is added?
3. How does the helix strength change relative to the case without bars?
4. How does the spatial evolution of the flow alter the flow in the third bend relative to the first one?

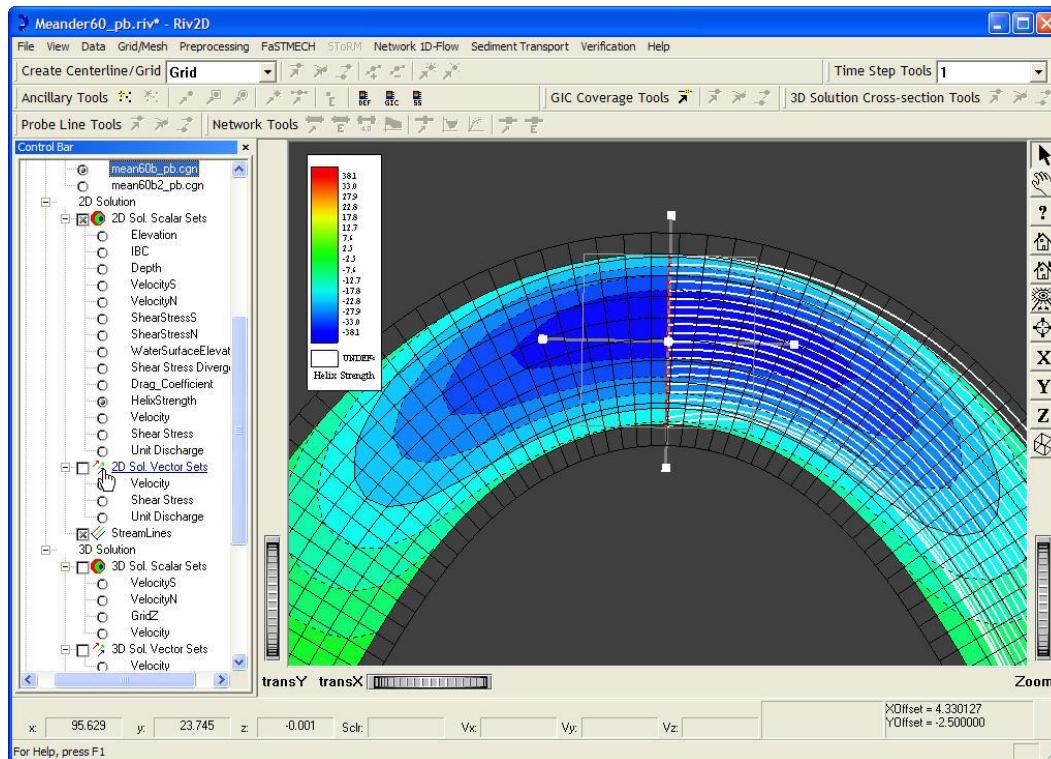


Figure 3.2.3e. Example image of Helix Strength (with no streamline curvature) plotted with streamlines of velocity in white and the grid lines in black.

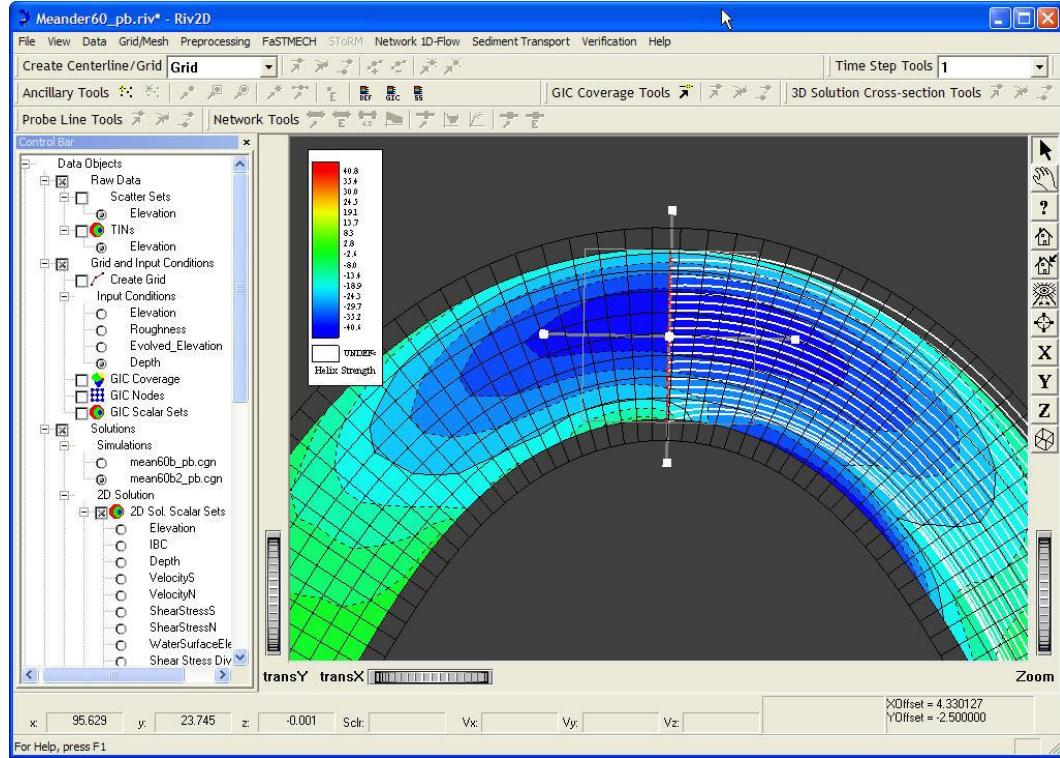


Figure 3.2.3e. Example image of Helix Strength (with streamline curvature) plotted with streamline of velocity in white and the grid lines in black.

Part F – Repeat Steps A-D for a Straight Channel with Alternate Bars

Save your existing model run, the select **File -> New**, then select **Preprocessing -> Channel Builder**. Fill in the dialog as shown in Figure 3.2.3a with the exception of the value for the point bar amplitude, which you should set to 0.5, and the value of the crossing angle, which you should set to 0. This will generate a straight channel with simple low-amplitude alternate bars. As above, look over the new topography and then prepare an input file using the values given in Figure 3.2.3b with the exception of the drag coefficient, which you should set to the value you determined in B above. Run the model and look at the solutions. Is the reach-scale flow uniform? If not, adjust the drag coefficient as in B, above, until it is approximately uniform. Again, investigate the flow solution using the graphics tools. Save your results and then rerun the model with the streamline curvature turned on. Again, compare the solutions with and without streamline curvature effects (some example plots are shown in Figures 3.2.3g and 3.2.3h). Try to answer the following questions.

1. How does the amplitude of the secondary flow compare to that in the meandering channels above?
2. How does the helix strength change with the addition of streamline curvature?
3. Can you speculate on how the secondary flow might affect the growth and shape of alternate bars?
4. Looking in more general terms at the pattern of secondary flow produced by streamline curvature around low-amplitude alternate bars, can you infer anything about the shape of islands in rivers?

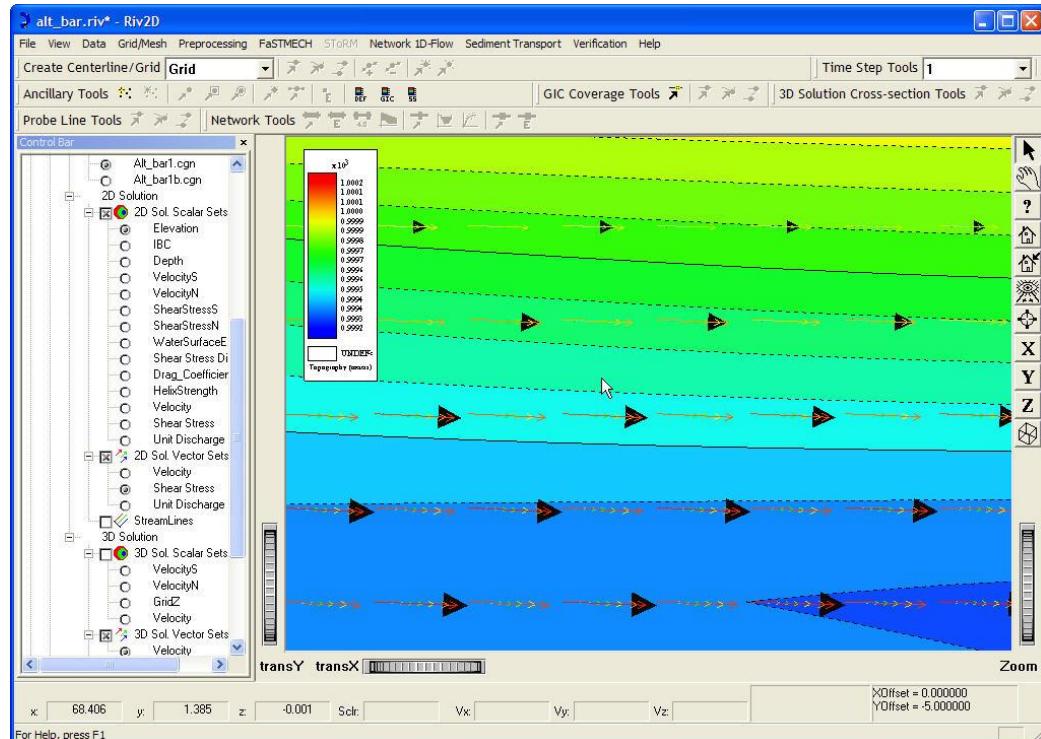


Figure 3.2.3g. Alternating bar and quasi-3d velocity without streamline curvature. Close-up of alternating bar topography contours at a location just upstream of a bar which is located at the top of the figure and the channel at the bottom. In addition, 3d-velocity vectors (colored) are shown along with the bed shear stress vectors (black).

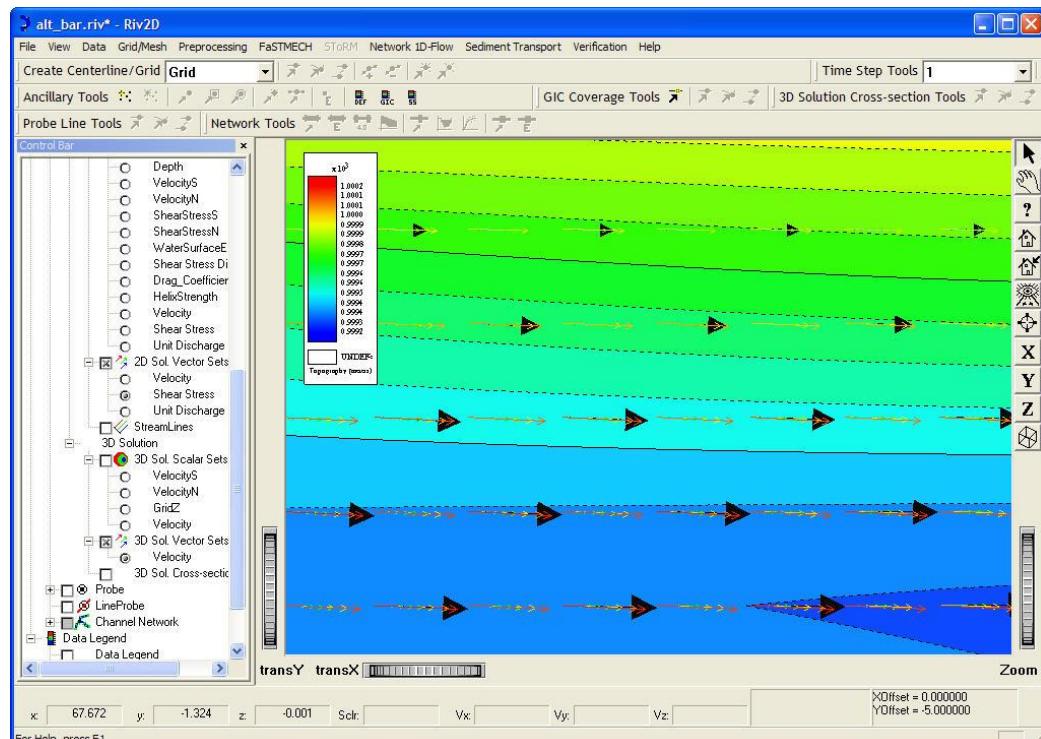


Figure 3.2.3h. Alternating bar and quasi-3d velocity with streamline curvature. See figure 3.2.3g for details of the figure.

3.2.4 FaSTMECH Tutorial 4 – 3D model in a natural channel

From this point on, the tutorials will assume students know how to construct and run models using real data sets or the Channel Builder and also have a good grasp of graphics tools in iRIC. To review this information in a realistic situation, this tutorial concentrates on a simple 3-D application of iRIC and FaSTMECH. The problem considered here is a bed material mobility determination for a short reach of the Kootenai River, Idaho, so this tutorial will also introduce the beginnings of sediment-transport computations. In addition, this tutorial will illustrate the importance of upstream velocity boundary conditions for short reaches. The steps in this tutorial are as follows:

Tutorial 4 steps:

- Create model grid and mapped topography for Kootenai reach
- Create a sequence of simulations in the Kootenai reach to determine sediment mobility at selected points in the channel bend
- Repeat each simulation using a measured velocity profile at the upstream boundary condition.

Part A - Create model grid and mapped topography for Kootenai reach

Start a new modeling project under the Tutorial 4 folder. Import the topography for the Kootenai reach (Koot.tpo). In addition a geo-referenced image is provided. Import the image to use as a background to your model results by selecting **File -> Import -> Ancillary Data -> Image** from the menu and open the file doc_all_small.jpg from the Tutorial 4 folder. Create a centerline (flow here is from bottom to top) and a suitable model grid with an approximate grid spacing in the downstream and cross-stream directions of 10 meters. Map the topography on to the grid using the TIN method. An example grid and topography is shown in Figure 3.2.4a, although your own grid need not look identical. Be sure that the grid lines do not overlap near the inside apex of the bends.

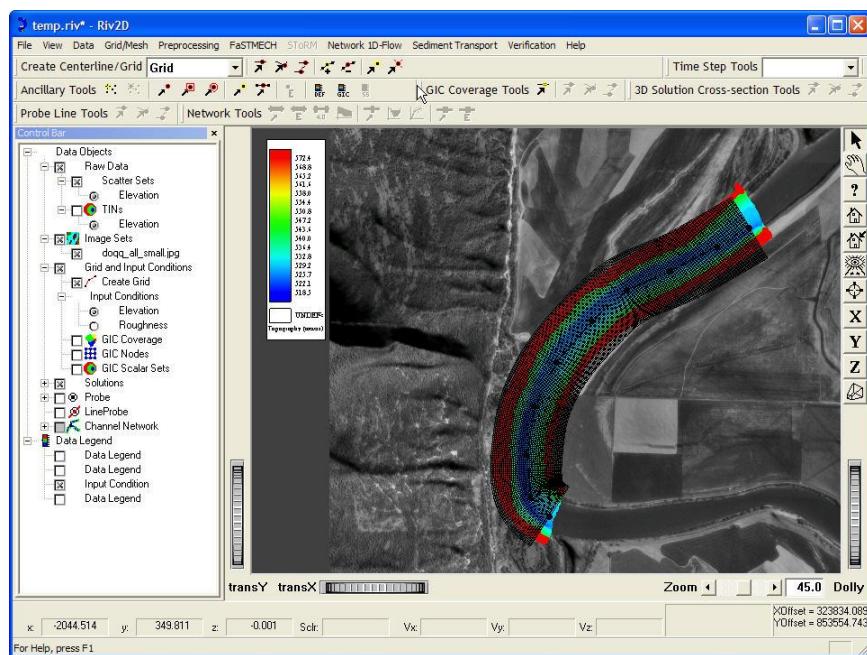


Figure 3.2.4a Model grid and mapped topography with aerial photograph in background.

Part B - Calibrate the model over a range of flows

Once a grid is developed and the topography mapped to it, create a new FaSTMECH simulation. The next step is to create an associated input file. Rather than providing a graphic image of that input file here, only the governing physical parameters are prescribed along with some suggestions. The goal of the simulations is to discover the approximate discharge at which material of a given size will begin to move in a specific location in the reach. Thus, the project will involve running the model at a range of discharges, calibrating each for roughness based on ancillary water-surface information. Typically, calibration data for roughness are provided either by measurements or by information from a longer-reach one-dimensional model. In this case, nine water-surface elevation profiles, representing flows that range from 6000 - 75,000 cubic feet per second (cfs), are provided from 1d simulations of the Kootenai River spanning the reach of interest. The files are formatted to be read as ancillary data into iRIC and can be found in the Tutorial 4 folder. The naming convention for these files identifies the associated discharge for example: CalibratedPts6Kcdf.anc where the 6K represents a discharge of 6000 cfs. Note: FaSTMECH requires all values to be entered in MKS units so you will have to convert the discharge from cubic feet per second to cubic meters per second.

In summary you will perform the following steps (Using **File -> Save** often):

1. Import the topography, a background image and a water-surface elevation file for calibration. Start with the lowest discharge first.
2. Create a model grid by first drawing the centerline and then building a grid with nodes approximately 10 meters apart.
3. Using the Ancillary Tools select individual water-surface elevation points to determine the downstream stage boundary condition for your model run
4. Create a new input file, set the input parameters and run FaSTMECH.
5. Verify the model calibration using the simulated 1d water-surface elevations. Iterate if necessary until you have a calibrated model.

Because you will need a different water-surface elevation ancillary file for each simulation you might find it convenient to create a new project for each discharge simulated. An easy way to do this is to save each completed simulation first by saving itself with the **File -> Save** command and then by using the **File -> Save As** and saving it to a new folder. The Save As command save the whole project including the simulation files into a new folder. Before you calibrate to a new discharge first calculate the mobility of the sediment as illustrated below.

Create a Grainsize input condition

In order to test the sediment mobility using iRIC you will first need to define a grainsize input condition. To do this, first create a grainsize input condition by right clicking **Grid and Input Condition / Input Condition** and selecting **Create Input Condition -> Grainsize_d50** in the pop-up menu. Set the default value as 0.0022. Finally create a coverage region as in Figure 3.2.4b that encompasses the entire grid by using the GIC Coverage Tools toolbar. Using the create new Coverage Region Polygon tool draw a polygon around the entire computational grid by left clicking to define the points of the polygon and selecting Enter to finish. Define the grainsize associated with the polygon by first expanding the **GIC Coverage Region** branch of the Control Bar followed by right clicking on the **GIC Coverage Region / Region_0** and selecting **Properties** from the resulting pop-up menu and entering 0.0022. Therefore the value at each node defined within the polygon is 0.0022.

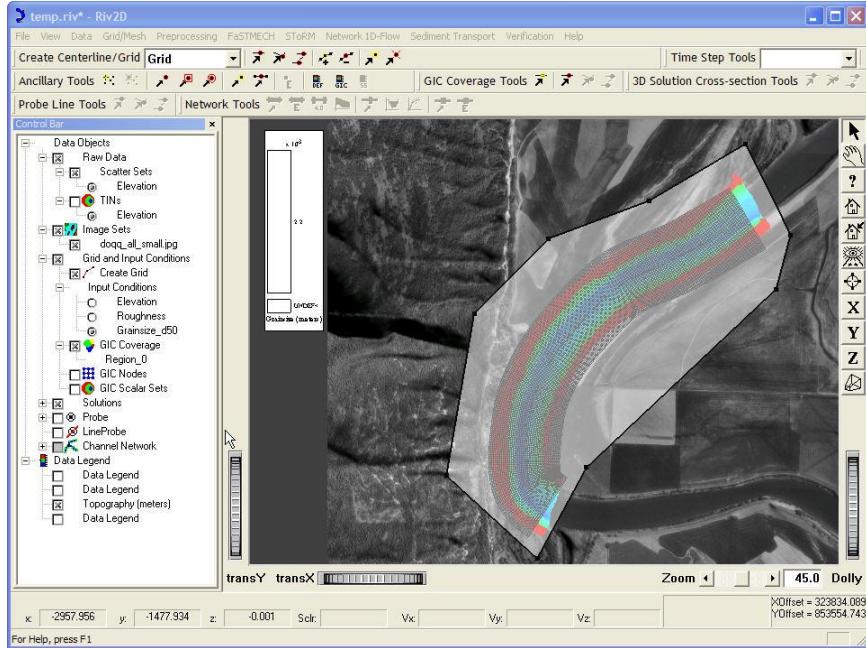


Figure 3.2.4b GIC Coverage defined for the Grainsize_d50 input condition.

Determine the Sediment Mobility

Additional relevant information required for your analysis of sediment mobility are that the median grainsize through the reach is 2.2 mm. Also a series of longitudinal topographical profiles through the reach (table 3.2.4a) illustrate the change in bedform geometries through a range of flows. The region of interest in this study is the channel thalweg in the first meander bend in the region of topography less than 522 meters in elevation. At each calibrated flow test the mobility of the bed using the **Sediment Transport -> Calculate Sediment Transport Values** dialog. An example of the dialog is shown below in Figure 3.2.4c.

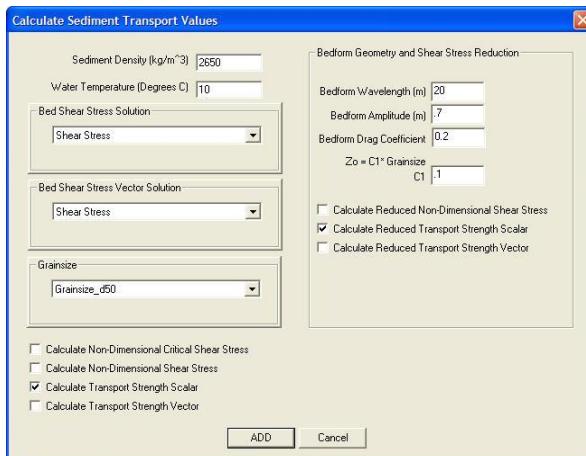


Figure 3.2.4c The Calculate Sediment Transport Values dialog.

Because of the large bedforms present at lower discharges you will want to calculate the reduced shear stress or grain shear stress by subtracting the fraction of the form drag due to the dune bedforms. Mobility is calculated as the Transport Strength defined by $(Tb - Tc)/Tc$, where Tb is either the bed shear stress or grain stress for the Transport Strength and Reduced Transport Strength respectively and Tc is the critical shear stress. Values greater than zero

represents potential transport. With each calibrated flow calculate both the Transport Strength and Reduced Transport Strength. These values are added to the **2d Sol.Scalar Sets** field in the Control Bar but are not saved and will be lost if the model is re-run or if the project is closed. Therefore, to keep track of your results for each discharge make whatever plot you feel is necessary and save with the **File -> Save Special** (figure 3.2.4d). In summary, to calculate the sediment mobility you will complete the following steps:

1. For the first project you create you will define a Grainsize_d50 Input Condition as described above. You will create the input condition only for the first project because after you save the project you will use **File -> Save As** to save the project into a new folder for each new discharge project.
2. Upon completion of the model calibration for each discharge calculate both the Transport Strength and Reduced Transport Strength Scalar fields.
3. Use the post-processing visualization tools to create a summary plot for review later and save using **File -> Save Special**

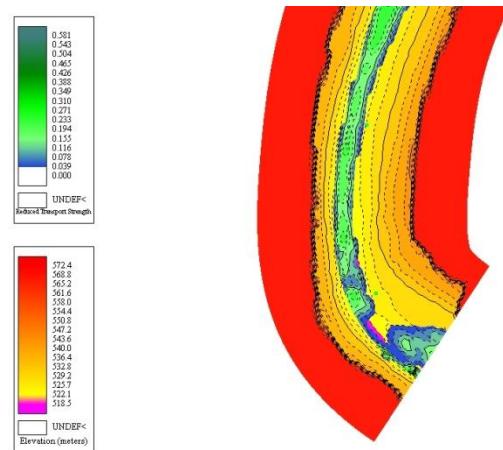


Figure 3.2.4d Plot of channel elevation and reduced transport strength. The region with elevation below 522 meters (the region in which you are evaluating the the sediment mobility) is highlighted in pink.

Date	Longitudinal profile F-F'					
	Streamflow		Average amplitude		Average wavelength	
	(m ³ /s)	(ft ³ /s)	(m)	(ft)	(m)	(ft)
05-24-2002	1,150	40,700	0.20	0.64	12.2	40.0
06-06-2002	1,110	39,100	.30	.99	14.0	46.1
07-23-2002	663	23,400	.51	1.68	18.9	62.0
09-04-2002	229	8,070	.69	2.27	20.4	67.0

Table 3.2.5a Bedform geometry surveyed over a range of flows in the model reach.

Part C - Repeat solutions using measured upstream velocity boundary condition

For each calibrated flow in Part B re-run the simulation using a prescribed upstream velocity boundary condition. From the menu select **Preprocessing -> Define Velocity Boundary Condition** and in the Velocity Boundary Condition dialog select the Import button. In the Open Velocity Boundary Condition Data dialog select the velbc3.txt file. The measured boundary condition can be viewed as in figure 3.2.4e. Select the OK button and save the project using **File -> Save**. Create a new input file and set all parameters the same as the original input file except the Discharge input condition, which is set to use the Specified Velocity Distribution as shown in figure 3.2.4e. Try to answer the following questions:

1. How does the measured velocity distribution affect your analysis of sediment mobility?
2. Over how many channel widths from the upstream boundary does it take the velocity solution, both with and without the specified velocity distribution, to become essentially independent of the upstream velocity assumption?

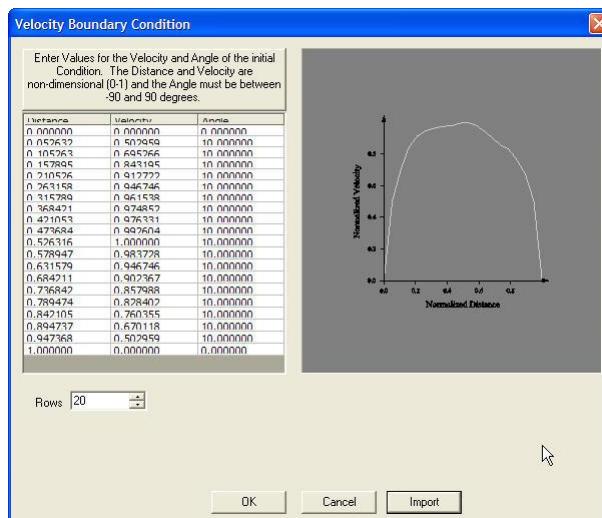


Figure 3.2.4e Velocity Boundary Condition dialog showing the input velocity boundary condition. Distance is from river right to river left and both distance and velocity magnitude are normalized to one.

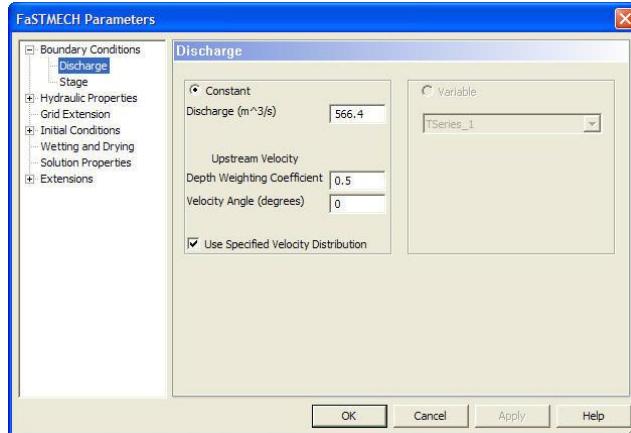


Figure 3.2.4f FaSTMECH Parameters input dialog for Discharge. To use the measured velocity boundary condition select the Use Specified Velocity Distribution check box.

3.2.5 FaSTMECH Tutorial 5 – Evolution of Point Bars During Constant Discharge in Simple Meandering Channel

This tutorial introduces simple time-dependent model runs using a constant discharge and the sediment-transport model extension. As in Tutorial 3 you will use the channel builder to create a simple meandering channel with a uniform parabolic cross-section. The simulation will be run for a short period of time and the evolution of point bars in the channel will be observed. In this tutorial you will perform the following steps:

Tutorial 5 Steps:

1. Using the Channel Builder, create a simple meandering channel with uniform cross-section
2. Create a time-dependent solution using a constant discharge and the sediment-transport model.
3. Observe the simulated channel evolution and development of point bars.
4. Repeat for different crossing angles or width-to-depth ratios

Create meandering channel

From the menu select **Preprocessing -> Channel Builder** and create a meandering channel with parameters as shown in Figure 3.2.5a.

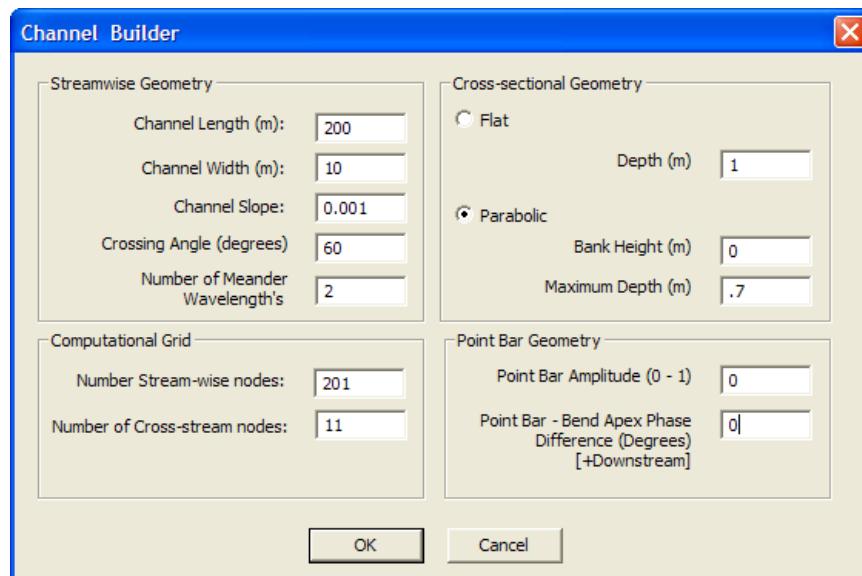


Figure 3.2.5a Channel Builder parameters for Tutorial 5.

Create time-dependent solution

Time-dependent solutions are developed by first running the model for the initial time-step with the Number of Iterations set to obtain a well converged solution. At each subsequent time-step the previous solution is used as the initial condition and therefore the number of iterations required is usually much less than the initial number, especially if the time step is chosen so that the change in the channel geometry is small. Create a new FaSTMECH simulation naming it as you like and then enter the following parameters in the input file and run the simulation.

- A constant discharge of 2.5 cms
- Downstream stage of 10.0 meters
- Constant drag coefficient of 0.008
- Lateral Eddy Viscosity of 0.05 m²/second
- 10 grid extensions with a slope of 0.001
- An initial water-surface elevation using the 1d-model. Enter the appropriate parameters.
- Enter the solution properties as in Figure 3.2.5b.
- Be sure to select the Solution Output you would like to visualize.
- The Quasi-3D solution extension on with 11 vertical nodes.
- Enter the Sediment Transport parameters as in Figure 3.2.5b.
 - We assume no bedforms in this solution so there is no bedform correction.
 - Use the Yalin bedload equation with a single pass of smoothing between each time-step.
 - Set the grain-size equal to 0.002 meters.
 - Calculate a gravitational correction using the gravitational pseudo stress technique

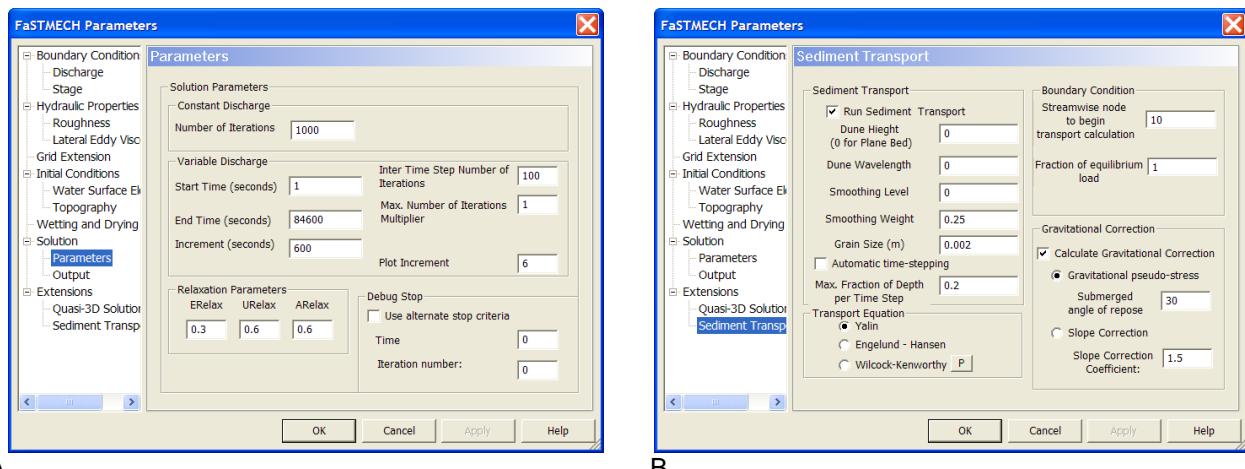


Figure 3.2.5b (A) Solution properties for time-dependent constant discharge simulation. Note that the Constant Discharge – Number of Iterations is the number of iterations the initial solution will be run after which the final solution will be used as the initial condition for all other time steps. The number of iterations for subsequent time steps is set as the Inter Time Step Number of Iterations in the Variable Discharge field. (B) Sediment transport properties.

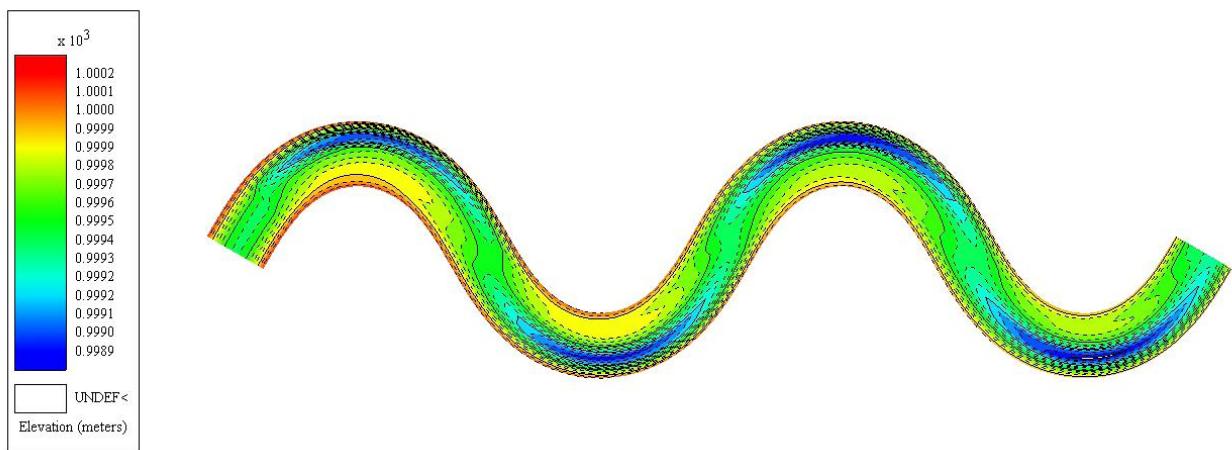
Observe point-bar evolution in time dependent solution

iRIC currently has a limited set of tools to view time-dependent solutions. The Time-Step Tools toolbar has a simple pull down menu to select the time-step to view. Click the pull down menu to select a time step to view. Note, at this time the color map is adjusted at each time-step. You can use the Data Mapping dialog to set a consistent color map for each time step and then use the **File -> Save Special** to export an image for the first and last time-step to create a figure something like that shown in Figure 3.2.5c.

Repeat for different crossing angles or width-to-depth ratios

Try to set up and run some other evolution cases with different crossing angles or other parameters. Pay attention to the roughness and how the water surface elevation changes as the bars form. Learning to model channel evolution requires careful attention to the flow solution and how bar drag changes the solution as the bed evolves. This is only more true as one moves on to predicting evolution of bed morphology in real channels.

A



B

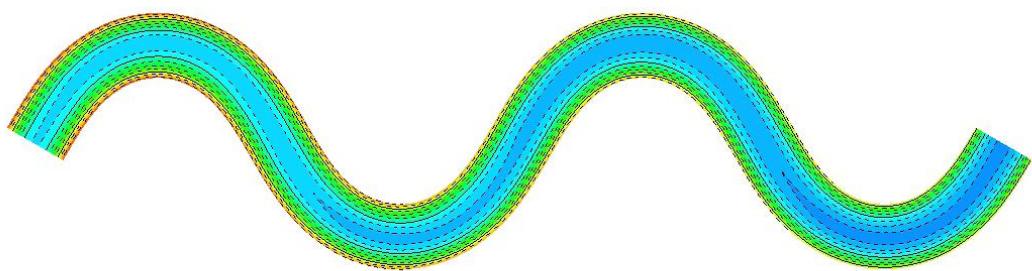


Figure 3.2.5c (A) Final topography and (B) initial topography

3.2.6 FaSTMECH Tutorial 6 – Evolution of Point Bars during Time-Dependent Discharge

This tutorial introduces variable discharge model runs. Using the simple meandering channel created in Tutorial 5 you will explore the effect of three different hydrographs (Figure 3.2.6a), each using the same total volume of water, on point bar evolution. In this tutorial you will perform the following steps:

Tutorial 6 Steps:

1. Save the project file created in Tutorial 5 into a new folder using the Save As command.
2. Import stage-discharge rating curve and two time series of discharge.
3. Create three time-dependent solutions using variable discharge time series, a stage-discharge rating curve, and the sediment-transport model extension.
4. Observe the effect of time-varying discharge on simulated channel evolution and development of point bars.

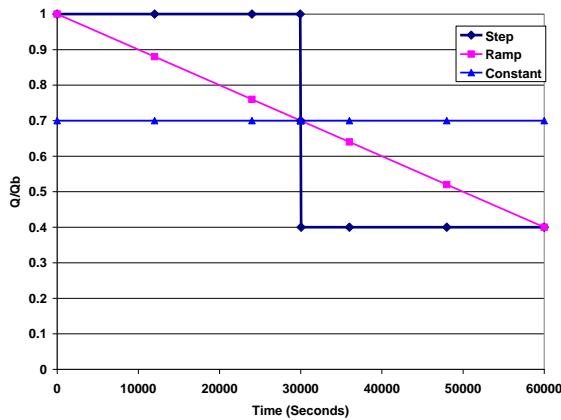


Figure 3.2.6a The three hydrographs used in Tutorial 6.

Open project file created in Tutorial 5 and save into a new folder

Open the project file created in Tutorial 5 and use the **File ->Save As** command to save into a new folder as Ramp_Q.riv. This will save both the .riv project file and the .cgn model I/O file into the new folder and is a simple way to build upon a pre-existing project.

Import stage-discharge rating curve and two time series of discharge

iRIC contains tools for creating rating curves and data time series. In this tutorial you will use a rating curve of the stage boundary condition and two discharge time series. From the menu select **Preprocessing->Define Rating Curve Boundary Condition**. In the Rating Curve Builder dialog select the Import Button and open the Stage-Discharge.txt file. A plot of the rating curve is shown in the Graphics Panel. Be sure the Boundary Condition option is set to Stage and set the name to Stage Rating as shown in Figure 3.2.6b.

To import the time series of discharge select **Preprocessing->Define Time-Series Boundary Condition**. In the Time Series Builder dialog select the Import button and open the Ramp_Q.txt file. In the dialog select the Discharge option and name the time series Ramp_Q as shown in Figure 3.2.6c. An additional time series can be imported by first selecting the New button to create a new time series, followed by the Import button and then opening the file

Step_Q.txt. Notice that the Which Time Series box in the dialog contains 2. You can move back and forth between the two time-series by selecting the up or down arrow buttons. Be sure that the Discharge option is selected and the Time Series Name is Step_Q.

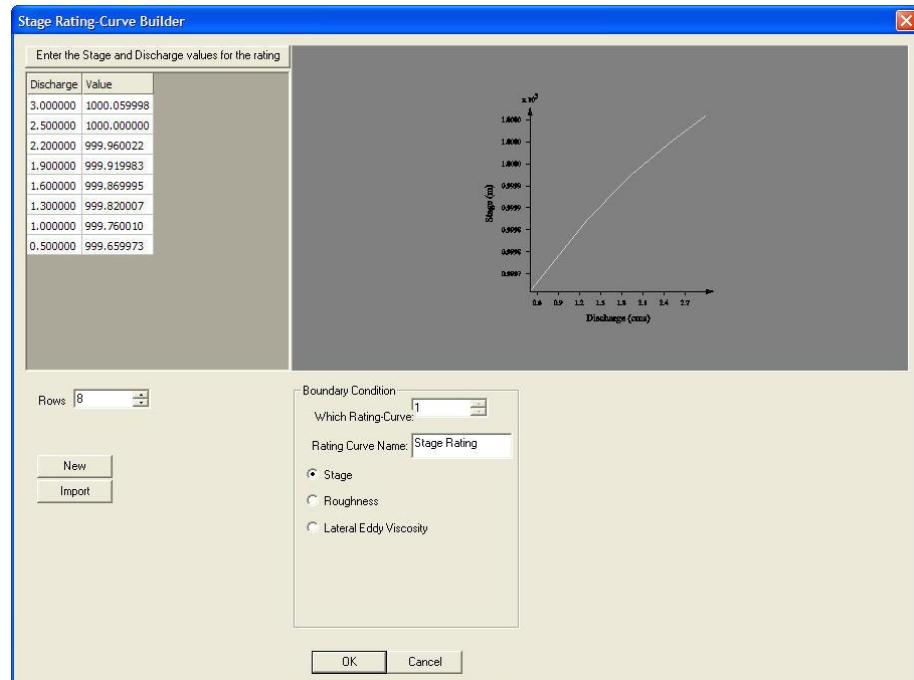


Figure 3.2.6b The Rating Curve Builder Dialog.

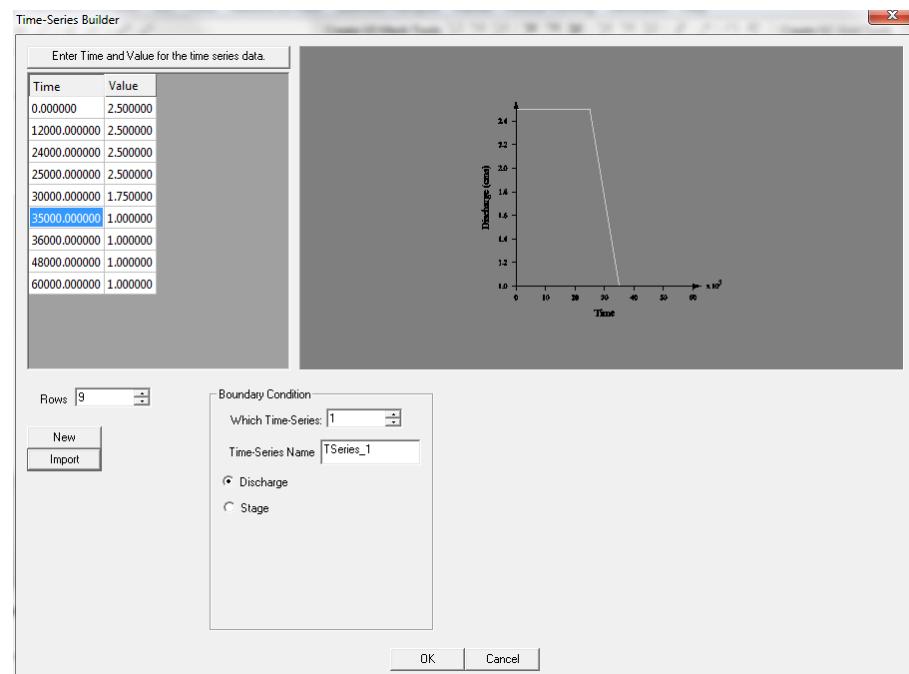


Figure 3.2.6c The Time-Series Builder Dialog.

Run channel evolution simulations using three different hydrographs

In this part of the tutorial you will run three simulations using the three hydrographs shown in Figure 3.2.6a. Each hydrograph represents the same volume of water. You will compare and contrast the resulting topography on the two middle meander bends. The results illustrate in a very simple way the potential to simulate the effect of flow magnitude, duration and hydrograph shape on the resulting channel topography.

Point bar evolution using a ramp time-series

From the menu select **FaSTMECH->Edit Input File**. Keep all parameters the same with the following exceptions as shown in Figure 3.2.6d.

1. Discharge: make sure to select the variable option and select the Ramp_Q time-series.
2. Stage: select the Variable Rating Curve option and select the Stage Rating curve.
3. FaSTMECH Parameters: Set the Max. Number of Iterations Multiplier to 1. Setting this value greater than 1 would change the number of Inter Time Step Number of Iterations, by a multiple of its value, when the number of active nodes changes due either to drying or wetting.
4. Sediment Transport. Calculate the Gravitational Correction using the Gravitational pseudo-stress.

Run the simulation.

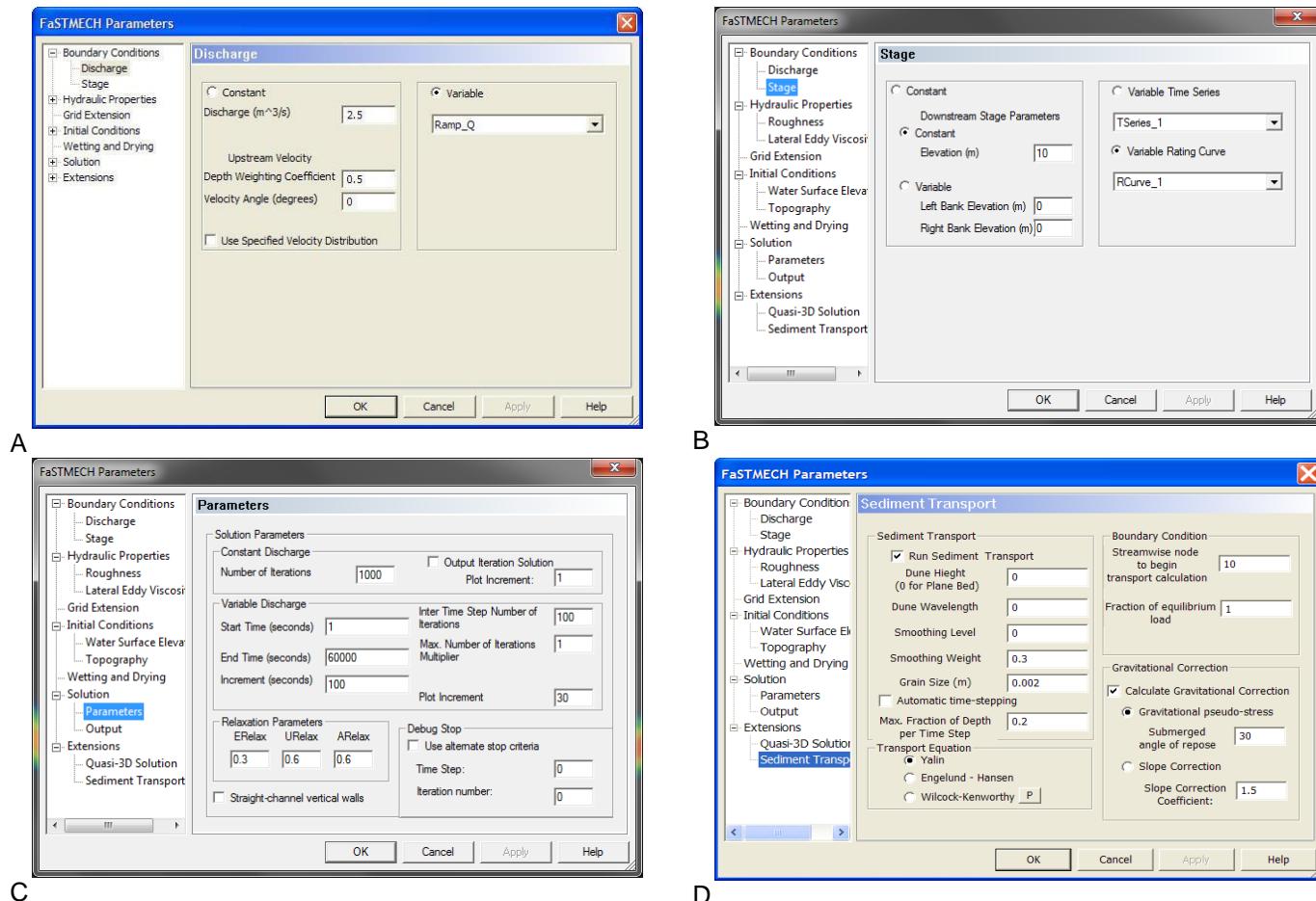


Figure 3.2.6d Enter the following parameters into the FaSTMECH Parameters dialog. (A) Discharge, (B) Stage, (C) Parameters, (D) Sediment Transport.

Point bar evolution using a step discharge time series

Save the previous project in a new folder using the **File->Save As** command. Name the file Step.riv. From the menu select **FaSTMECH->Edit Input File**. Keep all parameters the same with the following exception:

1. Discharge: make sure to select the variable option and select the Step_Q time-series.

Run the simulation.

Point bar evolution using a constant discharge

Save the previous project in a new folder using the **File->Save As** command. Name the file Step.riv. From the menu select **FaSTMECH->Edit Input File**. Keep all parameters the same with the following exception:

1. Discharge: make sure to select the constant option and enter 1.75 for the constant discharge.

Run the simulation.

Discussion

The topography of the two middle meander bends for each of the three hydrographs is shown in Figure 3.2.6e. For each figure of topography the elevation range was set to a minimum elevation of 9.15 and a maximum elevation of 10.19 so that the point bar topography can be compared. The difference in topography between each of the three runs is subtle, especially between the Ramp and Step hydrographs. Can you explain the differing topography in each of the three simulation as a function of the discharge history?

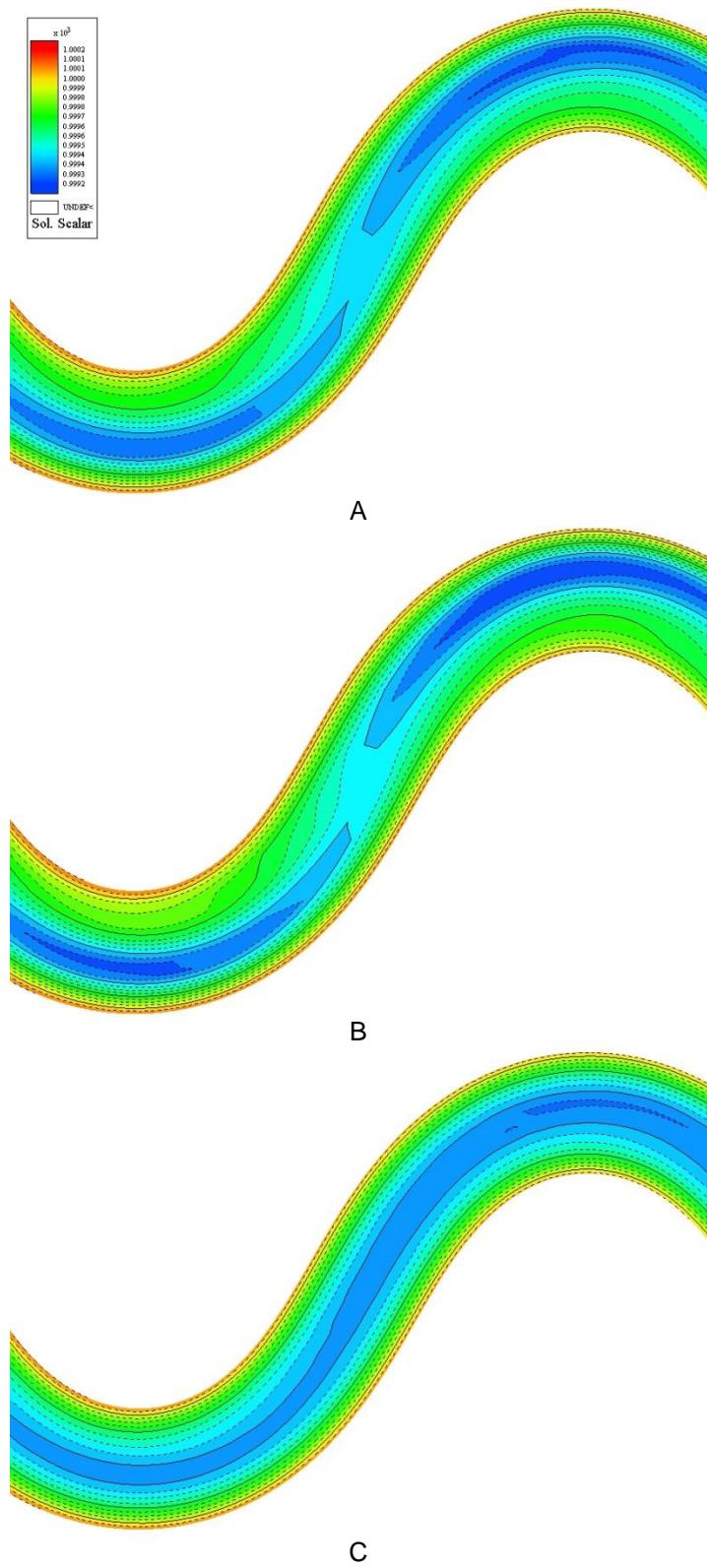


Figure 3.2.6e (A) Ramp Simulation, (B) Step Simulation, (C) Constant Simulation

3.2.7 FaSTMECH Tutorial 7 – Habitat Analysis

This tutorial assumes students have experience with iRIC through at least Tutorial 3. This means you should know how to construct and run models using real data sets or the Channel Builder and also have a good grasp of graphics tools in iRIC. To review this information in a realistic situation, this tutorial concentrates on a simple 2-D application of iRIC and FaSTMECH for habitat analysis. The problem considers the determination of habitat availability for a short reach of the Green River, Utah, so this tutorial will also introduce the Habitat Builder application within iRIC. The steps in this tutorial are as follows:

Tutorial 7 steps:

- Create a model grid and mapped topography for the Green River, UT
- Calibrate the model over a range of flows in the Green River using measured water surface elevations.
- Using the Habitat Builder calculate the Weighted Usable Area (WUA) for spawning and larval life stages of the Colorado Pikeminnow and develop a plot of WUA over the range of calibrated flows.

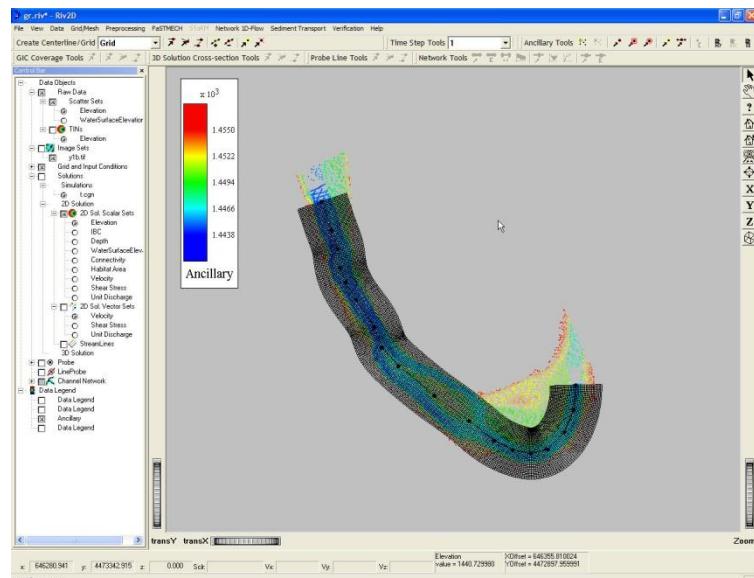


Figure 3.2.7a Model grid and mapped topography.

Part A - Create model grid and mapped topography for the Green River reach

Start a new modeling project under the iRIC Tutorials\FaSTMECH\Tutorial 7 folder. Import the topography for the Green River reach (GR.tpo). In addition a geo-referenced image is provided. Import the image to use as a background to your model results by selecting **File -> Import -> Ancillary Data -> Image** from the menu and open the file y1b.jpg from the Tutorial 7 folder. Create a centerline (flow here is from right to left) and a suitable model grid with an approximate grid spacing in the downstream and cross-stream directions of 10 meters. Map the topography on to the grid using the TIN method. An example grid and topography is shown in Figure 3.2.7a, although your own grid need not look identical. Be sure that the grid lines do not overlap near the inside apex of the bends.

Part B - Calibrate the model over a range of flows

Once a grid is developed and the topography mapped to it, create a new FaSTMECH simulation. The next step is to create an associated input file. Rather than providing a graphic image of that input file here, only the governing physical parameters are prescribed along with some suggestions. The goal of the simulations is to calibrate the model over a range of discharges. Thus, the project will involve running the model at a range of discharges, calibrating each for roughness based on ancillary water-surface information. Typically, calibration data for roughness are provided either by measurements or by information from a longer-reach one-dimensional model. In this case, nine water-surface elevation profiles, representing flows that range from 50 - 1000 cubic meters per. second (cms), are provided from measured water-surface elevations in the Green River spanning the reach of interest. The files are formatted to be read as ancillary data into iRIC and can be found in the Tutorial 7 folder. The naming convention for these files identifies the associated discharge, for example, the file wse_50.anc is water-surface elevation at a discharge of 50 cms. An example of the calibrated drag coefficients over a range of flows is shown in Figure 3.2.7b

In summary you will perform the following steps (Using **File -> Save** often as always):

6. Import the topography, a background image, and a water-surface elevation file for calibration. Start with the lowest discharge first.
7. Create a model grid by first drawing the centerline and then building a grid with nodes approximately 10 meters apart.
8. Using the Ancillary Tools select individual water-surface elevation points to determine the downstream stage boundary condition for your model run.
9. Create a new input file, set the input parameters and run FaSTMECH.
10. Verify the model calibration using the measured water-surface elevations. Iterate if necessary until you have a calibrated model. Figure 3.2.7b shows the calibration over the entire range of flows

Because you will need a different water-surface elevation ancillary file for each simulation you might find it convenient to create a new project for each discharge simulated. An easy way to do this is to save each completed simulation first by saving with the **File -> Save** command and then by using the **File -> Save As** and saving it to a new folder. The Save As command saves the whole project including the simulation files into a new folder.

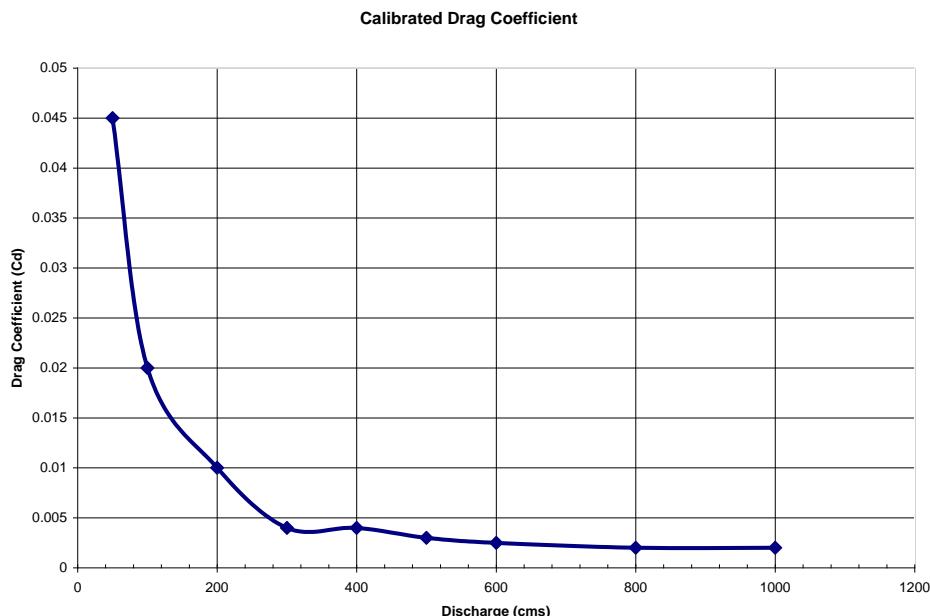


Figure 3.2.7 Calibrated drag coefficient for the Green River, Utah

Part C - Use the Habitat Builder application to calculate Weighted Usable Area

The Habitat Builder consists of a Builder dialog (Figure 3.2.7c) and one or more Predictor dialogs (Figure 3.2.7d). A function such as a user-defined power function, a threshold, or a user-defined curve is applied to each predictor variable to assess habitat suitability. These functions are known as Habitat Suitability Indices and are typically scaled from 0 – 1, where 1 is the most suitable and 0 is the least. The Habitat Builder dialog allows the user to define a habitat function made up of one or more predictors and to map the value of the habitat function to the model grid as well as calculate a Weighted Usable Area for the function. For example, the user might select the geometric mean of depth and velocity predictors as their habitat function. Predictors can be built from either solution scalars or Grid and Input Condition scalars, so any scalar value defined as an input condition (such as grainsize), could be used as a predictor. Additional ancillary data (such as cover) could be added to the model project and mapped to the input grid and used as a predictor. In this way there is some flexibility to allow addition of habitat predictors from sources other than predicted values from the model.

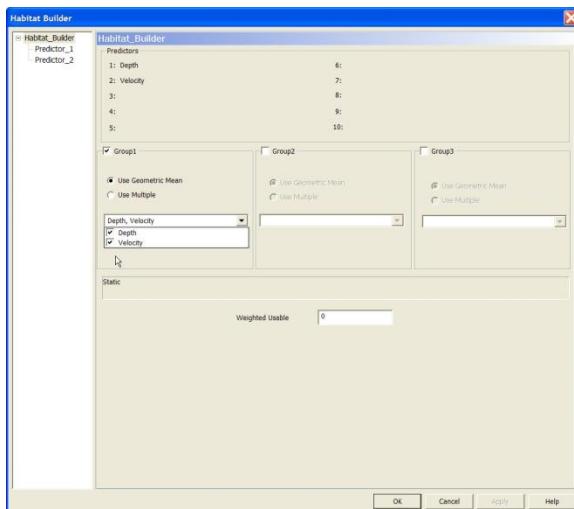


Figure 3.2.7c Habitat Builder dialog example

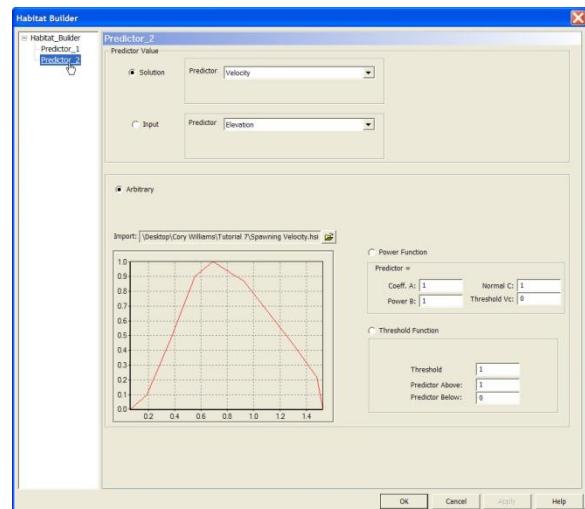


Figure 3.2.7d Habitat Predictor dialog example

This section introduces the Habitat Builder. The model calibrated solutions from Part B above are used to model habitat for two life stages of the Colorado Pikeminnow. Habitat Suitability Indices for velocity and depth collected for Colorado Pikeminnow from both the Green and Colorado Rivers are presented in Table 3.2.7a. These data are not necessarily applicable to the reach that is modeled here as they were collected in different parts of the Green River basin or they were collected, as the case for the spawning lifestage, in the Colorado River basin. These curves/indices are used simply to illustrate the methodology. For use with the Habitat Builder application these HSI curves have been included in the Tutorial 7 folder with descriptive filenames such as “Spawning_Velocity.hsi”, corresponding to the velocity HSI curve for the spawning lifestage.

Colorado Pikeminnow						
Depth	Spawning		Larvae			Suitability Index
	Suitability Index	Velocity	Suitability Index	Depth	Suitability Index	
0.00	0.00	0.00	0.00	0.00	0.00	1.00
0.16	0.00	0.06	0.00	0.04	0.00	0.48
0.31	0.28	0.19	0.10	0.15	0.38	0.01
0.47	0.82	0.37	0.48	0.30	0.92	0.12
0.55	0.95	0.55	0.90	0.38	1.00	0.03
0.62	1.00	0.69	1.00	0.60	0.92	0.03
0.70	0.93	0.83	0.92	1.05	0.73	0.05
0.78	0.88	0.92	0.87	1.50	0.58	0.06
0.94	0.70	1.11	0.66	1.80	0.47	0.01
1.09	0.51	1.29	0.45	2.10	0.36	0.09
1.25	0.32	1.48	0.22	2.40	0.23	
1.31	0.00	1.52	0.00	3.05	0.00	

Table 3.2.7 a Colorado Pikeminnow habitat Suitability Indices for both Spawning and Larval lifestages.

The habitat suitability will be calculated for each life stage using HSI curves for both depth and velocity. The habitat builder calculates the habitat suitability value at each node on the grid. For example, using the HSI curve of the spawning lifestage for velocity (Figure 3.2.7d and Table 3.2.7a) any node in the model grid with a value of 0.691 m/s would receive a habitat value of 1. The same process is applied to the values of depth and finally the calculated value of habitat is defined as the geometric mean of the depth and velocity values. The step-by-step guide the using the Habitat Builder is provided below.

Habitat Builder step-by-step guide:

1. Open an existing project with a calibrated solution for the 50 cms flow. Apply the steps below to each life stage separately. The process is illustrated using the larval lifestage from beginning to end. It will be left to the user to repeat these steps for the Spawning lifestage. The Habitat Builder is accessed from the menu by selecting **Habitat -> Habitat Builder**. As noted above the Habitat Builder consists of the Habitat Builder Dialog and one or more Habitat Predictor dialogs. The Control Bar to the left of the dialogs provides access to each dialog and functionality by either right or left clicking on any branch of the data-tree.
2. Define your first predictor; in this case, the predictor for depth. Using the mouse, left-click on the Habitat Builder | Predictor_1 branch of the data-tree to bring the Predictor_1 dialog forward.
 - a. The model predicted depth is used therefore, the Predictor Value box in the dialog should be set to the Solution option. Select "Depth" in the drop-down list associated with the Solution option.
 - b. Since HSI curves are used, select the "Arbitrary" option in the Prediction Method box. Select the brows button  to browse for the Larvae Depth.hsi file located in the Tutorial 7 folder. **Important Note: Selecting the more than one *.hsf file for each Habitat Predictor will cause errors. If you select more than one then simply start over by selecting Cancel in the Habitat Builder.**
 - c. The imported HSI curve should look like Figure 3.2.7e
3. Create a new Predictor for velocity. Right click the Habitat Builder brand of the data-tree and select Add Predictor in the resulting drop-down menu. You will now see the data-tree has Predictor_2 added. Left click Predictor_2 and repeat the steps in step 2 for the larvae velocity so that the Predictor_2 dialog resembles Figure 3.2.7f.
4. Return to the Habitat Builder dialog by left clicking the Habitat Builder branch of the data-tree (Figure 3.2.7g). In the Predictors box of the dialog, depth and velocity will be listed. To define habitat, predictors may be combined in one to three groups as defined by the user. In this case, one simple group of depth and velocity is used and the habitat is calculated as the geometric mean of the two quantities. To do this check the first group box which will enable the Geometric Mean and Multiple options as well as the drop-down list box. Select the Geometric Mean option and using the drop-down list box check both depth and velocity as in Figure 3.2.7h.
5. To calculate the Weighted Usable Area (WUA) of the habitat, right click on the Habitat Builder branch of the data-tree and select Calculate Habitat from the pull-down menu. This will add the calculated WUA to the dialog * and on exit of the habitat builder add to the model **2D Solution Scalar Sets** the Habitat Suitability scalar set. ***Take note of the value and write it down as you will use it to generate a plot of WUA for each life stage against Discharge at the end of this tutorial.**
 - a. To view the predicted Habitat Suitability, select the **2D. Sol. Scalar Sets / Habitat Value** from the Control Bar. A better view of the Habitat Value can be obtained by first creating a Data Legend of the 2D Solution Scalar Set. Double click on the **Data Legend / Sol. Scalar** to access the Data Mapping dialog. A quick change of the contour interval minimum value from 0 to 0.001 will make the presentation of the Habitat Value more informative. In the Contour Levels box of the Data Mapping dialog change the "min" value to 0.001 and then click the Calc. Levels button. Finish by exiting the Data Mapping dialog. The resulting Habitat Value plotted against the aerial photo should look something like that in Figure 3.2.7i.
6. Repeat these steps for the Spawning life stage, taking note of the calculated WUA.
7. Repeat for each discharge value and plot the resulting WUA as a function of discharge for each life stage as in Figure 3.2.7j.

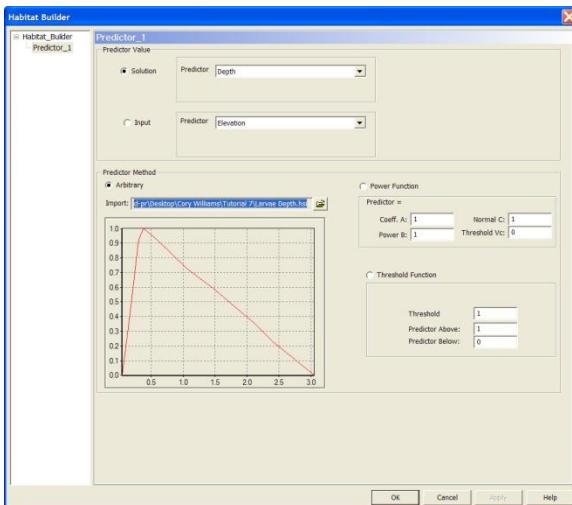


Figure 3.2.7e Depth predictor for the larval Colorado pikeminnow

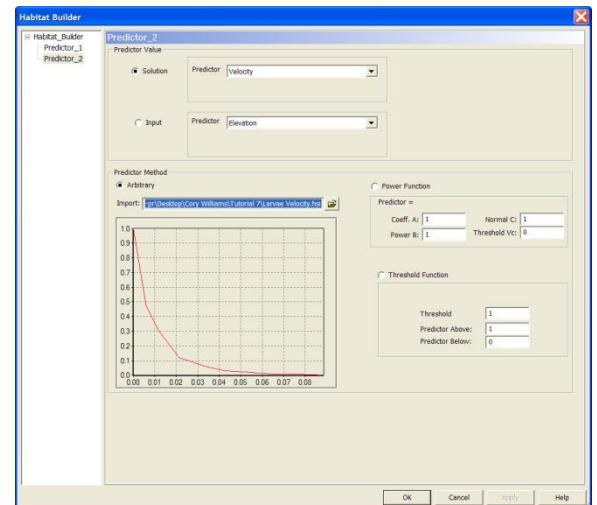


Figure 3.2.7f Velocity predictor for larval Colorado pikeminnow

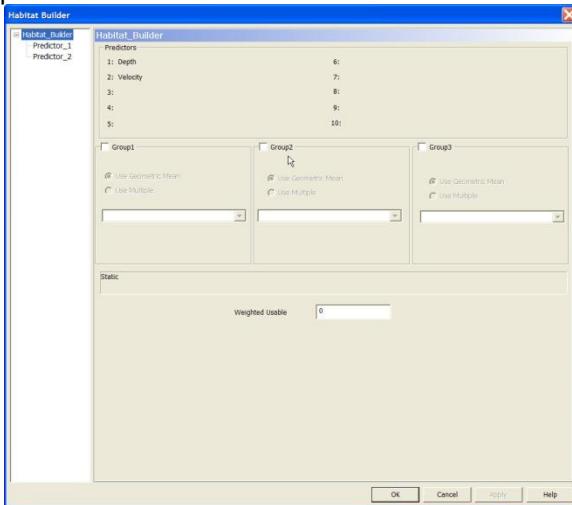


Figure 3.2.7g Habitat Builder dialog following the addition of depth and velocity predictors

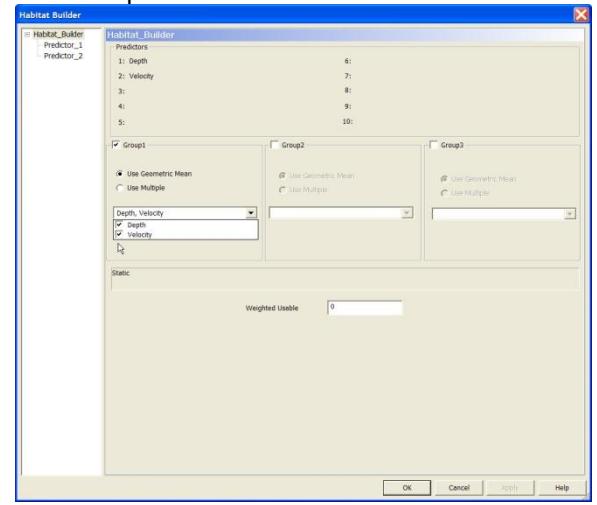


Figure 3.2.7h Habitat Builder dialog showing selection of depth and velocity

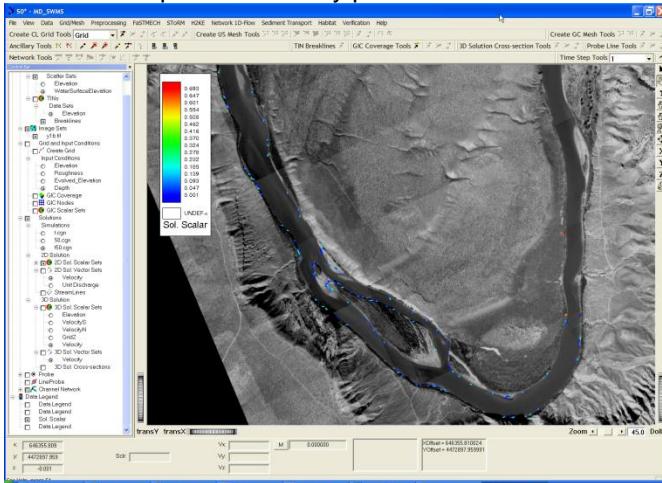


Figure 3.2.7i iRIC showing Habitat Value of the Larval lifestage from the 50 cms flow interval

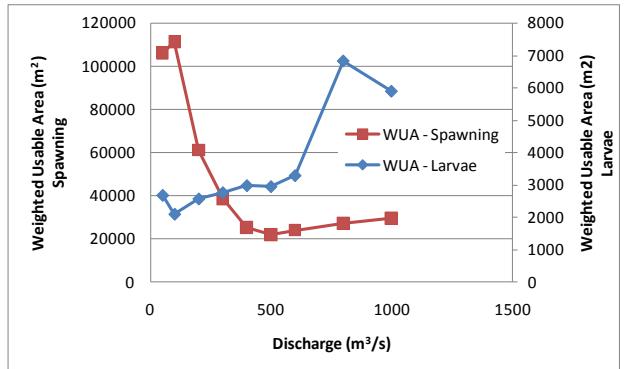


Figure 3.2.7j Plots of WUA for each lifestage over a range of flows from 50 – 1000 cms

3.2.8 FaSTMECH Tutorial 8 – Wilcock and Kenworthy Two Fraction Sediment-Transport Model

This tutorial assumes students have experience with iRIC through at least Tutorial 6. This means you should know how to construct and run models of both flow and sediment-transport, using real data sets or the Channel Builder and have a good grasp of graphics tools in iRIC. This tutorial introduces the Wilcock and Kenworthy two-fraction sediment-transport and mass balance model suitable for situations where prediction of the transport of fine sediment over gravel beds, whose coarse particles are immobile (i.e. sand over immobile gravel, pea gravel over immobile coarse gravel, etc.) is required. This model extends Tutorial 5 and 6 by introducing a new sediment-transport algorithm. It also extends Tutorial 2, providing more experience with creating Input Condition Coverages on the grid. The problem considered here is the transport of a pulse input of sand in a meandering reach under constant discharge over a fine-sediment clean gravel bed. It will identify the location where fine sediment preferentially deposits.

Tutorial 7 steps:

- Import topography and grid.
- Create input conditions for Sand Depth and Sand Fraction and draw polygons or Coverage Regions to specify the values of Sand Depth and Sand Fraction on the Grid.
- Run a time dependent flow and sediment-transport simulation and watch the evolution of a patch of sand over gravel migrates downstream.

Part A - Import Topography and Grid

Import Topography and Ancillary Data

Import the topography from the file BV-meandering.riv from the Tutorial 8 folder in the FaSTMECH – Tutorials directory. The data were collected with the Experimental Advanced Airborne Research Lidar (EAARL), which allows simultaneous surveying in both aquatic and terrestrial domains quickly and remotely. Points were collected approximately every two meters in the longitudinal and transverse directions, allowing a detailed map of the river geometry.

Import Existing Model Grid

From the menu select **File -> Import -> Curvilinear Grid -> XY**. In the File Open dialog select BV- meandering.riv. This will import the node points of the grid but without any of the topography or ancillary data mapped to the grid. Importing the grid in this way can be useful in keeping the model grid consistent from one iRIC project to another. In the Control Bar turn on the **Grid and Input Conditions** and the **Grid and Input Conditions / Create Grid** data objects (fig. 3.2.8a). Note the Centerline for the Curvilinear Grid is defined by points that are approximately one channel width apart and follow as closely as possible to the mean curvature of the channel.

Map Elevations to the Grid

The measured elevations used in this example are generally evenly distributed and provide a nice example of using the Map w/TIN method to map the elevation. From the menu select **Preprocessing -> Set Current Input Condition -> Map w/Tin** (fig. 3.2.8b). When using the Map w/TIN method, all points on the grid that are outside of the TIN boundary are given the maximum elevation in the TIN.

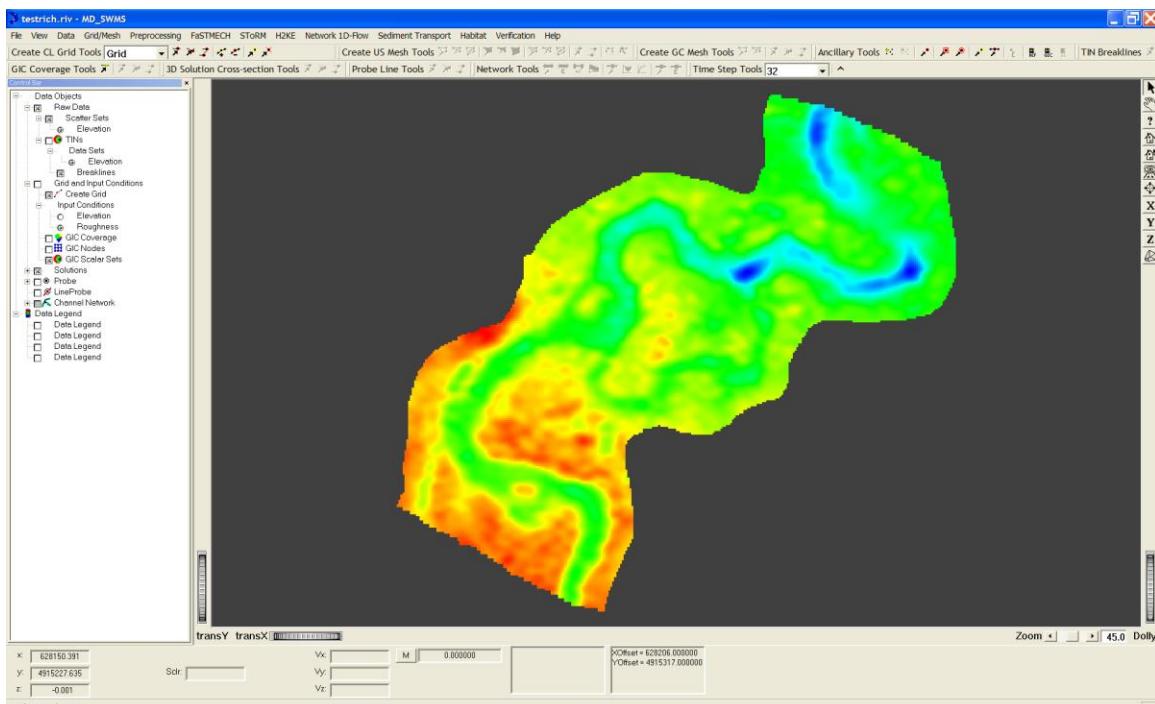


Figure 3.2.8a Imported topography

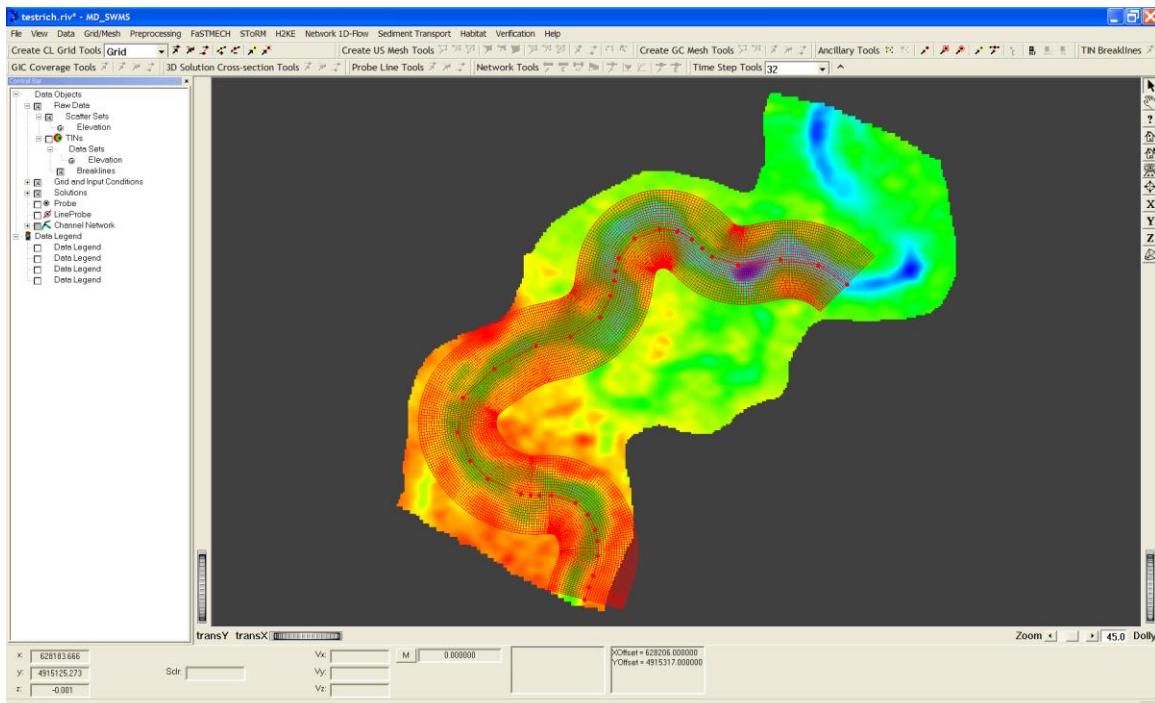


Figure 3.2.8b Imported grid

Part B Create Sand Depth and Sand Fraction Input Conditions

Specify the initial Sand Depth and Sand Fraction by creating new Input Conditions to the grid. Right click on the **Grid and Input Conditions / Input Conditions** branch in the Control Bar, and in the resulting pop-up menu select **Create Input Condition** and select **Sand Depth**. In the Set Sand Depth Default dialog enter the value 0. This will set the default value of Sand Depth to 0 everywhere on the grid. Repeat the previous step and select Sand Fraction and again enter 0 for the default value. Create a small pile of sand near the top of the reach using the GIC Coverage Tool to create three polygons, one atop the other, successively smaller in size, and with successively larger values as illustrated in the following steps:

- First select Sand Depth in the Grid **and Input Conditions / Input Conditions**.
- Turn on the GIC Coverage by selecting the check box. For each Input Condition there is a Coverage that may contain one or more user defined polygons, drawn on the grid and whose property will define the value of each grid node within the polygon. If there is more than one polygon over a grid node, the last polygon drawn will override all underlying polygons.
- To create a new Coverage Region (polygon), select  in the GIC Coverage Tools toolbar. Create a polygon similar to the largest one in Figure 3.2.8xxx
- To define the Sand Depth value of the polygon, right click on the GIC Coverage | Region_0 branch in the Control bar and in the resulting pop-up menu, select Properties. Enter 0.01 in the dialog.
- Repeat the previous step two more times creating successively small polygons and defining the property of each to be 0.02 and 0.04 respectively.

In addition to the steps described above, more information on GIC Coverage Regions can be found in Section 1.4.3 GIC Coverage Toolbar in the MD_SWMS User's Guide.

To edit the location of any Coverage Region, select the region you would like to edit such as **GIC Coverage / Region_1**. You will see that the polygon now is painted red, which indicates that it is selected to be edited. To move a point in the polygon use the  tool in the GIC Coverage Tools toolbar. Select a point of the polygon you would like to move and holding the left mouse button down, drag the point to a new location.

Create a single Coverage Region for the Sand Fraction Input Condition that has the same location as the first polygon for the Sand Depth Input Condition. Set the polygons property to 1.0.

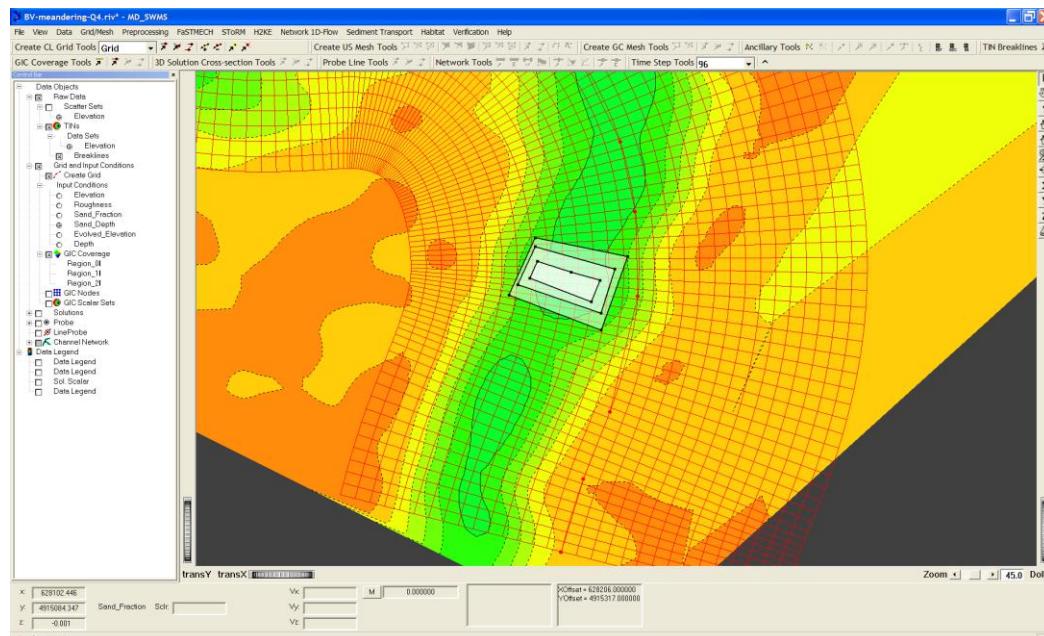


Figure 3.2.8c Location of three polygons used to define the Sand Depth Input Condition.

Part C - Run Simulation

From the menu select **FaSTMECH->New Simulation**. In the Save As Dialog type in a name for the simulation (for example, sim1). From the menu select **FaSTMECH->Edit Input File** and enter in the values shown in figure 3.2.8d and select OK. From the menu select **FaSTMECH->Run**.

A: Discharge

- Boundary Condition: Discharge
- Discharge: Constant (Value: 4)
- Upstream Velocity: Depth Weighting Coefficient (Value: 0.5), Velocity Angle (degrees) (Value: 0)
- Use Specified Velocity Distribution:

B: Stage

- Boundary Condition: Stage
- Downstream Stage: Constant (Elevation: 1953.6)
- Variable Time Series: TSeries_1
- Variable Rating Curve: RCurve_1

C: Roughness

- Boundary Condition: Roughness
- Roughness: Constant (Estimate: 0.006)
- Type: Drag Coefficient (z-naught)
 - Zo: 0.001
 - Cd Min: 0.001
 - Cd Max: 0.1

D: Lateral Eddy Viscosity

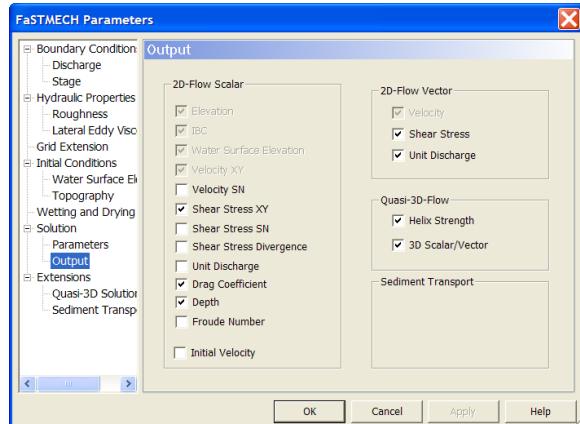
- Boundary Condition: Lateral Eddy Visc.
- Lateral Eddy Viscosity: Constant (Coefficient: 0.07)
- Start LEV: 0, End LEV: 0
- Beg Iter: 0, End Iter: 0
- Change Iter: 0

E: Water Surface Elevation

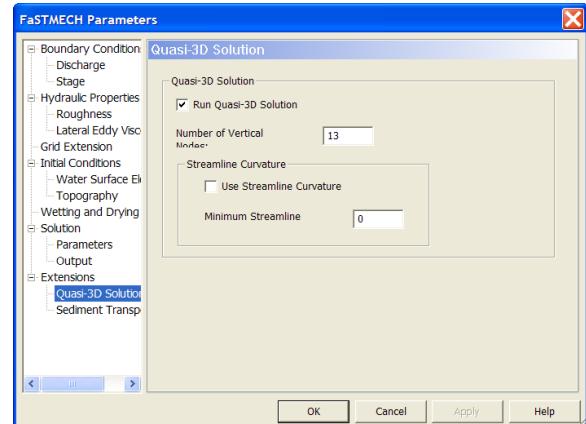
- Boundary Condition: Water Surface E.
- Water Surface Initial Guess: Create 1-D Solution (Upstream Water Surface Elevation: 0)
- Boundary Conditions for 1-D Solution: Discharge (Value: 4), Stage (Value: 1953.64), Drag Coefficient (Value: 0.006)
- Existing Solution: Q4.cgn
- Use Previous Solution: Not Available
- Solution Number: 1

F: Parameters

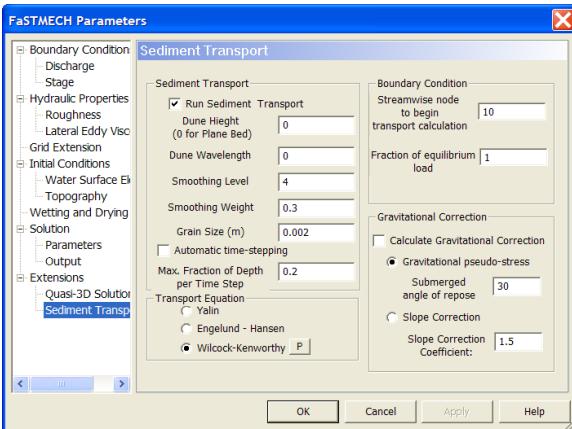
- Solution Parameters: Constant Discharge (Number of Iterations: 1000)
- Variable Discharge: Start Time (seconds): 1, End Time (seconds): 345600, Increment (seconds): 1200, Inter Time Step Number of Iterations: 100, Max. Number of Iterations: 2, Multiplier: 1, Plot Increment: 3
- Relaxation Parameters: ERelax: 0.25, URelax: 0.3, ARelax: 0.3
- Debug Stop: Use alternate stop criteria, Time: 0, Iteration number: 0



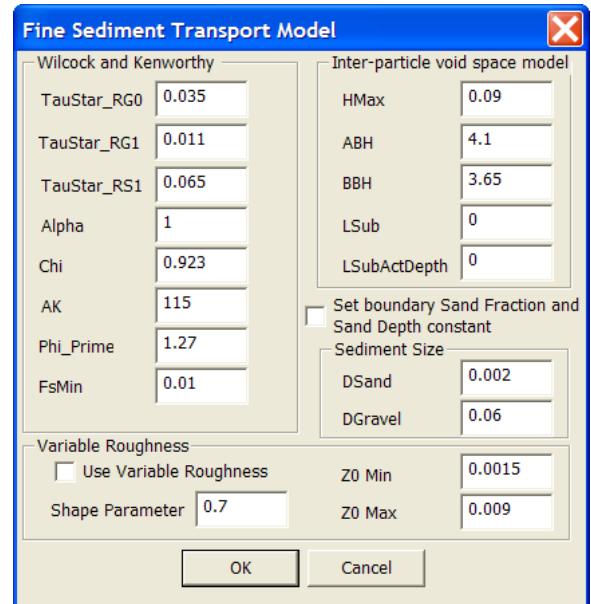
G



H



I



J

Figure 3.2.2d: Enter the following parameters into the FaSTMECH Parameters dialog. (A) Discharge, (B) Stage, (C) Roughness, (D) Lateral Eddy Viscosity, (E) Water Surface Elevation, (F) Solution Properties, (G) Solution Output, (H) Quasi-3D Solution, (I) Sediment Transport, (J) Fine Sediment Transport – Selected by pushing the P-Button in the Sediment Transport Parameters.

Part C – Visualize Results

The resulting Sand Depth and Sand Fraction are shown at 0, 1, 2 and 3 days of simulation in Figure 3.2.8e. At the given discharge, the sand input rapidly moves downstream spreading over the first meandering section and forming a deep deposit in the inner side of the bend developing a sand bar. Successively, large sand depositions occur in the inner part of bends where they tend to form sand bars. Simulations show transport in sand waves as observed in laboratory experiments. This effect is due to shear stress and mesh cell heterogeneity in the numerical model.

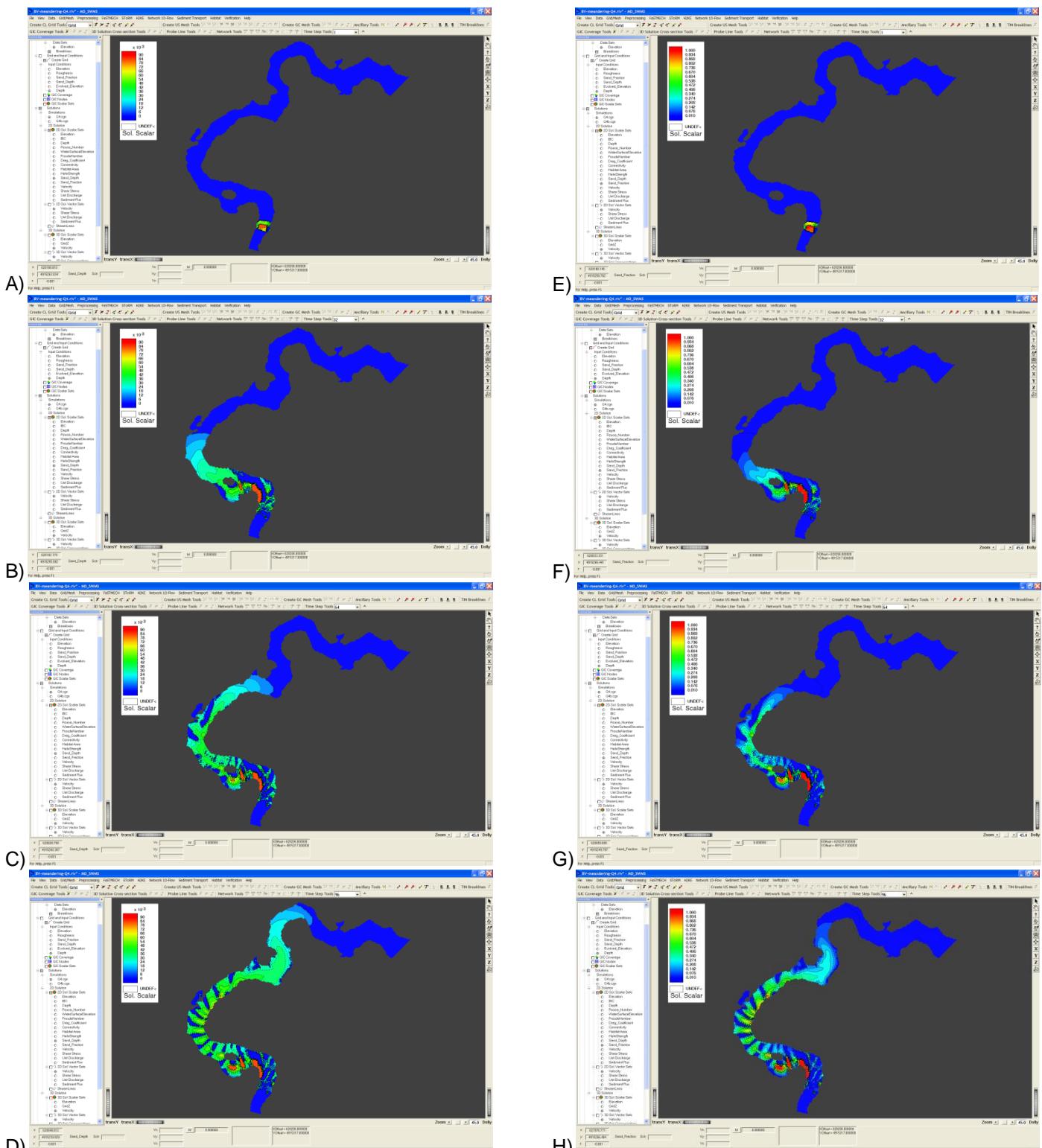


Figure 3.2.8: A-D) Sand Depth at 0,1,2, and 3 days of simulation. E-H) Sand Fraction at 0,1,2, and 3 days of simulation

3.4 Morpho2D Tutorials

3.4.1 Morpho2D Tutorial 1 – An introduction to iRIC: flow modeling using Morpho2D solver; flow, bed evolution and vegetation.

In this tutorial, the iRIC v.1.0b modeling system is used to model the flow through the reach of interest. The problem is approached in two ways. The first imports a grid and accompanying topography from text files created in software outside of iRIC. The second imports topography and then builds a grid using the general coordinate grid generating tools provided in iRIC. In each case the solver is run for a given set of parameters and boundary conditions and the results are visualized. Finally, the model will be extended to include variable grainsize fractions in the surface and subsurface layers, vegetation and specification of a fixed bed elevation.

The following conventions are used in this tutorial to identify Control Bar items or branches, and Menu Items. Menu items are in bold and the path is defined by arrows. Control Bar branches are in italicized bold and the path of the branch is defined by vertical lines. For example:

Menu: **File->Import->Topography**

Control Bar: ***Grid and Input Conditions / Create Gen. Coord. Grid***

Tutorial 1 steps:

First Approach – Import General Coordinate Grid

1. Initialize the Morpho2D solver by loading the Morpho2D.xml file.
2. Import grid and associated topography.
3. Define the magnitude of Roughness associated with the input condition
4. Create a simulation and edit the calculation conditions
5. Run the Morpho2D solver and visualize the results.

Second Approach – Create General Coordinate Grid in the iRIC modeling interface

1. Create a new project and import the topography
2. Create a grid using iRIC's general coordinate grid tools
3. Map topography to the grid
4. Repeat steps 3-5 in the first approach

Extend the model – Create General Coordinate Grid in the iRIC modeling interface

- Import ancillary data sets including: vegetation density, vegetation height, and fixed bed elevation
- Map new ancillary data sets the grid input condition using the TIN method
- Create new simulation, run the solver and visualize the results.

First Approach

1. Initialize iRIC to use the Morpho2D solver

Start the application by double clicking on the iRIC shortcut from your computers desktop. From the File Menu select **File->Initialize**. In the Open XML Definition File Dialog browse to the iRIC Tutorials\morpho2D\Tutorial 1 folder and select the Morpho2D.xml file. This file initializes the iRIC interface with the calculation grid and grid conditions required for the Morpho2D solver. As the tutorial progresses we will highlight how the Morpho2D.xml file interacts with and initializes the iRIC modeling interface.

2. Import Existing Model Grid

From the menu select **File->Import->Gen. Coord. Grid->.ini Format**. In the Locate and Select Takebayashi Grid File dialog select one of the Xini, Yini or Zbini.dat files. This will import the elevation associated with the grid, create a TIN of the elevation, import the grid geometry, and map the elevation to the grid. A series of dialogs are presented as shown in Figures 3.4.1.1 A&B. In each of these dialogs answer in the affirmative. The result of the import operation can be seen in the Control Bar (Figure 3.4.1.1 C). Notice that **Ancillary Data** branch of the Control Bar contains a **Scatter Set** of Elevation, and a **TIN** of Elevation. In addition, expanding the **Grid and Input Conditions** branch of the Control Bar reveals the branches for the **General Coord. Leftedge**, **General Coord. Leftedge**, and **General Coord. Centerline**. For now, their presence is noted; in the second approach documented later in this tutorial their function will be documented. Also notice that the Input Conditions are loaded with several values (Elevation, FixedBedElevation, Roughness, etc.) as defined in the Morpho2D.xml file (Figure 3.4.1.2).

The Control Bar is used to turn Data Objects, such as **Ancillary Data**, **TINs**, and **Grid and Input Conditions**, etc. On and Off by selecting or deselecting the adjacent checkbox and, where there is an option, to select which value is viewed. For example, selecting the box next to **Ancillary Data / Scatter Sets** will turn the scatter set of Elevation on, to view the entire data set, select the refresh button(on the right side of the iRIC interface which will center the Elevation Scatter Set as shown in Figure 3.4.1.3. The **TIN**, and **Grid and Input Conditions** can also be turned On and Off by selecting the select box next to each respectively.

3. Define the magnitude of Roughness associated with the Input Condition

Save the project first by selecting **File->Save** from the menu and naming the project Tut1.riv. For each **Input Condition** under the **Grid and Input Conditions** branch in the Control Bar there are three data types, **GIC Coverage**, **GIC Nodes**, and **GIC Scalar Sets**. The last two are for viewing the value of the **Input Condition** and the first, **GIC Coverage**, is for interactively defining the value of the **Input Condition** as described below. In this tutorial only the Elevation and Roughness **Input Condition** are used. The effects of Vegetation, a Fixed Bed Elevation or Obstacles to the flow will not be modeled and the default values for these Input Conditions are acceptable.

For each **Input Condition** there is a **GIC Coverage** and for each **GIC Coverage** one or more **Regions** can be defined by a user drawn polygon with an associated property. Create a polygon around the entire grid to define the value of roughness for all nodes of the grid using the GIC Coverage Tool as illustrated in the following steps:

- Turn on **Grid and Input Conditions** and **Create Gen. Coord Grid**.
- Select Roughness in the **Grid and Input Conditions / Input Conditions**.
- Turn on the **GIC Coverage** by selecting the check box. For each **Input Condition** there is a **GIC Coverage** that may contain one or more **Regions** that are user defined polygons, drawn on the grid, and whose property will define the value of each grid node that is contained within the polygon. If there is more than one polygon over a grid node, the last polygon drawn will override all underlying polygons.
- To create a new **GIC Coverage / Region** (polygon), select in the GIC Coverage Tools toolbar. Create a polygon similar to the one in Figure 3.4.1.4. Using the left mouse button click to define the location of polygon nodes and press Enter to finish.

- To define the Roughness value of the polygon, if necessary first expand the **GIC Coverage** branch, right click on the **GIC Coverage / Region_0** branch in the Control bar, and in the resulting pop-up menu, select **Properties**. Enter 0.035, the Manning's n value, in the dialog.

To edit the location of any point in a Coverage Region polygon, use the left mouse button to select the region you would like to edit, such as **GIC Coverage / Region_0**. This will paint the polygon red which indicates that it is selected to be edited. To move a polygon point use the  tool in the GIC Coverage Tools toolbar. Be aware that there are similar looking buttons in other toolbars, so make sure you are selecting from the GIC Coverage Tools toolbar. Select a point of the polygon you would like to move and holding the left mouse button down, drag the point to a new location. Select  every time that a new point needs to be moved.

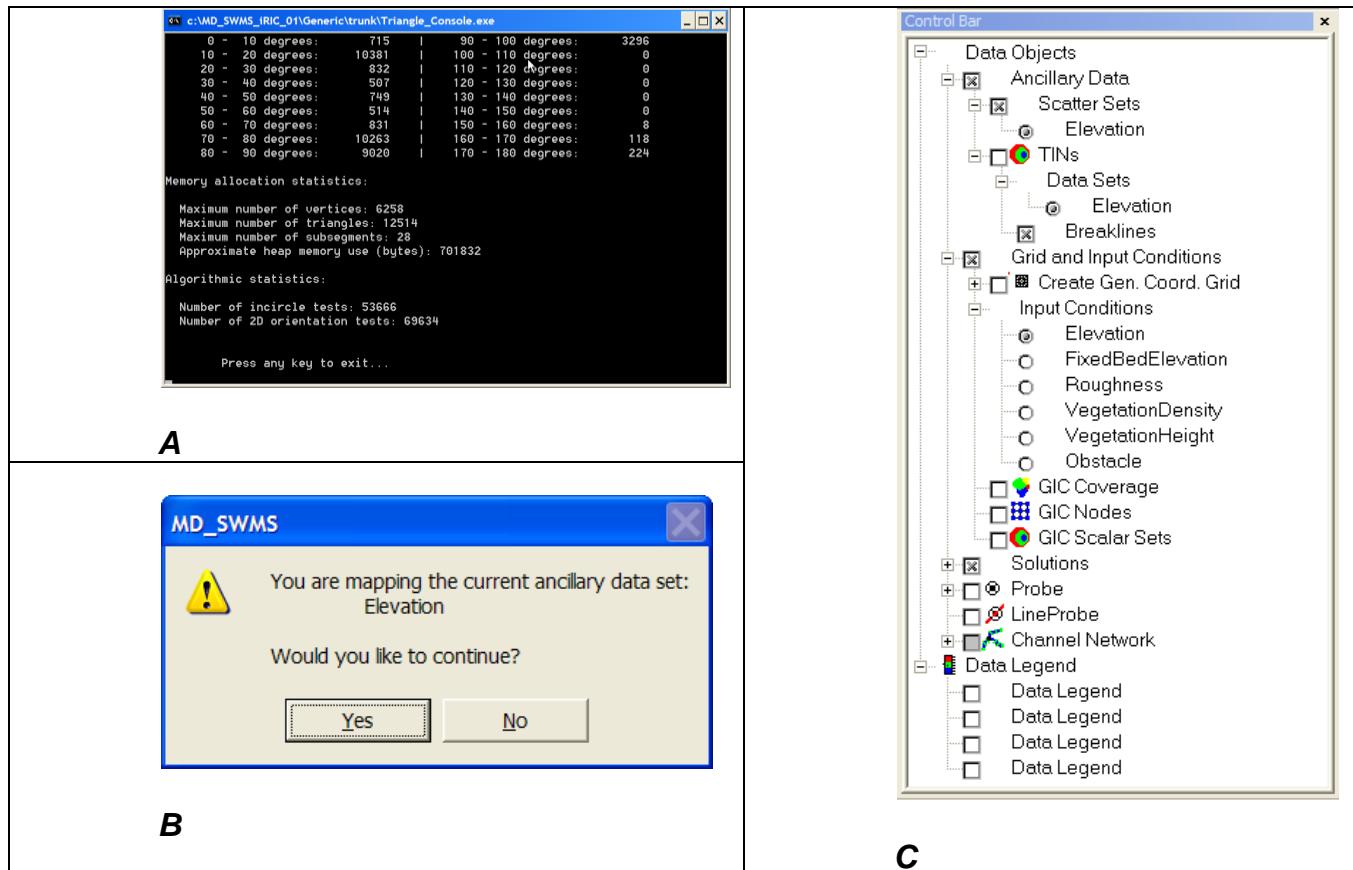


Figure 3.4.1.1 A) By default a TIN of the elevation is created. The console window shows the results of that TIN; pressing any key will close the window. B) By default the elevation will be mapped to the imported grid using the TIN of the elevation data. C) The Control Bar indicates the **Data Objects** created on the import of the General Coordinate System Grid.

```

<GridRelatedCondition>
    <Item name="elevation" caption="Elevation">
        <Definition position="node" valueType="real" default="0" />
    </Item>
    <Item name="roughness" caption="Roughness">
        <Definition position="node" valueType="real" default="0" />
    </Item>
    <Item name="fixed_bed_elevation" caption="FixedBedElevation">
        <Definition position="node" valueType="real" default="0" />
    </Item>
    <Item name="vegetation_density" caption="VegetationDensity">
        <Definition position="node" valueType="real" default="0" />
    </Item>
    <Item name="vegetation_height" caption="VegetationHeight">
        <Definition position="node" valueType="real" default="0" />
    </Item>
    <Item name="obstacle" caption="Obstacle">
        <Definition position="node" valueType="integer" default="1" option="true">
            <Enumerations>
                <Enumeration value="1" caption="Normal" />
                <Enumeration value="2" caption="Obstacle" />
            </Enumerations>
        </Definition>
    </Item>

```

Figure 3.4.1.2 A piece of the Morpho2D.xml file showing the GridRelatedCondition node that defines the input conditions in the **Grid and Input Conditions** branch of the Control Bar. Note the items are Elevation, Roughness, FixedBedElevation, VegetationDensity ,VegetationHeight and Obstacle all of which have been dynamically loaded into the iRIC modeling interface.

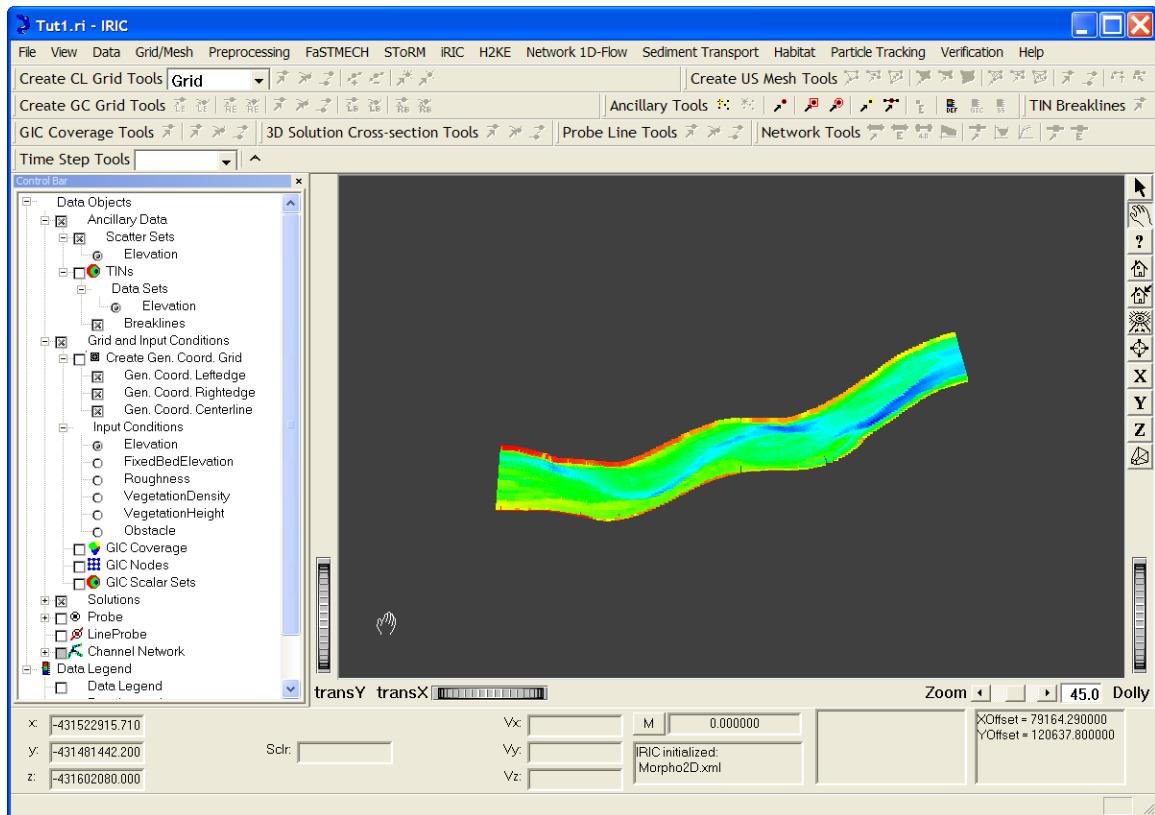


Figure 3.4.1.3 The Ancillary Scatter Set of Elevation that has been centered by selecting the refresh button . Flow is from left to right.

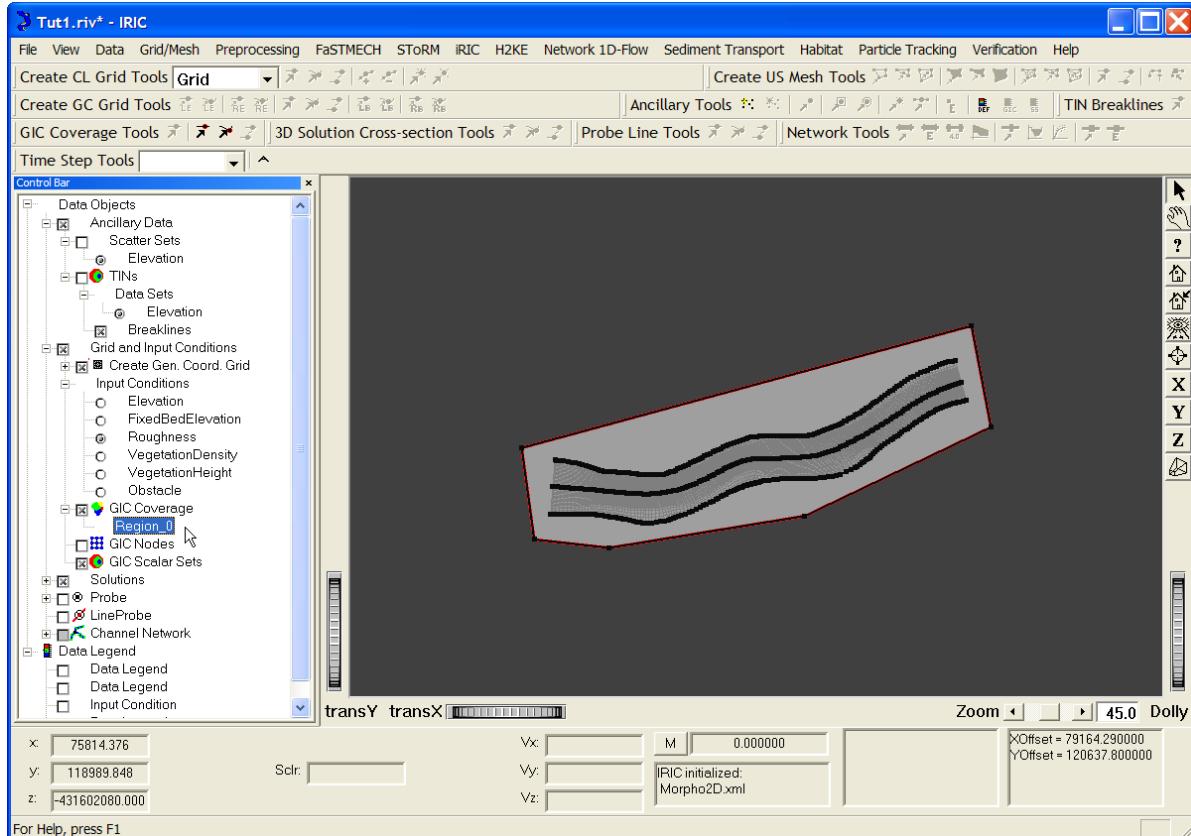
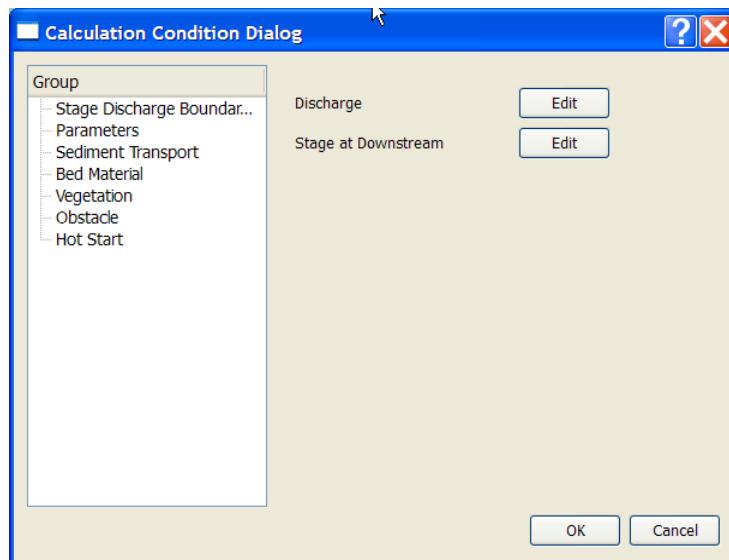


Figure 3.4.1.4 Coverage Region polygon for Roughness. Note that Roughness is the currently selected input. The **Grid and Input Condition** and **GIC Coverage** branches of the Control Bar are both turned on, and under the **GIC Coverage** branch is **Region_0** which represents the user-defined polygon surrounding the grid.

4. Create a simulation and edit the calculation conditions

From the menu select **iRIC->Create New Simulation**. In the Save As Dialog type in a name for the simulation (for example, Tut1a). From the menu select **iRIC->Edit Calculation Conditions**. The top level group is the Stage-Discharge Boundary Condition. For both Discharge and Stage we will select the Edit button in the dialog and in the resulting dialog select the Import button. For Discharge select the Discharge1.txt file to import and for the Stage at Downstream select the Stage1.txt file to import. The table in each case will be filled with the values as shown in Figure 3.4.1.5 (B and C). For the remaining Groups enter the values as shown in Figure 3.4.1.5 (D-H).



A

	Time	Discharge
1	0	3556.5
2	100	3556.5
3	200	3556.5
4	300	3556.5
5	400	3556.5
6	500	3556.5
7	600	3556.5
8	600	3556.5
9	800	3556.5
10	900	3556.5
11	1000	3556.5
12	1100	3556.5
13	1200	3556.5
14	1300	3556.5
15	1400	3556.5
16	1500	3556.5
17	1600	3556.5
18	1700	3556.5

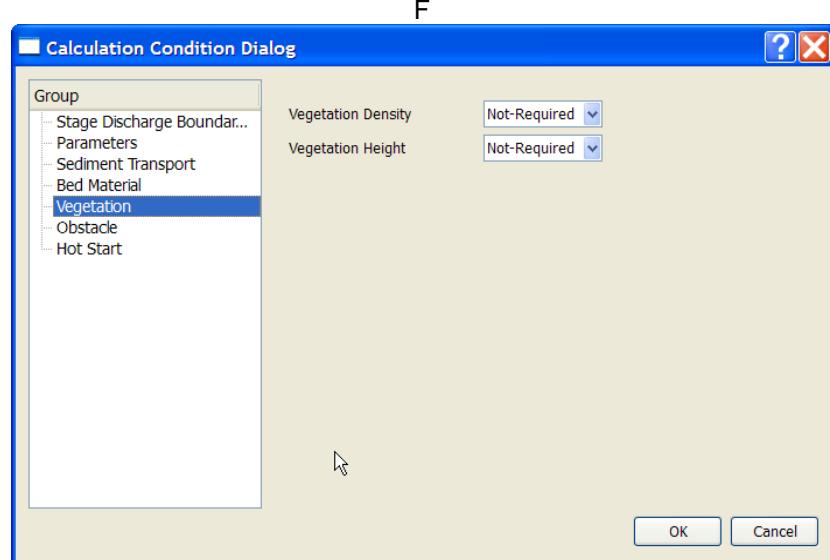
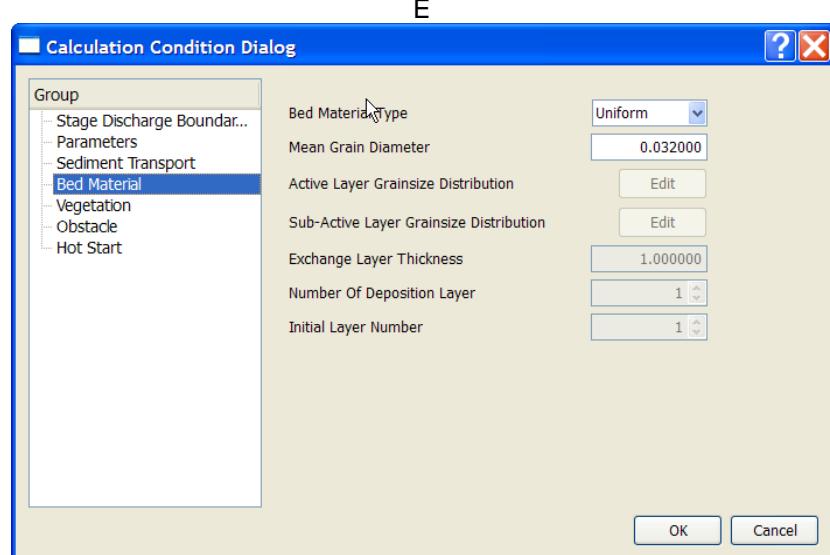
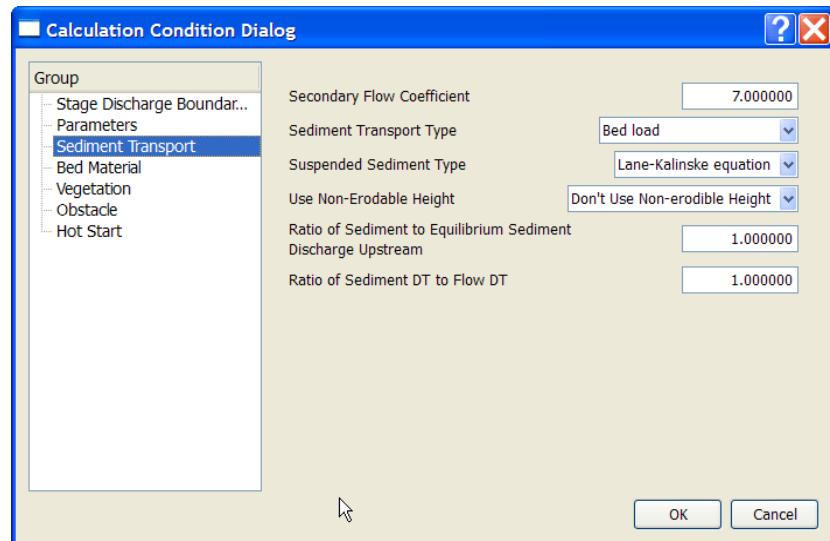
B) Select Import and open the Discharge1.txt file

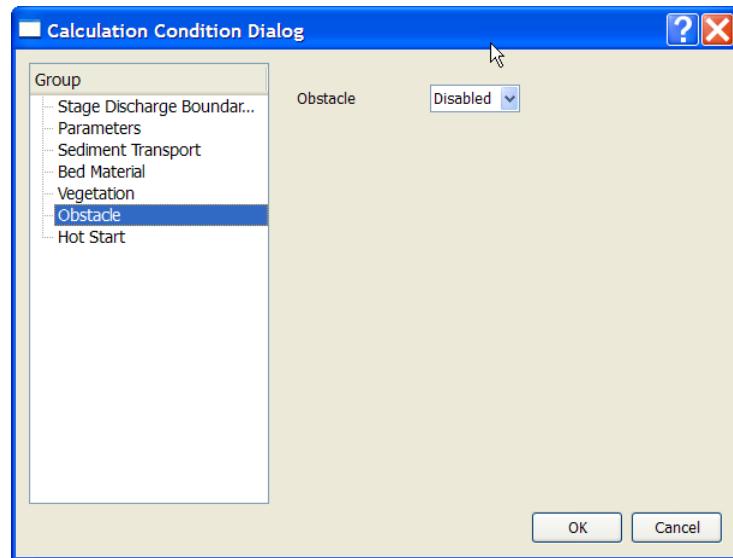
	Time	Stage
1	0	7.440999985
2	100	7.440999985
3	200	7.440999985
4	300	7.440999985
5	400	7.440999985
6	500	7.440999985
7	600	7.440999985
8	600	7.440999985
9	800	7.440999985
10	900	7.440999985
11	1000	7.440999985
12	1100	7.440999985
13	1200	7.440999985
14	1300	7.440999985
15	1400	7.440999985
16	1500	7.440999985
17	1600	7.440999985
18	1700	7.440999985

C) Select Import and open the Stage1.txt file

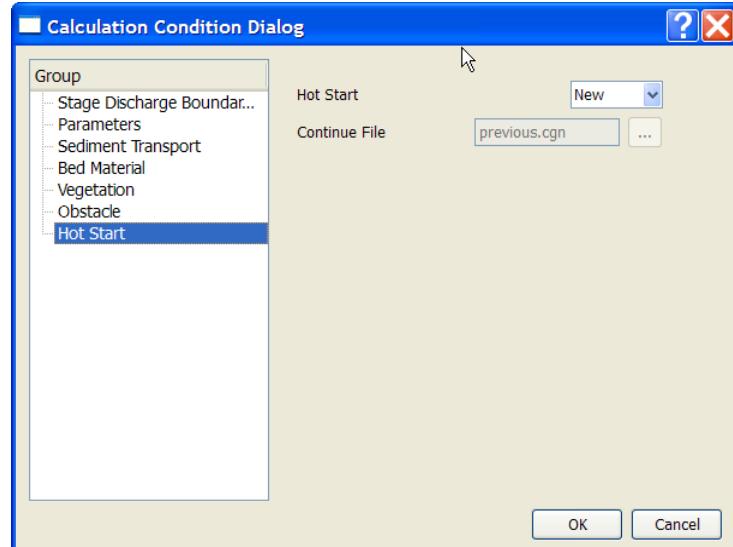
The screenshot shows the 'Calculation Condition Dialog' window. On the left, 'Parameters' is selected in the 'Group' list. On the right, there are several input fields: 'Calculation Type' (Flow Only), 'Start Time' (0.000000e+00), 'End Time' (2000.000000), 'Computational Timestep' (0.500000), 'Output Timestep for File' (100.000000), 'Output Timestep for Screen' (10.000000), and 'Bed Evolution Time' (1000.000000). At the bottom are 'OK' and 'Cancel' buttons.

D





H



I

Figure 3.4.1.5: (A) Edit the Discharge and Stage at Downstream by selecting each Enter button respectively. In each of the resulting dialogs select the Import button and select the Discharge1.txt and Stage1.txt respectively. The resulting dialogs should look like (B) and (C) respectively. For each of the following groups enter the parameters as shown in (D-I)

The Calculation Condition Dialog is dynamically built from the Morpho2D.xml file. A piece of the Morpho2D.xml file is shown in Figure 3.4.1.6 for the Parameters group. It is evident from an examination of the Dialog and the xml file how the Parameters Group is built.

```

<Tab name="parameters" caption="Parameters">
    <Content>
        <Items>
            <Item name="calculation_type" caption="Calculation Type">
                <Definition conditionType="constant" valueType="integer" option="true" default="0">
                    <Enumerations>
                        <Enumeration value="0" caption="Flow Only" />
                        <Enumeration value="1" caption="Bed Variation" />
                    </Enumerations>
                </Definition>
            </Item>
            <Item name="start_time" caption="Start Time">
                <Definition conditionType="constant" valueType="real" option="false" default="0" min="0" max="1000000000000000000.0">
                <Dependency>
                    <Condition type="always" target="start_time" />
                    <Action type="setMinimum" target="start_time" />
                </Dependency>
            </Item>
            <Item name="end_time" caption="End Time">
                <Definition conditionType="constant" valueType="real" option="false" default="0" max="1000000000000000000.0">
                <Dependency>
                    <Condition type="always" target="end_time" />
                    <Action type="setMaximum" target="end_time" />
                </Dependency>
            </Item>
            <Item name="computational_timestep" caption="Computational Timestep">
                <Definition conditionType="constant" valueType="real" option="false" default="0" min="0" max="1000000000000000000.0">
            </Item>
            <Item name="output_timestep_for_file" caption="Output Timestep for File">
                <Definition conditionType="constant" valueType="real" option="false" default="0" min="0" max="1000000000000000000.0">
            </Item>
            <Item name="output_timestep_for_screen" caption="Output Timestep for Screen">
                <Definition conditionType="constant" valueType="real" option="false" default="0" min="0" max="1000000000000000000.0">
            </Item>
            <Item name="bed_evolution_time" caption="Bed Evolution Time">
                <Definition conditionType="constant" valueType="real" option="false" default="0" min="0" max="1000000000000000000.0">
            </Item>
        </Items>
    </Content>
</Tab>

```

Figure 3.4.1.6 A piece from the Morpho2D.xml file showing the Parameters Tab (Figure 3.4.1.5D) and its contents.

5. Run the Morpho2D solver and visualize the results.

From the menu select **iRIC->Run**. When the solver executes, a new Console window will appear- a snapshot is shown in Figure 3.4.1.7. As defined in the Parameters Group in the Edit Calculation Conditions dialog the Console window will refresh every 10 seconds of computation time so that the progress can be viewed.

```

Non-uniform sediment model Ver.1

Present calculation time      = 400.0000000 s
Start time of bed deformation = 100.0000000 s
End time                      = 1000.0000000 s
Water discharge at upstream end = 3556.5000000 m3/s
Time step of flow              = 1.0000000 s
Water level at downstream end = 7.4159699 m
Initial bed slope              = 0.0008829
Coefficient for secondary flow = 7.0000000
Thickness of exchange layer   = 0.3000000 m
(Supplied)/(Equilibrium sediment discharge)= 1.0000000
(dt of bed deformation)/(dt of flow) = 1.0000000

Type of calculation: Bed deformation
Type of sediment transport: Bed load only
Bed material: Uniform
Vegetation density data: Not in use
Vegetation height data: Not in use
Rigid bed height data: Not in use

```

Figure 3.4.1.7 An snapshot of the Takabayashi Console window during the execution of the solver.

When the computation is completed, the Console window will close and the Solutions branch in the Control Bar will display the available Scalar and Vector values available for viewing. The **2D Solutions | 2D Sol. Scalar Sets** is set to Velocity and the **2D Solutions | 2D Sol. Vector Sets** is set to Velocity. To scroll through the saved solutions use the Time Step Tools toolbar and using the drop-down list select the solution to view, currently the time step is set to 101, the last time step of the simulation. Save the project using **File->Save**, and give the project a name such as Tut1.

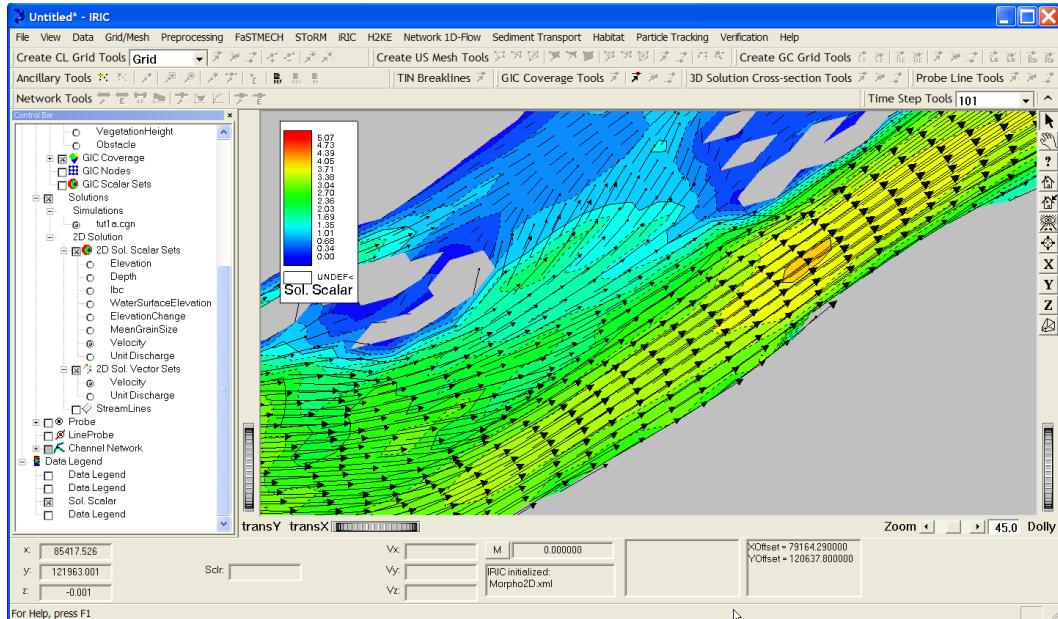


Figure 3.4.1.8 The solution at time step 101. Note that Velocity is selected in both the scalar and vector sets. Using the mouse double left click **Solutions / 2D Sol Vector Sets** in the Control bar to set the vector attributes. The General attributes have been set to use a constant color (black) for both the body and tail, and the Arrow End attributes have been set to use a constant color and triangle shape

Second Approach

1. Create a new Project and import the topography

Begin by selecting **File->New** from the Menu. This will reinitialize the interface to begin a new project. If the iRIC application has been closed and re-started, then you will need to re-initialize the solver definition.xml file, by selecting **File->Initialize** from the menu and the selecting the Morpho2D.xml file in the iRIC Tutorials\morpho2D\Tutorial 1 folder. From the Menu select **File->Import->Topography**. In the resulting Select Topography File to Import dialog, select the previously saved .riv file (for example, Tut1.riv). This will load the saved topography from our previous project. As in the first approach, the console window for the TIN application will appear, selecting any key as noted in the console window will close it. Another dialog, the Project File Folder will appear with the path to the Tut1.riv file. This indicates that all the data required for the project can be found in this folder, and all data files created by the iRIC application will be saved to this directory, select OK to use the default directory. Note that in the Control Bar, only the Ancillary branch has data associated with it, in this case a Scatter Set of Elevation and a TIN of Elevation.

2. Create a grid using iRIC's General Coordinate Grid tools

This version of iRIC has tools for generating three different types of grids, two of which are structured, and one of which is unstructured. Morpho2D can be used with a structured but general coordinate system. Select **Grid/Mesh->2D General Coordinate Grid** from the menu. This will activate the general coordinate grid generating tools located in the Create GC Grid Tool toolbar (Figure 3.4.1.9).

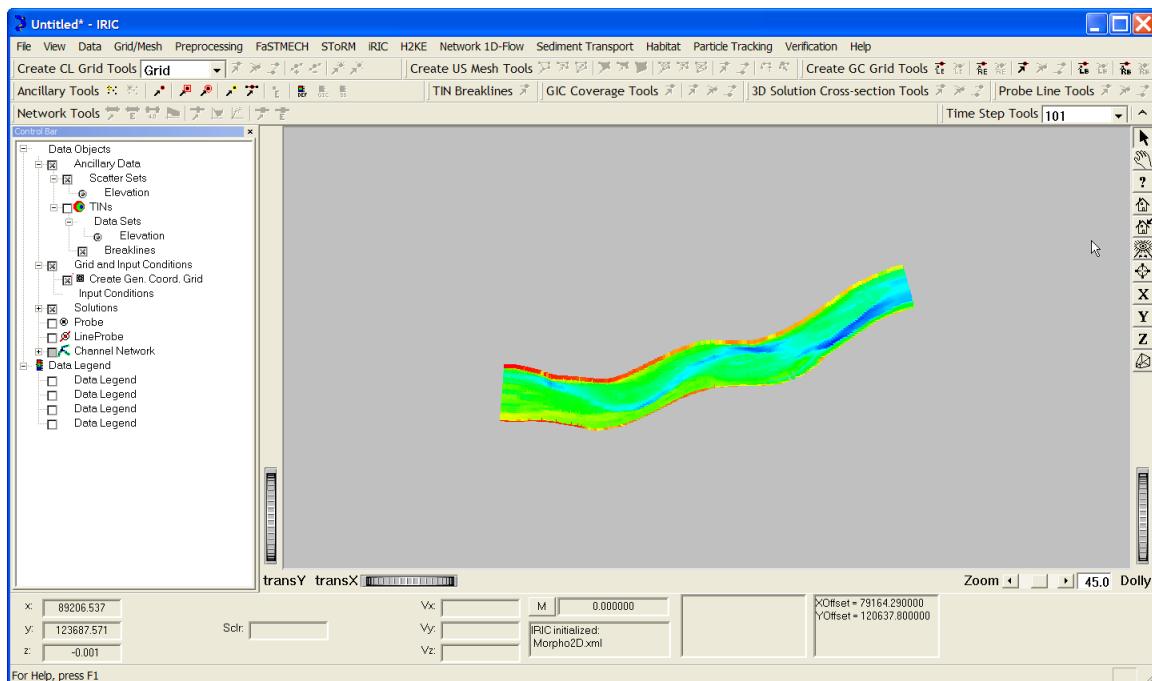


Figure 3.4.1.9 Notice that the **Grid and Input Condition** and the **Create Gen. Coord Grid** branches in the Control bar are turned on and in the Create GC Grid Tools two buttons have been activated. The first is used to create a polygon to define the grid boundary, and the second used to create a center line.

3. Create a grid using iRIC's General Coordinate Grid tools

The following steps will illustrate how to create a general coordinate grid using the tools provided in iRIC. All the tools used will be from the Create GC Grid Tools toolbar (Figure 3.4.1.10). When the mouse is placed over any toolbar button, the status bar at the bottom of the iRIC interface contains information about that button. To access the Create GC Grid Tools make sure that the **Grid and Input Condition** and the **Create Gen. Coord Grid** branches in the Control bar are turned on.

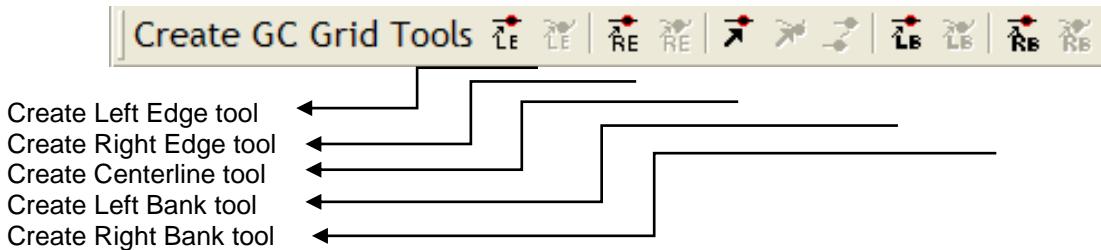


Figure 3.4.1.10 The Create GC Grid Tools. These tools are used to define the shape and extent of the general coordinate grid.

- A. **Create Left Edge Line:** Create a left edge line from upstream to downstream, in this case from left to right. Left and right are defined relative to looking downstream. Select the Create Left Edge tool and use the left mouse button to define individual points creating a line that defines the left edge of the model domain, finishing the left edge line by selecting Enter. The resulting left edge line should look like that in Figure 3.4.1.11, beginning well upstream and ending well downstream of the measured topography. To zoom or pan the topography data during the polygon creation simply use the Dolly or TransX or TransY wheels at any time. Alternatively, using the Alt key with the left mouse button will zoom and Alt + Ctrl with the mouse button will pan.
- B. **Create Right Edge, Left Bank and Right Bank lines:** Repeat the process for creating the left edge line as above for the Right Edge, Left Bank, and Right Bank lines. Use Figure 3.4.1.11 to guide the placement of individual lines, or use the topography as a guide.
- C. **Create Centerline:** Create a centerline from upstream to downstream, in this case from left to right. Select the Create Centerline tool and using the left mouse button select centerline points and finish the centerline by selecting Enter. Your centerline should look like that in Figure 3.5.1.11 and 3.5.1.12. The centerline points must be within the Left Bank and Right Bank lines. From the centerline, cross-sections are drawn perpendicular to the centerline out to the edge of the left and right bank lines.
- D. **Define Grid Parameters:** Define the number of nodes in the streamwise and cross-stream directions by selecting **PreProcessing->Set 2D General Coord. Grid Parameters** from the menu. Set the Num. Streamwise Nodes to 149. Leave the Set with Boundary Fitted Coordinates box unchecked at this time. The result shows the intersection of each streamwise node with the bank and edge lines, and can be used as a guide to when editing the grid so the boundaries are within the topographic data and the grid boundaries are perpendicular to the incoming and outgoing flow direction.
- E. **Edit Centerline Points:** Step D must be completed before you can edit any of the lines you created. You may notice that at either the upstream or downstream Boundary there are problems with the grid extending out of the bounds of the measured topography, as in Figure 3.4.1.12. Edit the centerline by first selecting with the mouse the **Gen. Coord. Centerline** branch in the Control Bar. This will paint the centerline red which indicates that it is in edit mode, and the Edit Centerline tool will be enabled. Using the Edit Centerline tool, select the centerline point with the left mouse button and holding the button down drag the point to a new location. Note that the refresh rate of the graphics may be slow when you drag the point to a new location. Keep holding the mouse button down and drag slowly to the new location. You will have to reselect the tool

each time to use it. The geometry of the grid should have rows that are parallel to the upstream and downstream boundaries as in Figure 3.4.1.14

- F. **Create Boundary fitted Grid:** Select **PreProcessing->Set 2D General Coord. Grid Parameters** from the menu. Enter the parameters as in Figure 3.4.1.15A and select OK. It may take 5 – 10 minutes for the grid to generate. The resulting grid is show in Figure 3.4.1.15B.

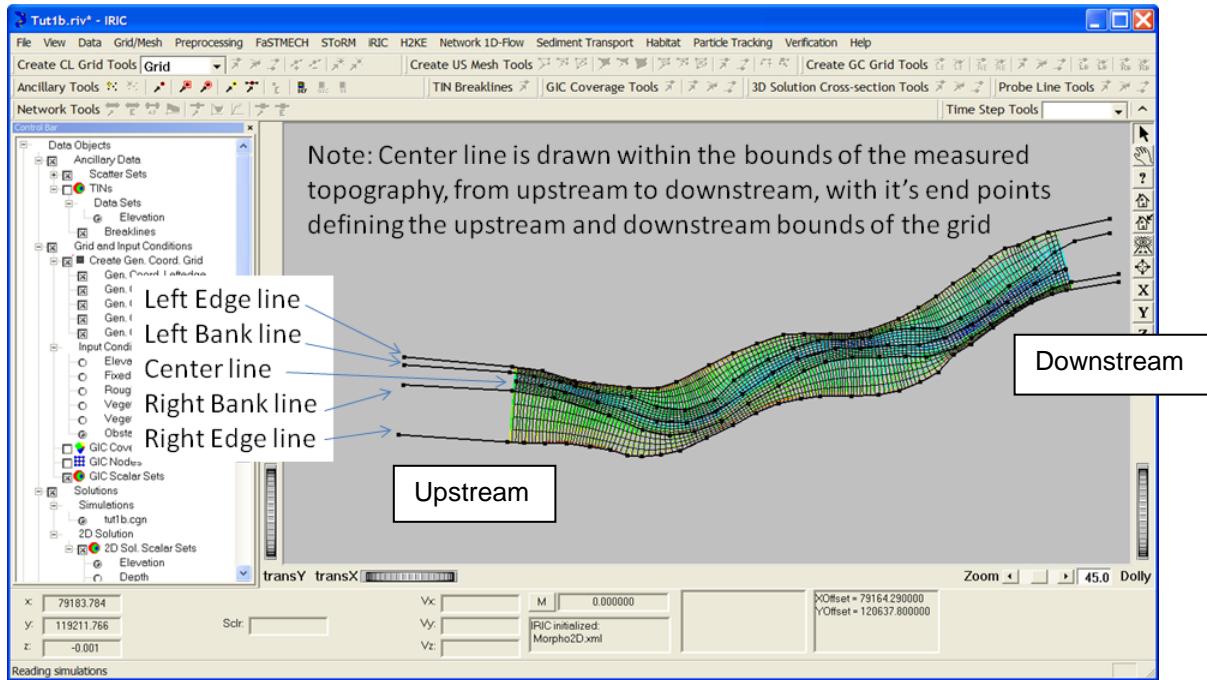


Figure 3.4.1.11 The **Gen Coord. Left Edge**, **Gen Coord. Right Edge**, **Gen Coord Left Bank**, **Gen Coord. Right Bank** and **Gen Coord. Centerline** are shown. The Edge and Bank lines are extended from the top and bottom boundary of the grid and the Centerline is drawn within bounds of the topography.

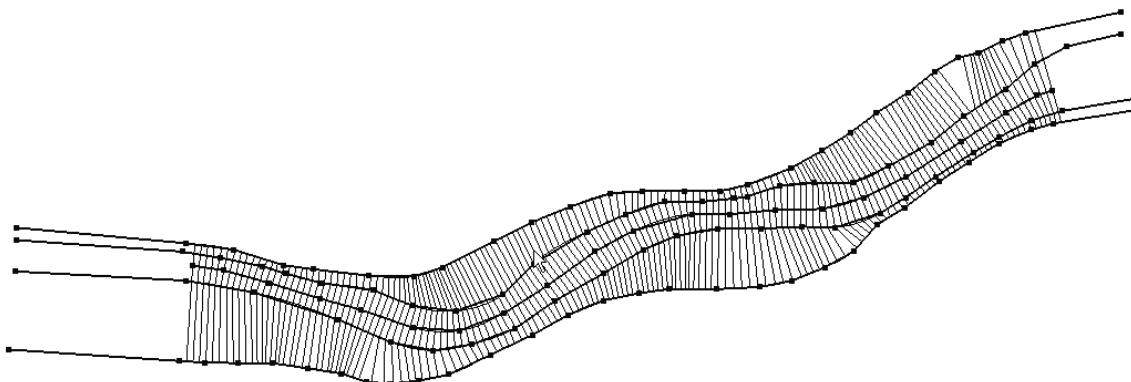


Figure 3.4.1.12 The left edge, left bank, center, right bank and right edge lines are drawn from upstream to downstream, in this case, from left to right. The left and right edge lines define the edge of the flood plain. The left and right bank lines define a meandering channel within the floodplain. The Center line defines the upstream and downstream extent of the grid. Here the grid is shown without the boundary fitting option.

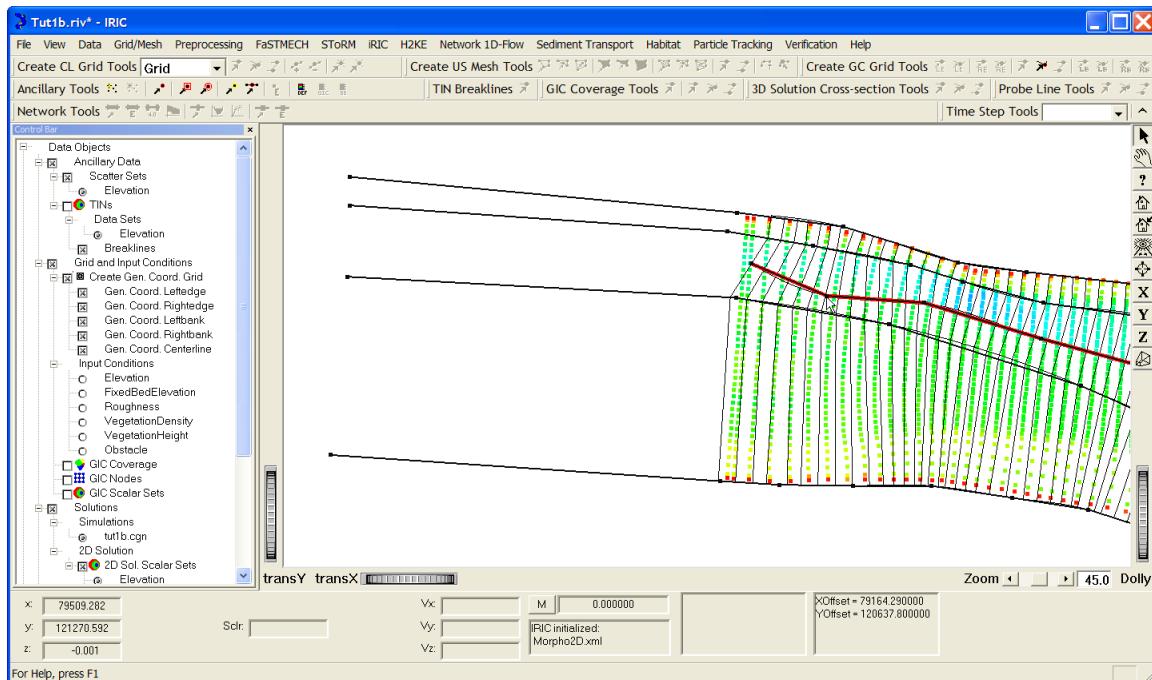


Figure 3.4.1.13 Often the **Gen Coord. Centerline** needs to be edited at the upstream or downstream boundary so that the rows of the grid are perpendicular to the flow direction and approximately parallel to the boundary. Follow the directions in Step 4 above to edit the grid so that the result looks like that in Figure 3.4.1.14.

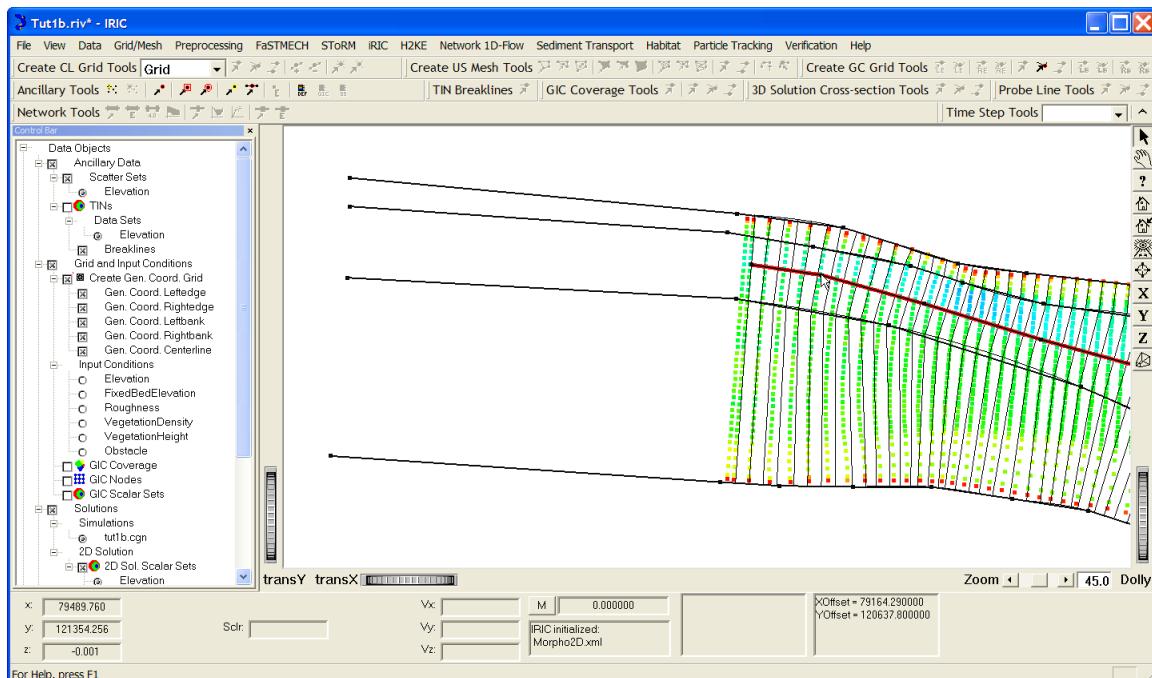
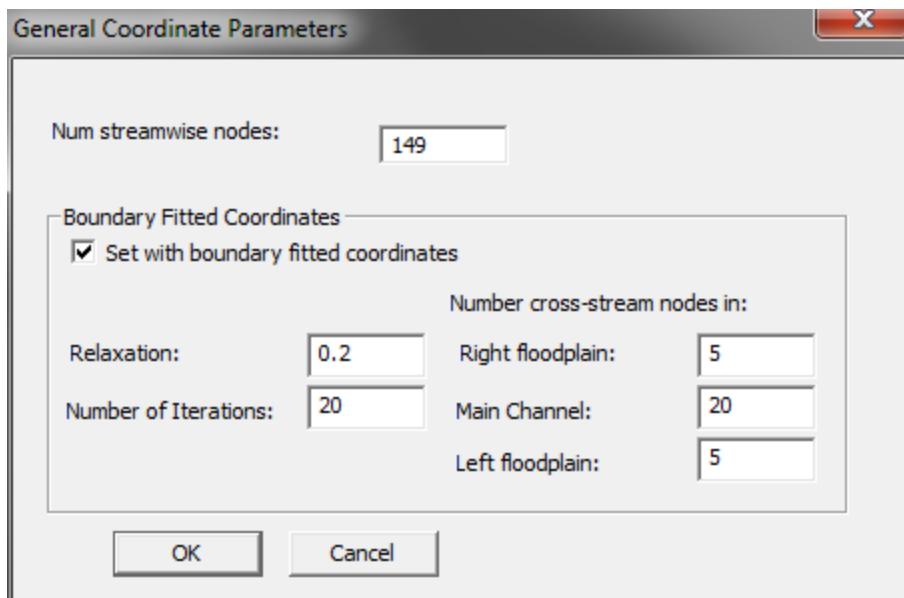
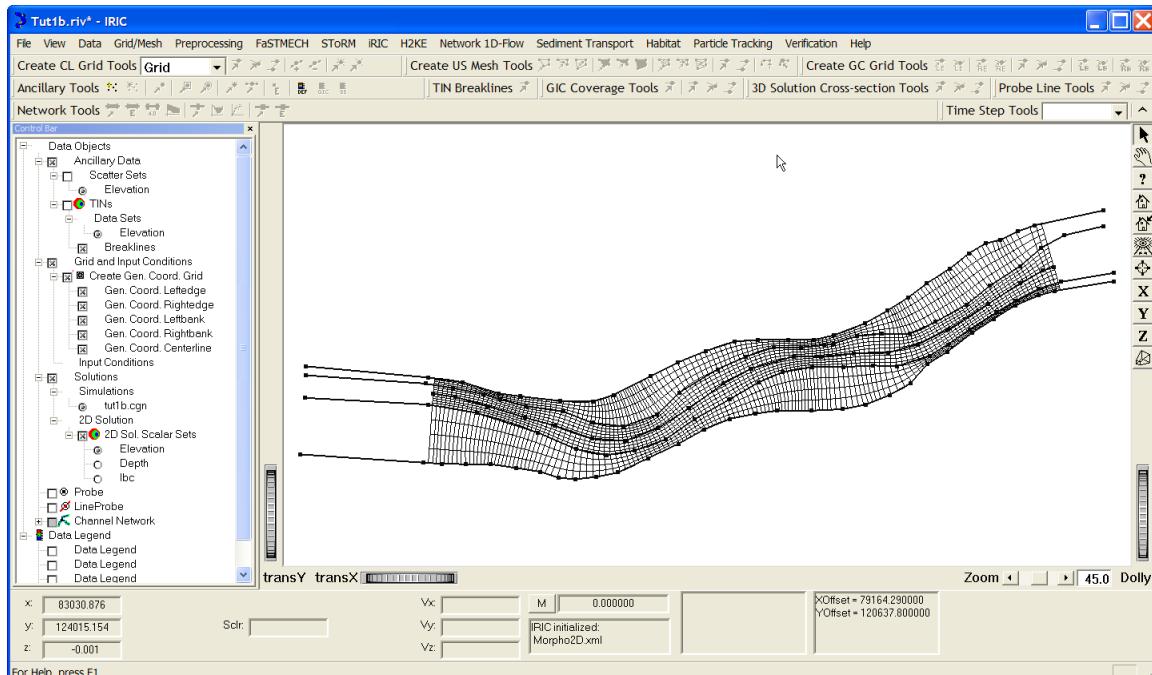


Figure 3.4.1.14 The **Gen Coord. Centerline** has been edited so that the grid rows near the downstream boundary are approximately perpendicular to the flow direction and parallel to the boundary.



A



B

Figure 3.4.1.15 (A) The boundary fitted general coordinate grid parameters and (B) the resulting grid.

4. Map topography to the grid

In the previous step we created a general coordinate grid. This defined the spatial location of the grid, but there are no Input Conditions associated with that grid. The next step is to map the measured topography to the grid. Using the TIN of topography, we can map the topography onto each node of the grid by interpolating the elevation at the location of each grid node from the TIN.

Select **PreProcessing->Set Current Input Condition->Map w/TIN** from the menu. This will do the mapping operation described above. The resulting grid and mapped topography can be viewed as in Figure 3.4.1.14. Make sure the **Grid and Input Conditions / GIC Scalar Sets** is turned on and the **Input Condition** is set to **Elevation**.

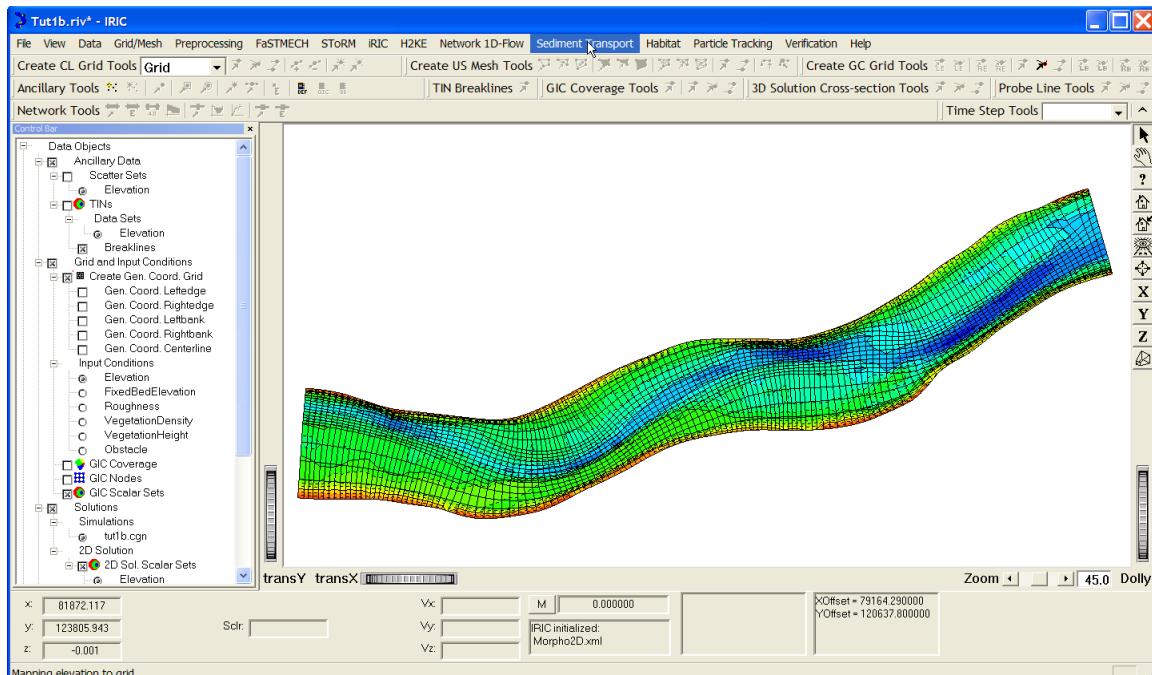


Figure 3.4.1.14 The Elevation mapped to the grid is shown. Note the **Grid and Input Condition** branches that are turned on in the Control bar to obtain this view.

5. Repeat steps 3-5 in the first approach

Repeat steps 3-5 in the first approach and visualize the results of the new simulation using the new grid. Name the new simulation Tut1b.

Extend Model to include variable grainsize and vegetation

In this part, ancillary data of vegetation height, vegetation density and fixed bed elevation will be used to map to the corresponding Grid and Input Conditions respectively. In addition grainsize distributions for the surface or active layer and the sub-surface layer will be used. The simulation can be run for a 72 hour period with a hydrograph that contains multiple peaks. The resulting morphodynamic change and grainsize change can be observed.

Import ancillary data sets including fixed bed elevation an vegetation

Choose **File->Import -> Ancillary Data ->Scalar** from the menu and in the Import Ancillary File dialog double click on the **Vegetation Data Types / Vegetation Height** branch of the data tree and then select the ellipsis button next to the

File to Import text box and browse to the file Vegetation Height.anc. Notice that **VegetationHeight** has been added to the **Ancillary / Scatter Set** branch in the Control Bar. Repeat the previous steps to import the Vegetation Density.anc file. Repeat again to import the Fixed Bed Elevation.anc file. The FixedBedElevation data type can be found in the **Sediment Data Types / FixedBedElevation** in the Import Ancillary File dialog.

Map new ancillary data sets to the corresponding Grid Input Condition

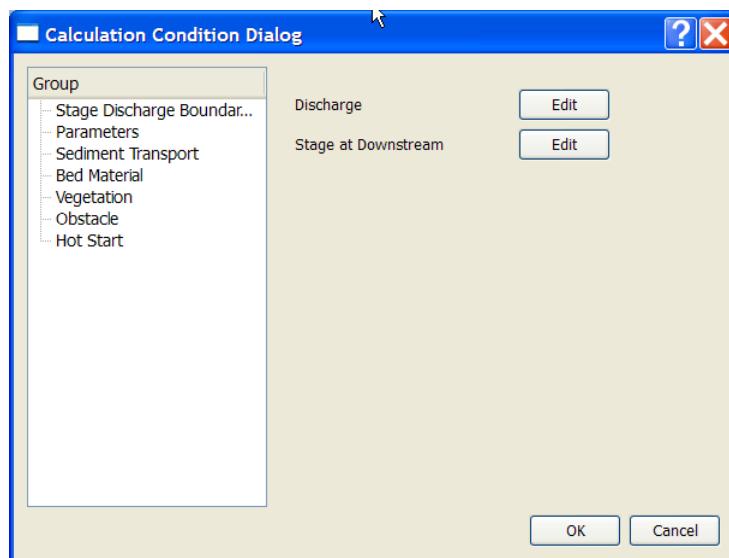
Map each of the ancillary data files imported in Step 10, following the example for Vegetation Height below:

- Select **Ancillary / Scatter Set / VegetationHeight** and select **Grid and Input Condition / Input Condition / VegetationHeight** to ensure that you are mapping like data types.
- Select **Preprocessing -> Set Current Input Condition -> Map w/TIN** from the menu. A TIN of the VegetationHeight ancillary data will be generated and used to map each node of the grid by interpolating the VegetationHeight at the location of each grid node from the TIN

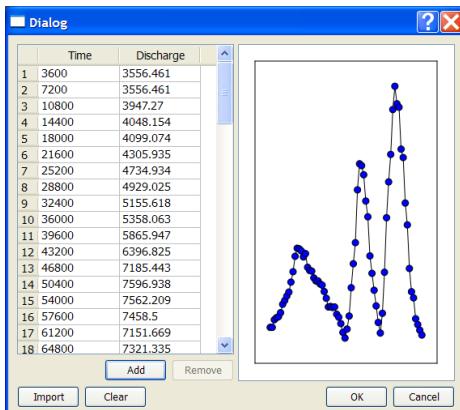
Repeat the above steps for VegetationDensity and FixedBedElevation.

Create new simulation, run the solver and visualize the results.

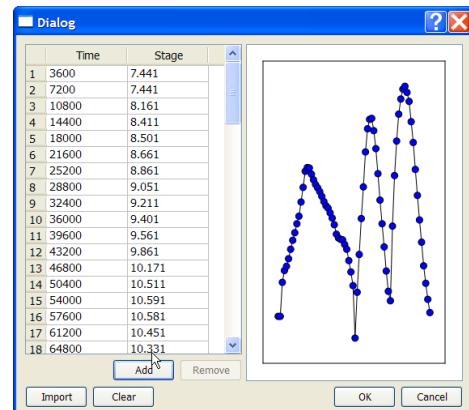
From the menu select **iRIC->Create New Simulation**. In the Save As Dialog type in a name for the simulation (for example, Tut1c). From the menu select **iRIC->Edit Calculation Conditions**. Enter values as shown in Figure 3.4.1.17



A



B) Select the Import button and browse for the Discharge2.txt file.



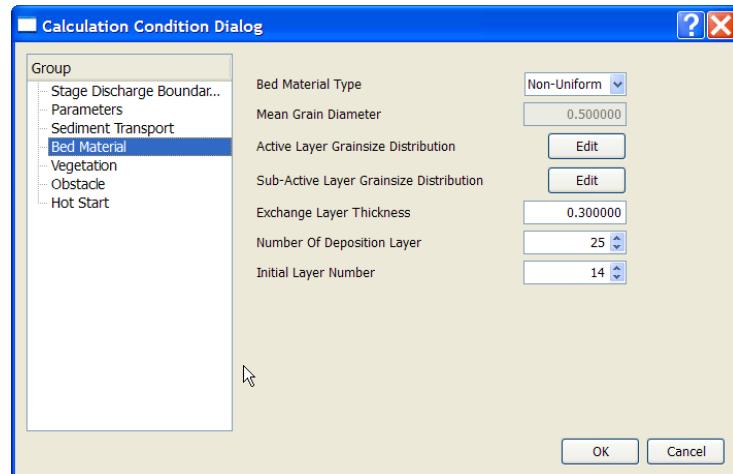
C) Select the Import button and browse for the Stage2.txt file.

This screenshot shows the 'Calculation Condition Dialog' with various parameters set. The 'Group' tree on the left includes 'Stage Discharge Boundary...', 'Parameters', 'Sediment Transport', 'Bed Material', 'Vegetation', 'Obstacle', and 'Hot Start'. The right side shows settings for 'Calculation Type' (set to 'Bed Variation'), 'Start Time' (0.000000e+00), 'End Time' (2.664000e+05), 'Computational Timestep' (0.500000), 'Output Timestep for File' (3600.000000), 'Output Timestep for Screen' (10.000000), and 'Bed Evolution Time' (1000.000000). Buttons for 'OK' and 'Cancel' are at the bottom.

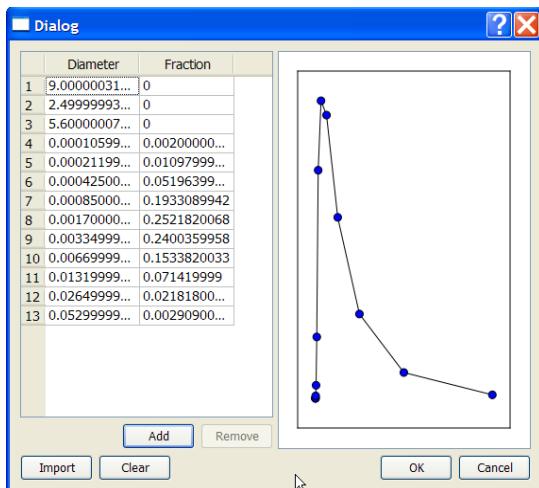
D

This screenshot shows the 'Calculation Condition Dialog' with the 'Sediment Transport' group selected in the 'Group' tree. Parameters shown include 'Secondary Flow Coefficient' (7.000000), 'Sediment Transport Type' (Bed load), 'Suspended Sediment Type' (Lane-Kalinske equation), 'Use Non-Erodable Height' (selected), 'Ratio of Sediment to Equilibrium Sediment Discharge Upstream' (1.000000), and 'Ratio of Sediment DT to Flow DT' (1.000000). Buttons for 'OK' and 'Cancel' are at the bottom.

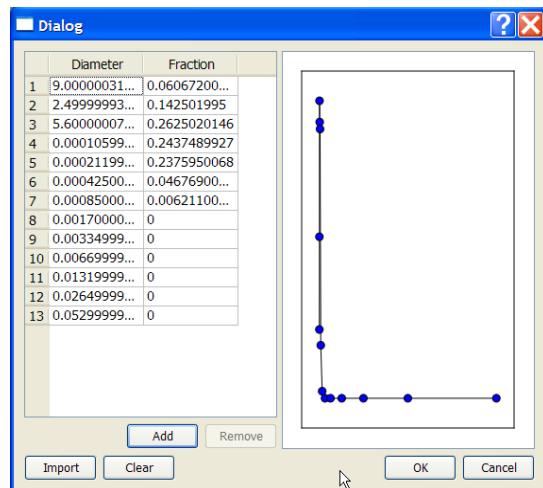
E



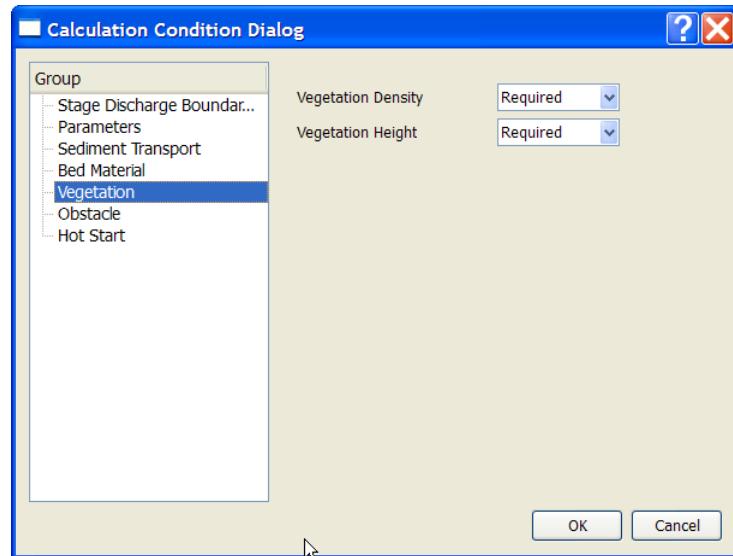
F



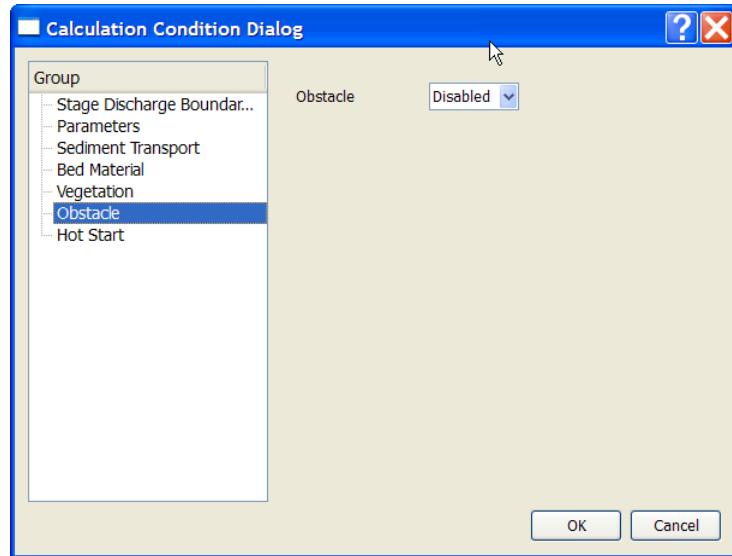
G) Select the Import button and browse for the SurfaceGS.txt file.



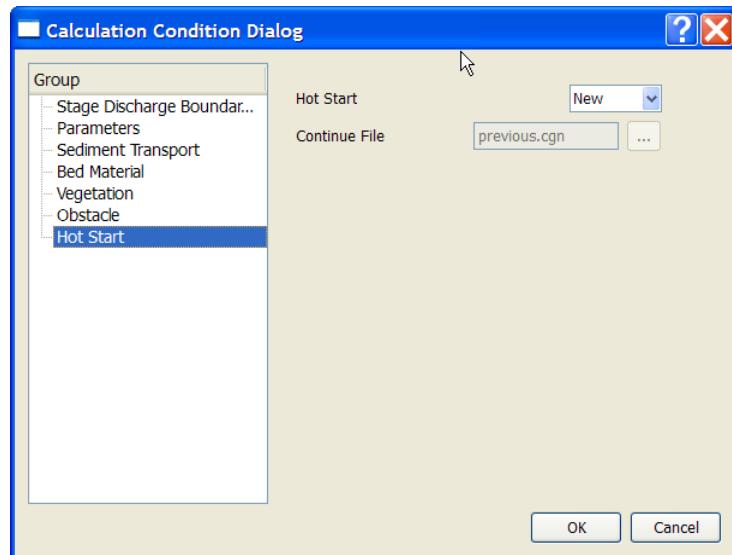
H) Select the Import button and browse for the SubsurfaceGS.txt file.



I



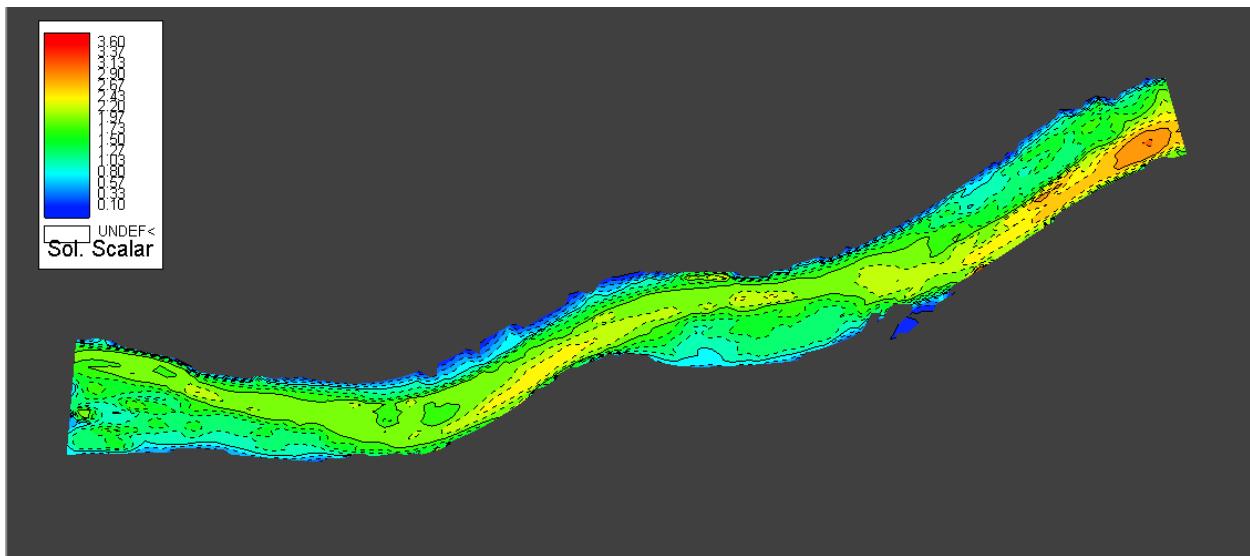
J



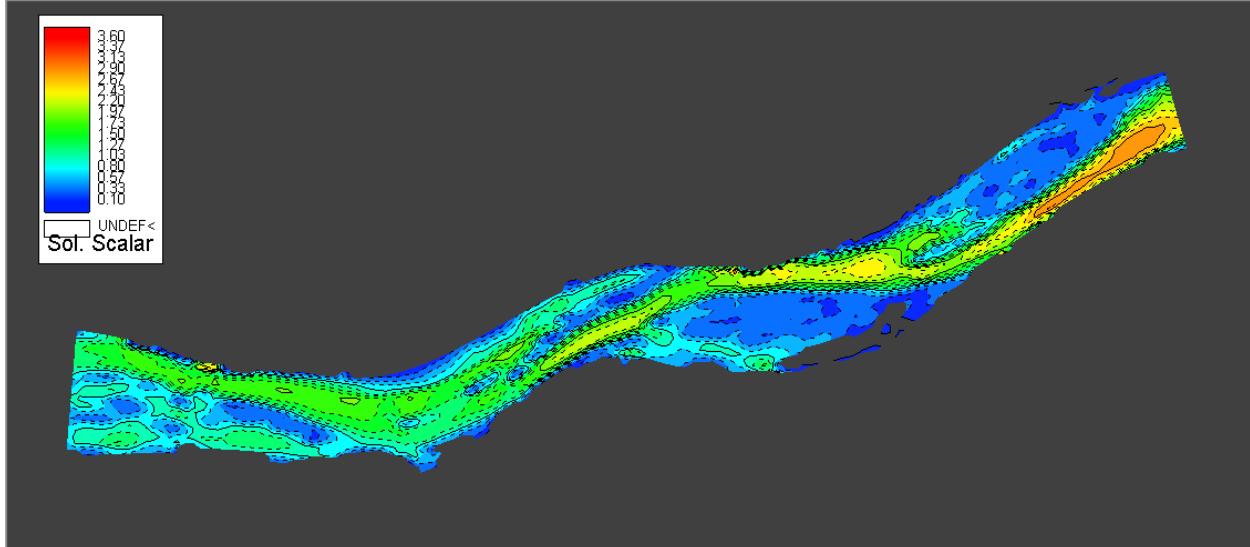
K

Figure 3.4.1.5: (A) Edit the Discharge and Stage at Downstream by selecting each Enter button respectively. In each of the resulting dialogs select the Import button and browse for the Discharge2.txt and Stage2.txt respectively. The resulting dialogs should look like (B) and (C) respectively. The grainsize distribution for the surface and sub-surface layers are entered by selecting the Edit button for each in the Bed Material group (F). Use the Import button and select SurfaceGS.txt and SubsurfaceGS.txt respectively. For each of the following groups enter the parameters as shown in (in the remaining figures).

The results of the two simulations, with and without vegetation, can be visualized by switching between the two simulations in the Control Bar.



Velocity Magnitude without vegetation



Velocity Magnitude with vegetation

Figure 3.4.1.18 For each solution an image of the Velocity Magnitude is generated by selecting **File->Save Special->JPEG** in the menu.

3.5 Nays Tutorials

3.5.1 Nays Tutorial 1 – An introduction to iRIC: flow modeling using the Nays solver; simple meandering channel and morphodynamics.

In this tutorial, the iRIC modeling system and the Nays solver are used to model the flow and sediment transport through a simple idealized meandering channel. The grid and accompanying topography are imported from a comma delimited file exported from the RIC-Nays application. With an imported grid all that is left to do is to specify the calculation conditions for the model, run the solver and visualize the results.

Tutorial 1 steps:

1. Initialize the Nays solver by loading the Nays.xml file.
2. Import comma delimited .grid file exported from the RIC-Nays application and the associated grid topography.
3. Create a simulation and edit the calculation conditions.
4. Run the Nays solver and visualize the results.
5. Modify the cell conditions by adding an obstacle to the flow.
6. Create a new simulation, run the solver, visualize the results and compare to previous simulation without the obstacle.

1. Initialize the Nays solver by loading the Nays.xml file

Start the application by double clicking on the iRIC shortcut from your computers desktop. From the File Menu select **File->Initialize**. In the Open XML Definition File Dialog browse to the iRIC Tutorials\Nays\Tutorial 1 folder and select the Nays.xml file. This file initializes the iRIC interface with the calculation grid and grid conditions required for the Nays solver. As the tutorial progresses we will highlight how the Nays.xml file interacts with and initializes the iRIC modeling interface.

2. Import comma delimited grid file exported from RIC-Nays and the associated grid topography

From the menu select **File -> Import -> Gen. Coord. Grid -> Nays .csv Format**. In the Locate and Select RIC-NAYS Grid File dialog select the meander.csv. This will import the elevation associated with the grid, create a TIN of the elevation, import the grid geometry, and map the elevation to the grid. A series of dialogs are presented as shown in Figures 3.5.1.1 A&B. In each of these dialogs answer in the affirmative. The result of the import operation can be seen in the Control Bar (Figure 3.5.1.1 C). Notice that **Ancillary Data** branch of the Control Bar contains a **Scatter Set** of Elevation and a **TIN** of Elevation. In addition, expanding the **Grid and Input Conditions** branch of the Control Bar reveals the **General Coord. Leftedge**, **General Coord. Rightedge** and **General Coord. Centerline**. Their presence is noted; in Nays Tutorial 2 their function will be documented. Also notice that the Input Conditions are loaded with the required values Elevation and Cell Condition as defined in the Nays.xml file (Figure 3.5.1.1 D).

The Control Bar is used to turn Data Objects On and Off and, where there is an option, to select which value is viewed. For example, selecting the box next to **Ancillary Data / Scatter Sets** will turn the scatter set of Elevation on, to view the entire data set, select the refresh button() on the right side of the iRIC interface which will center the entire Elevation Scatter Set as shown in Figure 3.5.1.2. The **TINs**, and **Grid and Input Conditions** can also be turned On and Off by selecting the select box next to each respectively.

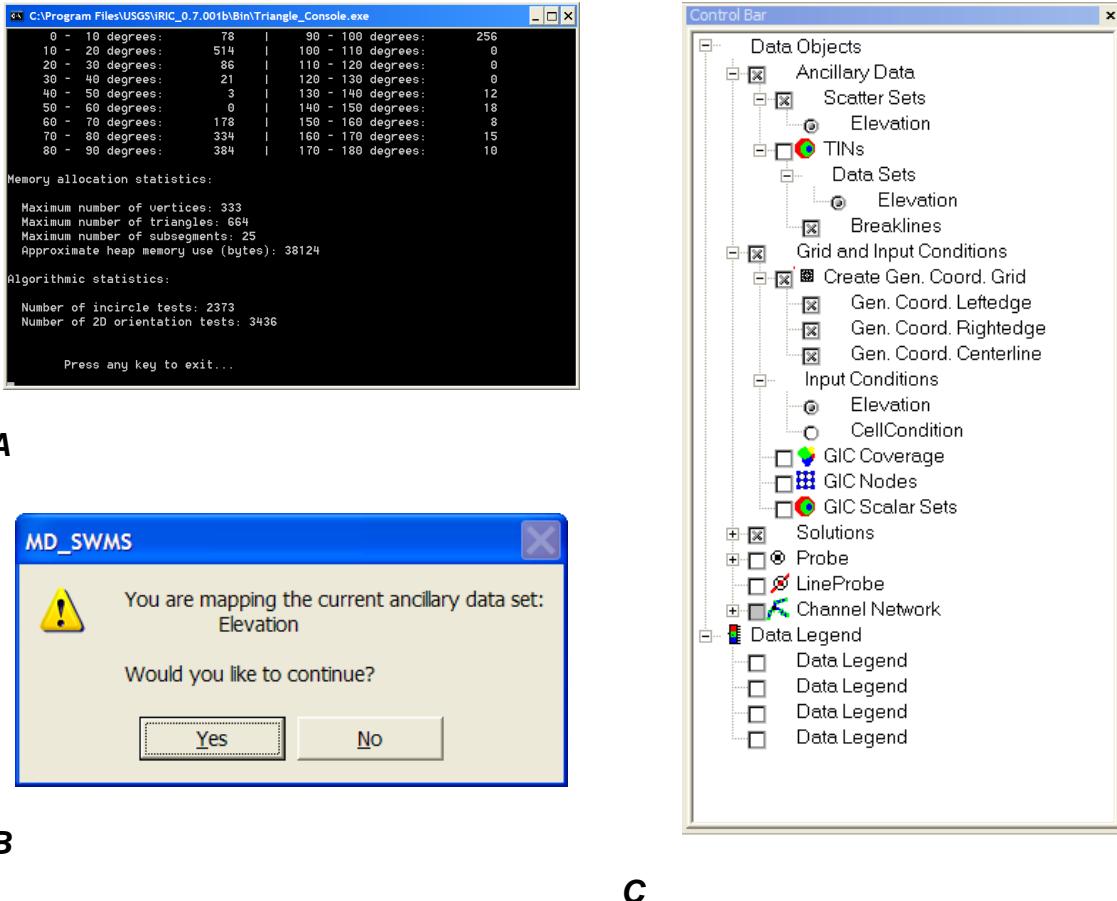


Figure 3.5.1.1: A) By default a TIN of the elevation is created. The console window shows the results of that TIN; pressing any key will close the window. B) By default the elevation will be mapped to the imported grid using the TIN of the elevation data. C) The Control Bar indicates the **Data Objects** created on the import of the General Coordinate System Grid. Note the grid conditions Elevation and CellCondition which are defined in the Nays.xml file.

```

<GridRelatedCondition>
  <!--
    RMCD changed Bed Elevation to Elevation for consistency in MD_SWMS
  -->
  <Item name="aerelevation" caption="Elevation">
    <Definition position="node" valueType="real" default="0"/>
  </Item>
  <Item name="cell_condition" caption="CellCondition">
    <Definition position="node" valueType="integer" default="0" option="true">
      <Enumerations>
        <Enumeration value="0" caption="Normal"/>
        <Enumeration value="1" caption="Obstacle"/>
        <Enumeration value="2" caption="Unerodible"/>
        <Enumeration value="3" caption="LWC_Erodible_without_vegetation"/>
        <Enumeration value="4" caption="HWC Unerodible without vegetation"/>
        <Enumeration value="5" caption="HWC fix-bed vegetation rough"/>
        <Enumeration value="6" caption="HWC fix-bed vegetation middle"/>
        <Enumeration value="7" caption="HWC fix-bed vegetation dens"/>
        <Enumeration value="8" caption="LWC moveable-bed vegetation rough"/>
        <Enumeration value="9" caption="LWC moveable-bed vegetation middle"/>
        <Enumeration value="10" caption="LWC moveable-bed vegetation dens"/>
        <Enumeration value="11" caption="LWC fix-bed vegetation rough"/>
        <Enumeration value="12" caption="LWC fix-bed vegetation middle"/>
        <Enumeration value="13" caption="LWC fix-bed vegetation dens"/>
      </Enumerations>
    </Definition>
  </Item>
</GridRelatedCondition>

```

Figure 3.5.1.2 A piece of the Nays.xml file showing the GridRelatedCondition node that defines the input conditions in the **Grid and Input Conditions** branch of the Control Bar. Note the items are Elevation and CellCondition all of which have been dynamically loaded into the iRIC modeling interface.

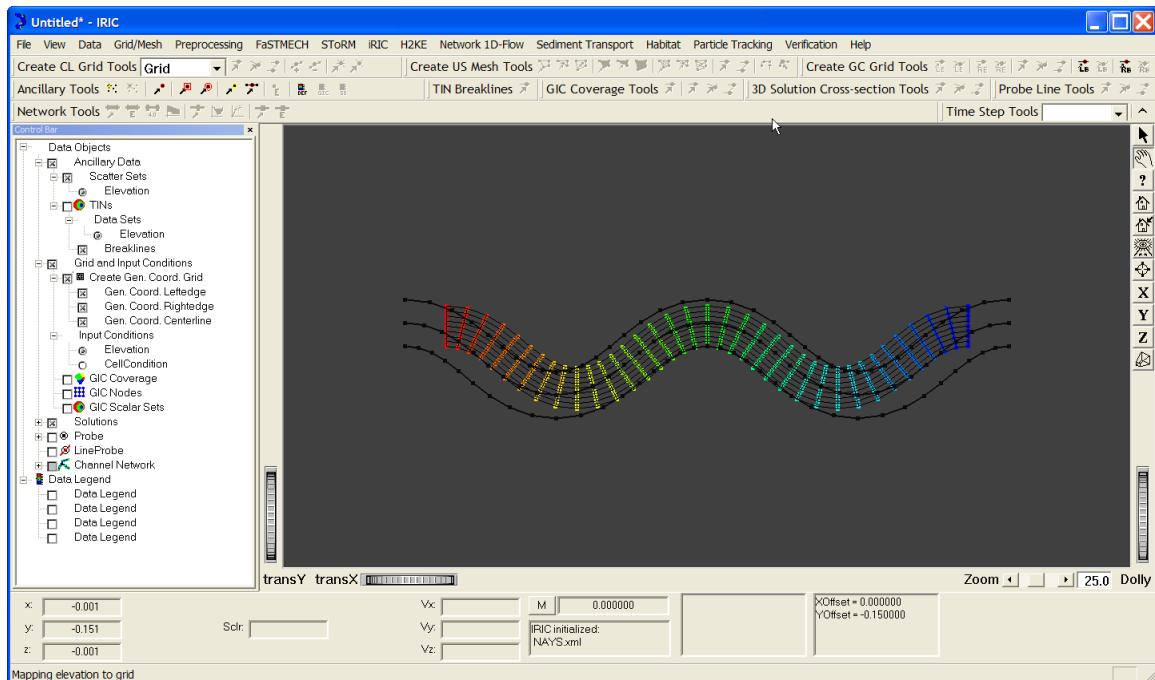


Figure 3.5.1.3 The Ancillary Scatter Set of Elevation that has been centered by selecting the refresh button

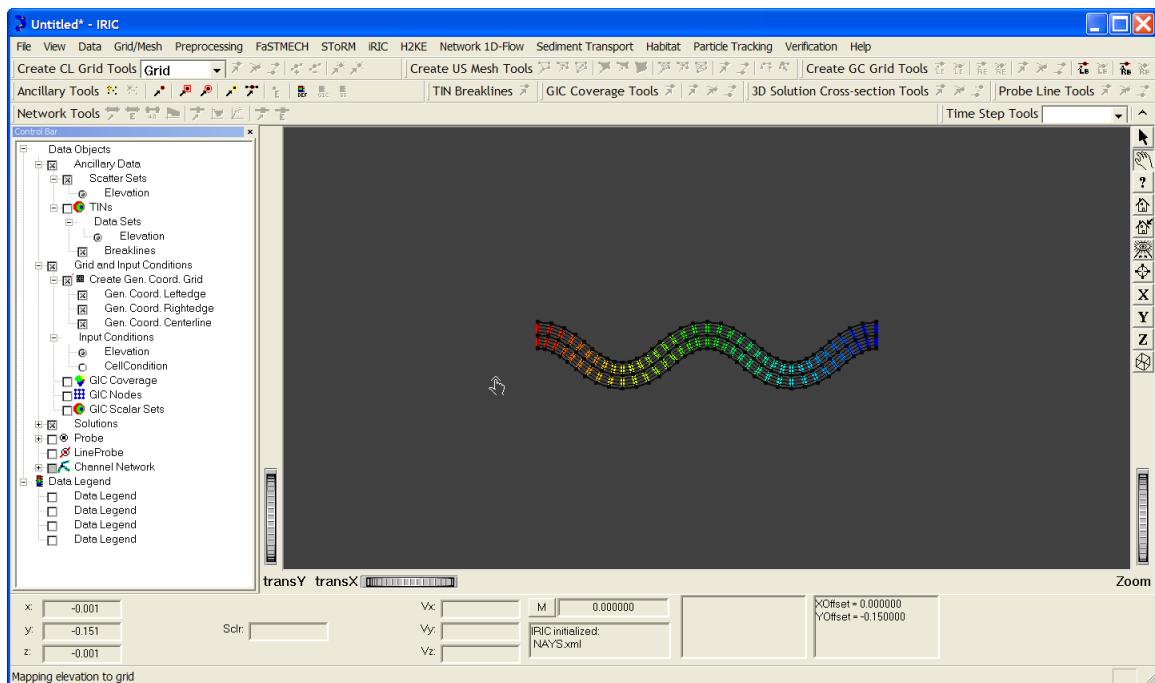
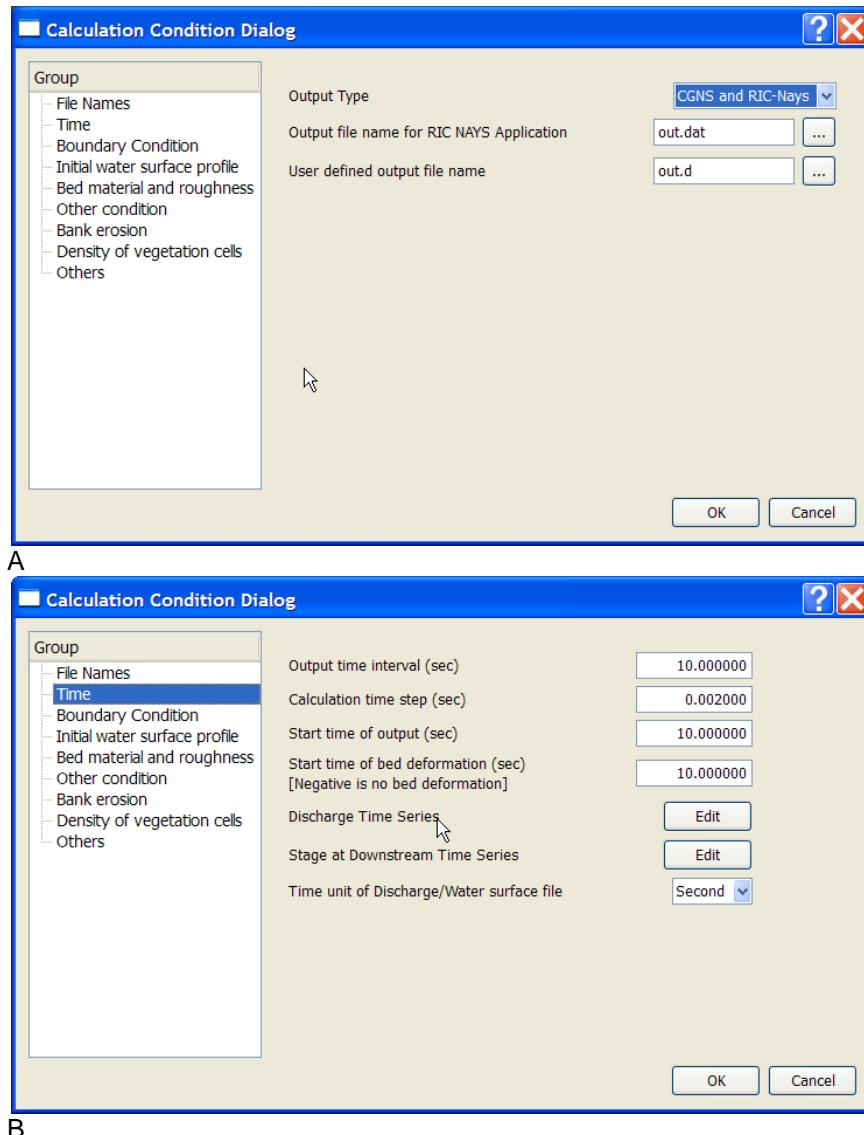
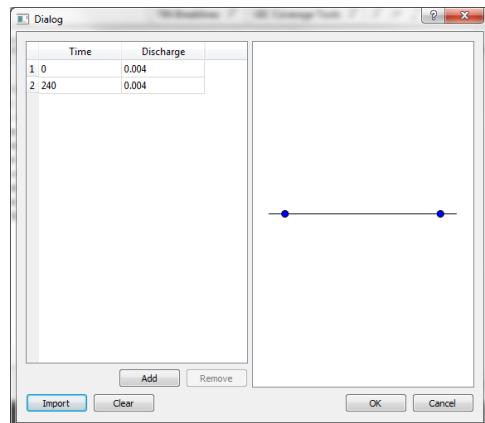


Figure 3.5.1.4 By selecting the you will change the view from a perspective view to a square view which ensures that view is looking straight down everywhere such that graphic elements that are at different elevations in the vertical, such as the grid, boundary line and centerline which are plotted at elevations slightly different from each other, will appear to be precisely on top of each other.

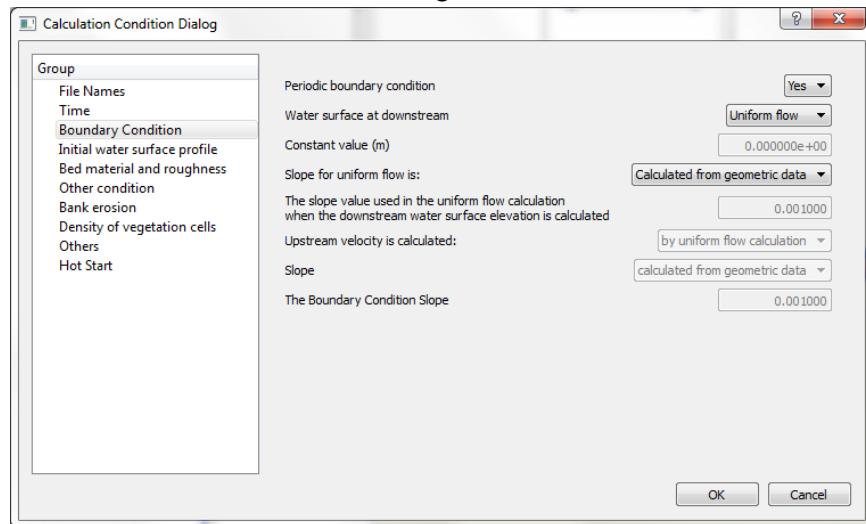
3. Create a simulation and edit the calculation conditions

From the menu select **iRIC->Create New Simulation**. In the Save As Dialog type in a name for the simulation (for example, sim1). From the menu select **iRIC->Edit Calculation Conditions**. The top level group is the File Names. Enter the values as shown in Figure 3.5.1.5.

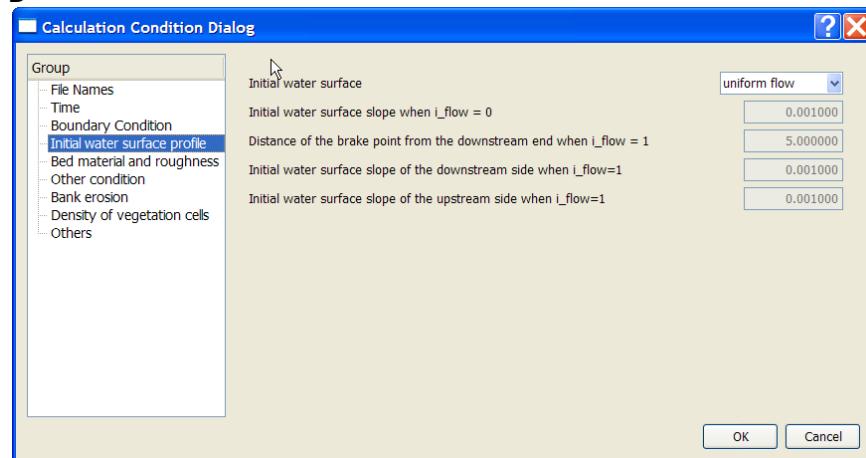




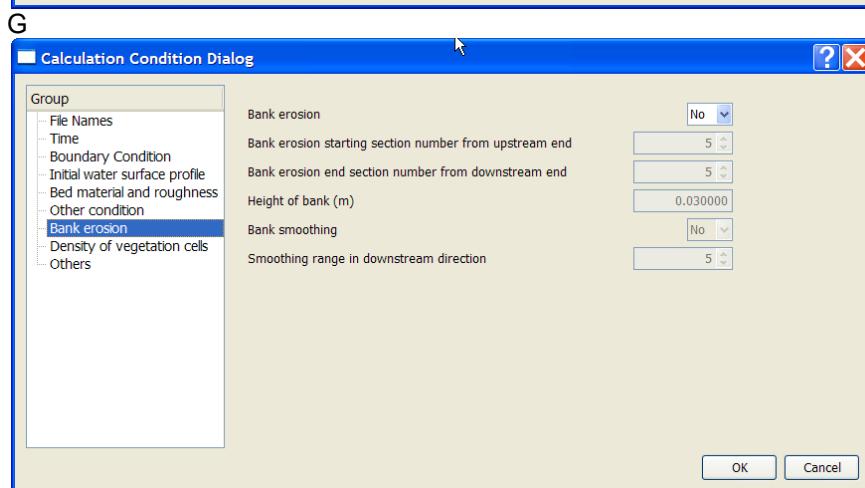
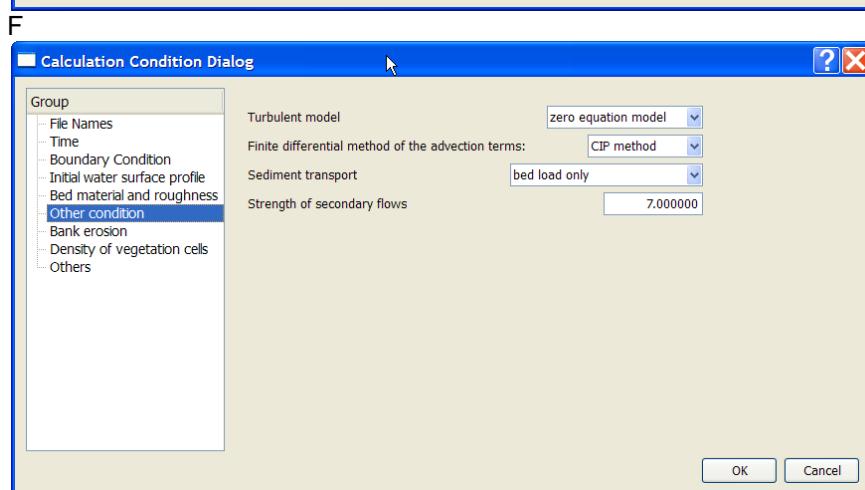
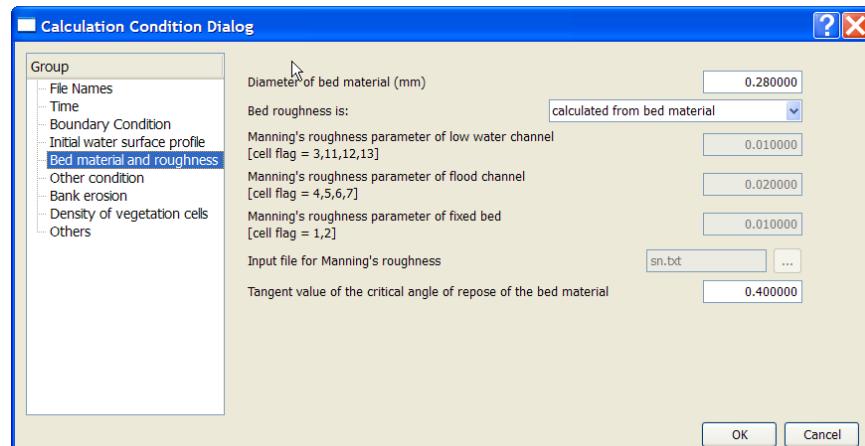
C



D



E



H

J

K

L

Figure 3.5.1.5 For each of the groups enter the parameters as shown. To enter the Discharge time series, in the Time Group (B), Select Edit for the Discharge Time Series value, and in the resulting dialog (C) select the Import Button and browse to the Discharge.txt file in the Nays\Tutorial 1 directory.

4. Run the Nays solver and visualize the results

First save the project by selecting **File -> Save** from the menu and naming the file Tut1.riv. From the menu select **iRIC->Run**. When the solver executes, a new Console window will appear- a snapshot is shown in Figure 3.5.1.6. As defined in the Time Group in the Edit Calculation Conditions dialog the Console window will refresh every 10 seconds of computation time so that the progress can be viewed.

```
C:\Program Files\USGS\iRIC_0.7.001b\Bin\shimizu_Solver.exe
Nays2d Solver Version 3.12 Last updated 2009/2/23
Copyright(C) by Yasuyuki Shimizu, Hokkaido Univ, Japan
Modified by Ichiro Kimura for Dynamic Array Allocationable Version 2009/1/17
10.000 0.0000 -0.0225out
20.000 0.0000 -0.0225out
30.000 0.0000 -0.0225out
40.000 0.0000 -0.0225out
```

Figure 3.5.1.6 An snapshot of the Nays Console window during the execution of the solver. The three columns represent time, discharge, and stage from left to right.

When the computation is completed, the Console window will close and the Solutions branch in the Control Bar will display the available Scalar and Vector values available for viewing. **The 2D Solutions | 2D Sol. Scalar Sets** is set to Velocity and the **2D Solutions | 2D Sol. Vector Sets** is set to Velocity. To scroll through the saved solutions use the Time Step Tools toolbar and using the drop-down list select the solution to view, currently the time step is set to 24, the last time step of the simulation. Save the project using **File->Save**, and give the project a name such as Tut1.

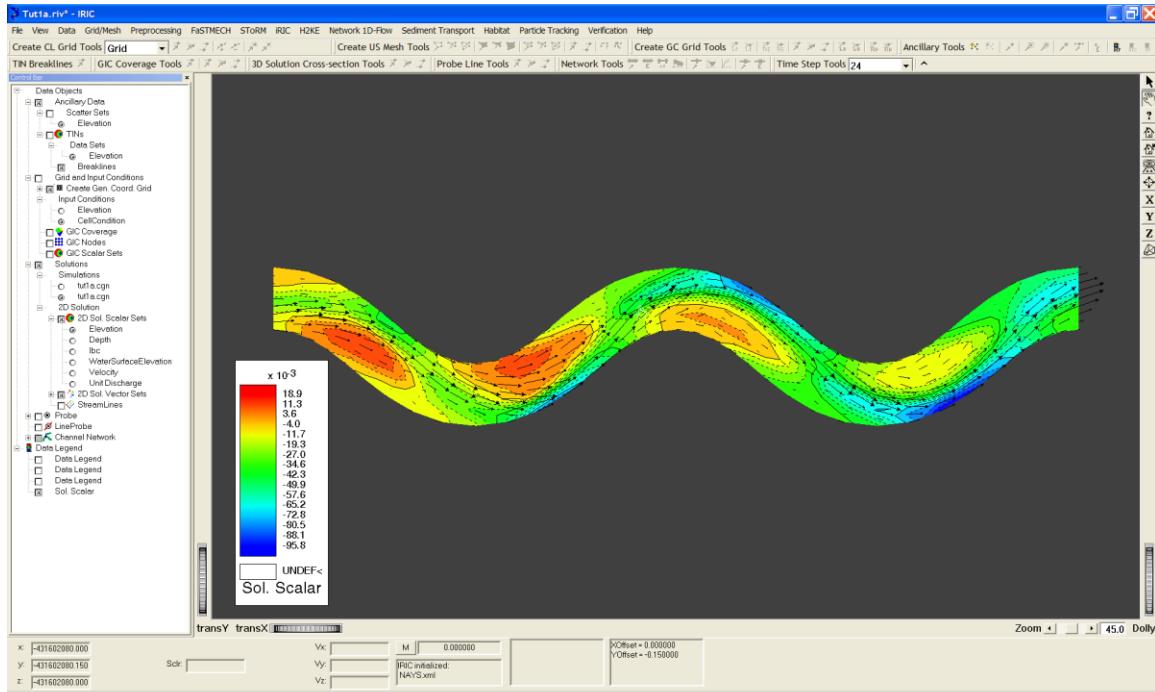


Figure 3.5.1.6 The solution at time step 24. Note that Elevation 2D Sol. Scalar Set is selected and the Velocity 2D Sol. Vector Sets is selected.

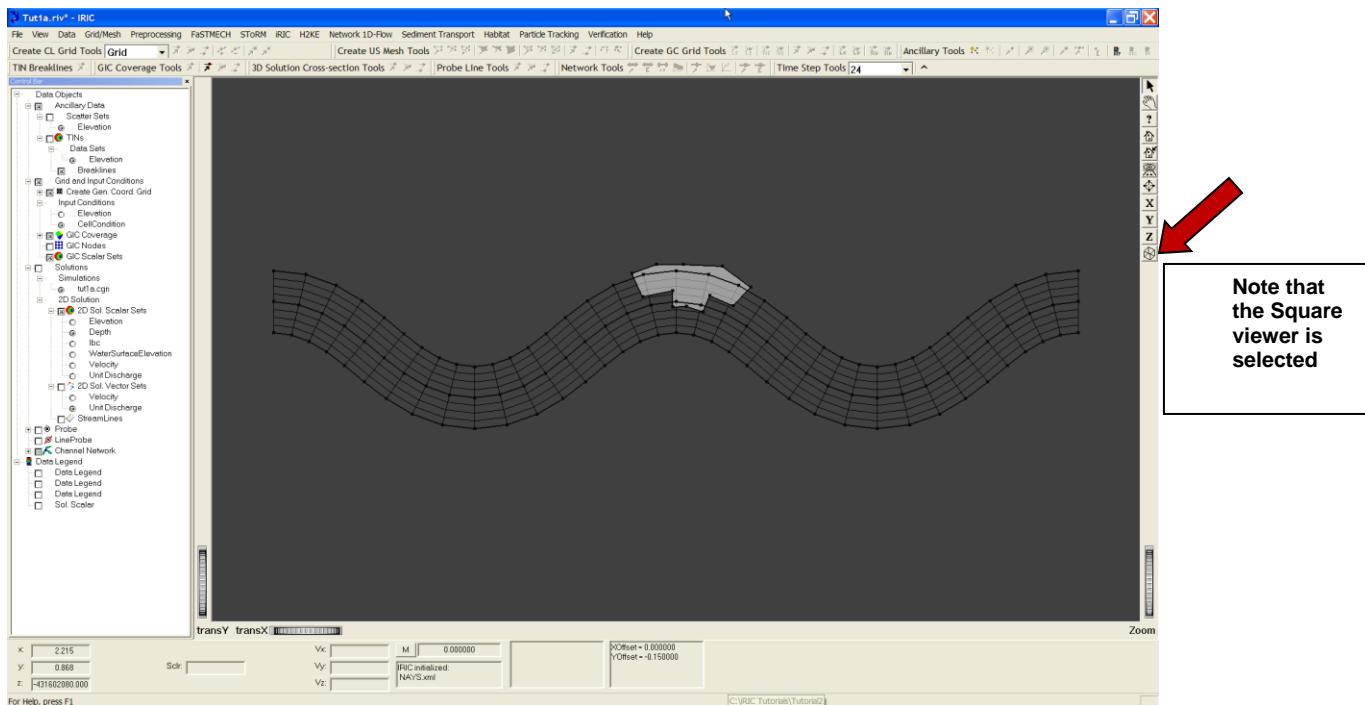
5. Modify the Grid and Input Conditions | Input Condition | Cell Condition to include an obstacle to flow

Create an obstacle to flow at the outside of the first full meander in the reach using the GIC Coverage Tool to create a polygon which will define the value of the nodes surrounded by the polygon. Because the cell conditions are properties of the cell, **the polygon must encompass the entire cell**, or all four nodes of the cell or cells we would like to define as an obstacle. The method is illustrated in the following steps:

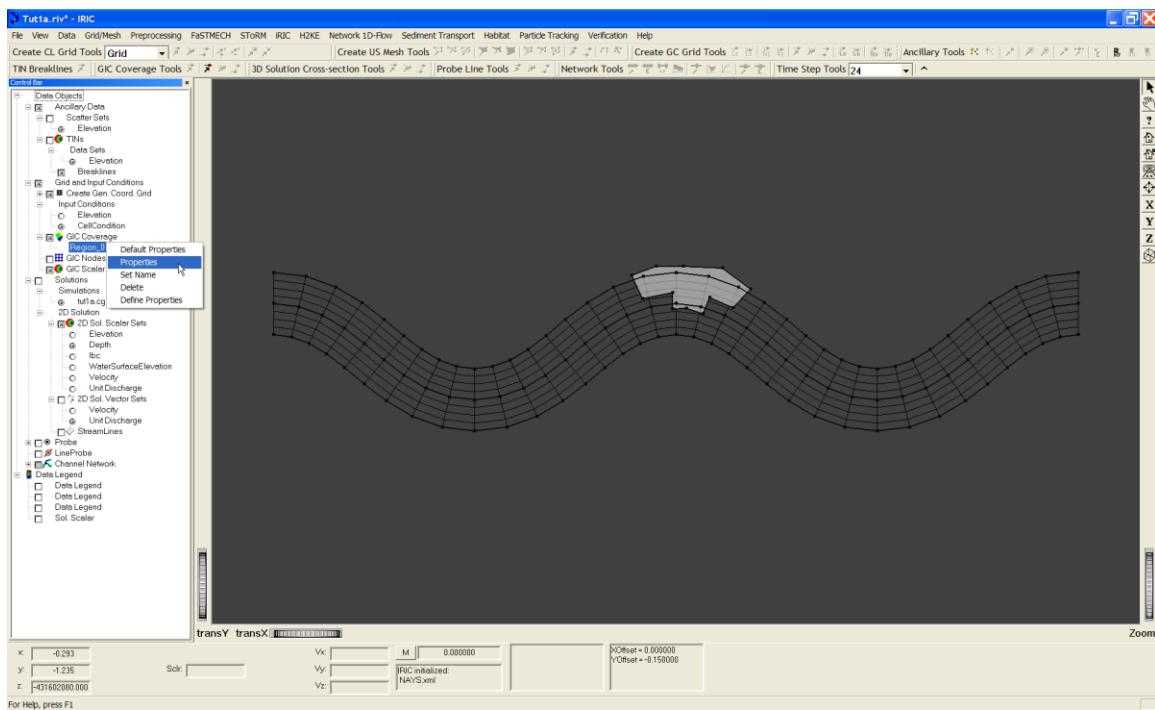
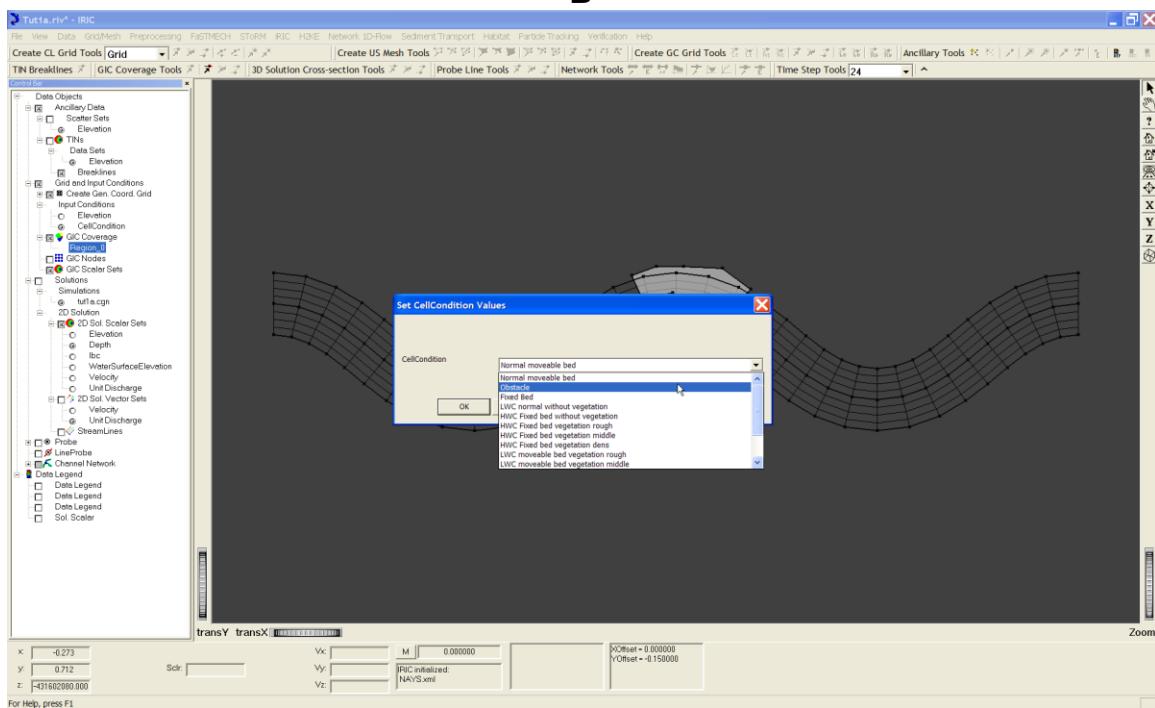
- First select **Cell Condition** in the Grid **and** **Input Conditions / Input Conditions**.
- Turn on the GIC Coverage by selecting the check box. For each Input Condition there is a Coverage that may contain one or more user defined polygons, drawn on the grid and whose property will define the value of each grid node within the polygon. If there is more than one polygon over a grid node, the last polygon drawn will override all underlying polygons.
- To create a new Coverage Region (polygon), select  in the GIC Coverage Tools toolbar. Create a polygon similar to the one in Figure 3.5.1.7A
- To define the Cell value of the polygon, right click on the **GIC Coverage / Region_0** branch in the Control bar (Figure 3.5.1.7B) and in the resulting pop-up menu, select **Properties**.
- In the Set CellConditions Value dialog, select Obstacle in the drop-down menu (Figure 3.5.1.7C).
- The result is shown in Figure 3.5.1.7D.

In addition to the steps described above, more information on GIC Coverage Regions can be found in Section 1.4.3 GIC Coverage Toolbar in the MD_SWMS User's Guide.

To edit the location of any Coverage Region, select the region you would like to edit such as **GIC Coverage / Region_0**. You will see that the polygon now is painted red, which indicates that it is selected to be edited. To move a point in the polygon use the  tool in the GIC Coverage Tools toolbar. Select a point of the polygon you would like to move and holding the left mouse button down, drag the point to a new location.



A

**B****C**

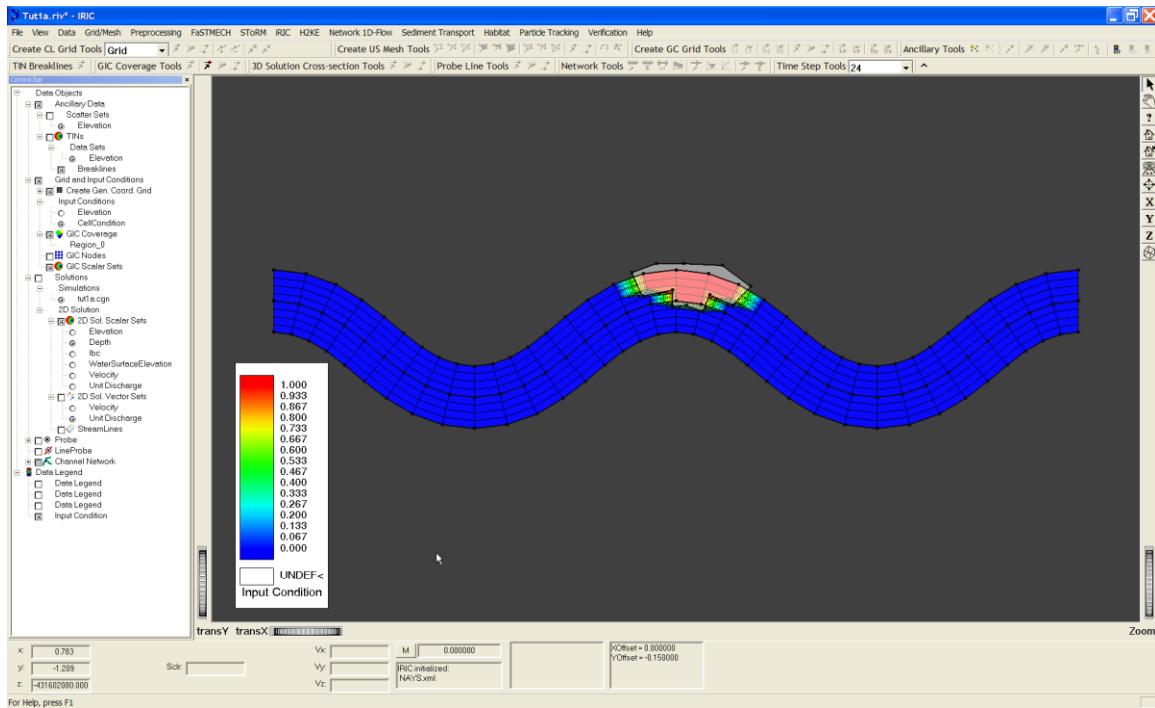
**D**

Figure 3.5.1.7 (A) User define polygon drawn with the Create Coverage Region tool. The polygon is drawn to encompass 8 cells in the outside of the meander bend. (B) Right click on GIC Coverage | Region_0 to define properties of polygon. (C) In the drop-down menu of the Set CellCondition Values dialog select Obstacle. (D) The resulting values of the CellCondition are visualized.

6. Create a new simulation, edit the calculation conditions, and visualize the results.

From the menu select **iRIC->Create New Simulation**. In the Save As Dialog type in a name for the simulation (for example, Tut1b). From the menu select **iRIC->Edit Calculation Conditions**. Enter the same values as in section 3 or Figure 3.5.1.5. From the menu select **iRIC->Run**.

When the computation is completed, the Console window will close and the Solutions branch in the Control Bar will display the available Scalar and Vector values available for viewing. The **2D Solutions | 2D Sol. Scalar Sets** is set to Velocity and the **2D Solutions | 2D Sol. Vector Sets** is set to Velocity. To scroll through the saved solutions use the Time Step Tools toolbar and using the drop-down list select the solution to view, currently the time step is set to 24, the last time step of the simulation. Save the project using **File->Save**.

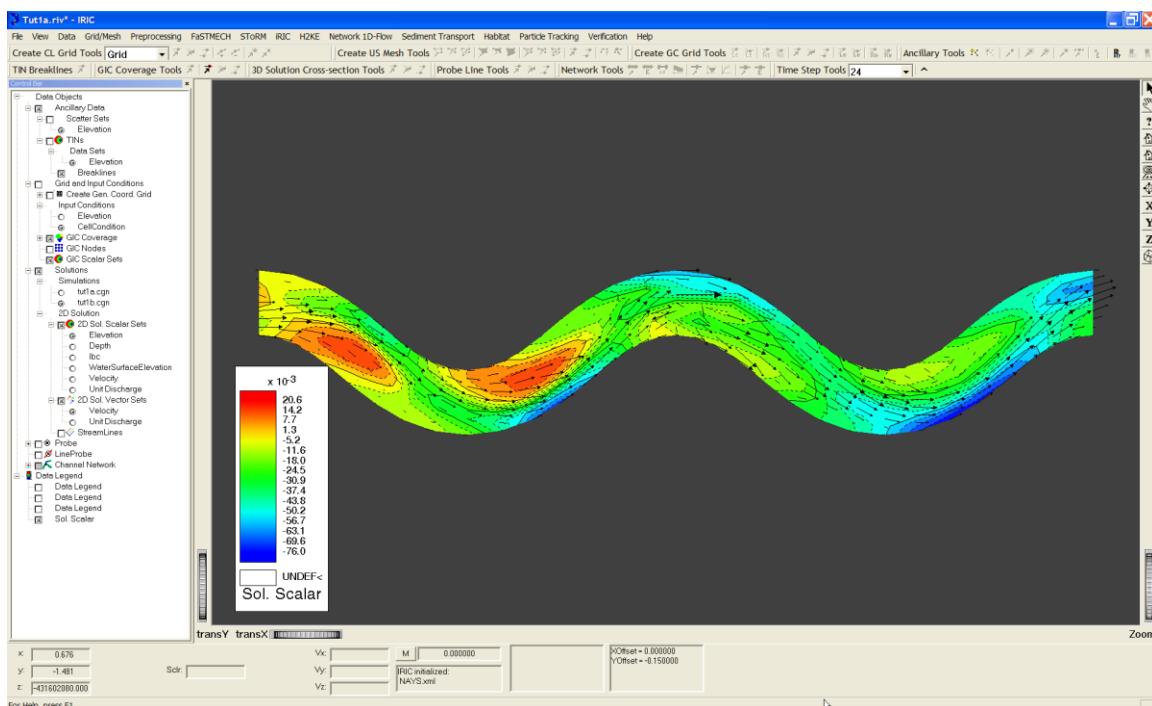


Figure 3.5.1.8 The solution at time step 24. Note that Elevation 2D Sol. Scalar Set is selected and the Velocity 2D Sol. Vector Sets is selected.

3.5.2 Nays Tutorial 2 – An introduction to iRIC: flow modeling using the Nays solver; grid generation and flood hydrograph.

In this tutorial, the iRIC v.1.0b modeling system is used to model the flow through the reach of interest. The tutorial begins by importing topography and then builds a grid using the general coordinate grid generating tools provided in iRIC. The solver is run for a given set of parameters and boundary conditions and the results are visualized.

The following conventions are used in this tutorial to identify Control Bar items or branches, and Menu Items. Menu items are in bold and the path is defined by arrows. Control Bar branches are in italicized bold and the path of the branch is defined by vertical lines. For example:

Menu: **File->Import->Topography**

Control Bar: ***Grid and Input Conditions* / Create Gen. Coord. Grid**

Tutorial 2 steps:

First Approach

1. Create a new project and import topography and background image.
2. Create a grid using iRIC's General Coordinate Grid tools
3. Map topography to the grid
4. Create a simulation and edit the calculation conditions
5. Run the Nays solver and visualize the results

Second Approach

1. Build grid using IRIG curvilinear orthogonal grid tools
2. Repeat steps 3 – 5 in the first approach

First Approach

1. Create a new Project and import the topography and background Image

Begin by selecting **File->New** from the Menu. This will reinitialize the interface to begin a new project. If the iRIC application has been closed and re-started, then you will need to re-initialize the solver definition.xml file, by selecting **File->Initialize** from the menu and the selecting the Nays.xml file in the iRIC Tutorials\Nays\Tutorial 2. From the Menu select **File->Import->Topography**. In the resulting Select Topography File to Import dialog, select the Ishikari.tpo file. The Console window for the TIN application will appear, selecting any key as noted in the console window will close it. Another dialog, the Project File Folder will appear with the path to the Ishikari.tpo file. This indicates that all the data required for the project can be found in this folder, and all data files created by the iRIC application will be saved to the directory, select OK to use the default directory. Note that in the Control bar that only the Ancillary branch has data associated with it, in this case a Scatter Set of Elevation and a TIN of Elevation. To import an image from the menu select **File->Import->Ancillary Data->Image** and open the Ishikari.jpg file. Select the check box next to **Image Sets** and **Ishikari.jpg** in the Control bar to put the image into the background.

2. Create a grid using iRIC's structured grid tools including the curvilinear orthogonal grid and boundary fitted general coordinate grid.

This version of iRIC has tools for generating three different types of grids, two of which are structured, and one of which is unstructured. Nays can be used with both, a structured but general coordinate system and a structured curvilinear orthogonal coordinate system. In the first approach you will build a grid using the General Coordinate Grid tools and in the second approach you will use the Curvilinear Orthogonal Grid tools.

3. Create a grid using iRIC's General Coordinate Grid tools

Select **Grid/Mesh->2D General Coordinate Grid** from the menu. This will activate the general coordinate grid generating tools located in the Create GC Grid toolbar (Figure 3.5.2.1).

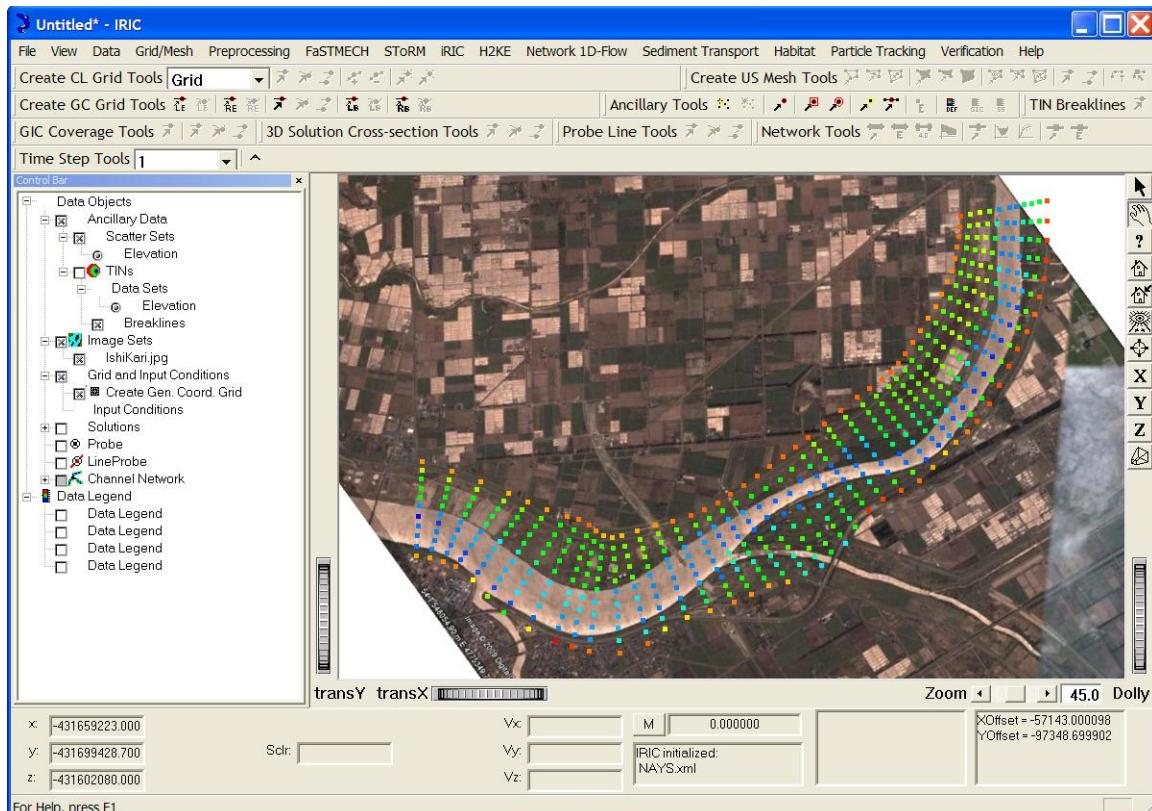


Figure 3.5.2.1 Notice that the **Grid and Input Condition** and the **Create Gen. Coord Grid** branches in the Control bar are turned on and in the Create GC Grid Tools two buttons have been activated.

The following steps will illustrate how to create a general coordinate grid using the tools provided in iRIC. All the tools used are located in the Create GC Grid Tools toolbar (Figure 3.5.2.2). When the mouse is placed over any toolbar button, the status bar at the bottom of the iRIC interface contains information about that button.

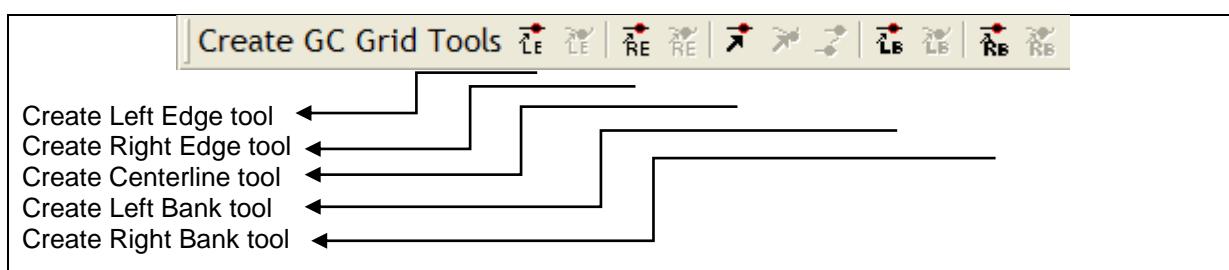


Figure 3.5.2.2 The Create GC Grid Tools. These tools are used to define the shape and extent of the general coordinate grid.

- Create Left Edge Line:** Create a left edge line from upstream to downstream, in this case from upper right to lower left. Select the Create Left Edge tool and use the left mouse button to define individual points creating a line that defines the left edge of the model domain, finishing the left edge line by selecting Enter. Left and right are defined relative to looking downstream. The resulting left edge line should look like that in Figure 3.5.2.3, beginning well upstream and ending well downstream of the measured topography. To zoom or pan

the topography data during the polygon creation simply use the Dolly or TransX or TransY wheels at any time. Alternatively, using the Alt key with the left mouse button will zoom and Alt + Ctrl with the mouse button will pan.

- B. **Create Right Edge, Left Bank and Right Bank lines:** Repeat the process for creating the left edge line as above for the Right Edge, Left Bank, and Right Bank lines. Use Figure 3.5.2.3 and 3.5.2.4 to guide the placement of individual lines, or use the topography and the background image as a guide.
- C. **Create Centerline:** Create a centerline from upstream to downstream, in this case from upper right to lower left. Select the Create Centerline tool and using the left mouse button select centerline points and finish the centerline by selecting Enter. Your centerline should look like that in Figure 3.5.2.3 and 3.5.2.4. The centerline points must be within the Left Bank and Right Bank lines. From the centerline, cross-sections are drawn perpendicular to the centerline out to the edge of the left and right bank lines.
- D. **Define Grid Parameters:** Define the number of nodes in the streamwise and cross-stream directions by selecting **PreProcessing->Set 2D General Coord. Grid Parameters** from the menu. Set the Num. Streamwise Nodes to 85. Leave the Set with Boundary Fitted Coordinates box unchecked at this time. The result shows the intersection of each streamwise node with the bank and edge lines, and can be used as a guide to when editing the grid so the boundaries are within the topographic data and the grid boundaries are perpendicular to the incoming and outgoing flow direction.
- E. **Edit Centerline Points:** You may notice that at either the upstream or downstream Boundary there are problems with the grid extending out of the bounds of the measured topography, as in Figure 3.5.2.5. Edit the centerline by first selecting with the mouse the **Gen. Coord. Centerline** branch in the Control Bar. This will paint the centerline red which indicates that it is in edit mode, and the Edit Centerline tool will be enabled. Using the Edit Centerline tool, select the centerline point with the left mouse button and holding the button down drag the point to a new location. Note that the refresh rate of the graphics may be slow when you drag the point to a new location. Keep holding the mouse button down and drag slowly to the new location. You will have to reselect the tool each time to use it. The geometry of the grid should have rows that are parallel to the upstream and downstream boundaries as in Figure 3.5.2.5 and 3.5.2.6
- F. **Create Boundary fitted Grid:** Select **PreProcessing->Set 2D General Coord. Grid Parameters** from the menu. Enter the parameters as in Figure 3.5.2.7A and select OK. It may take a few minutes for the grid to generate. The resulting grid is show in Figure 3.5.2.7B.

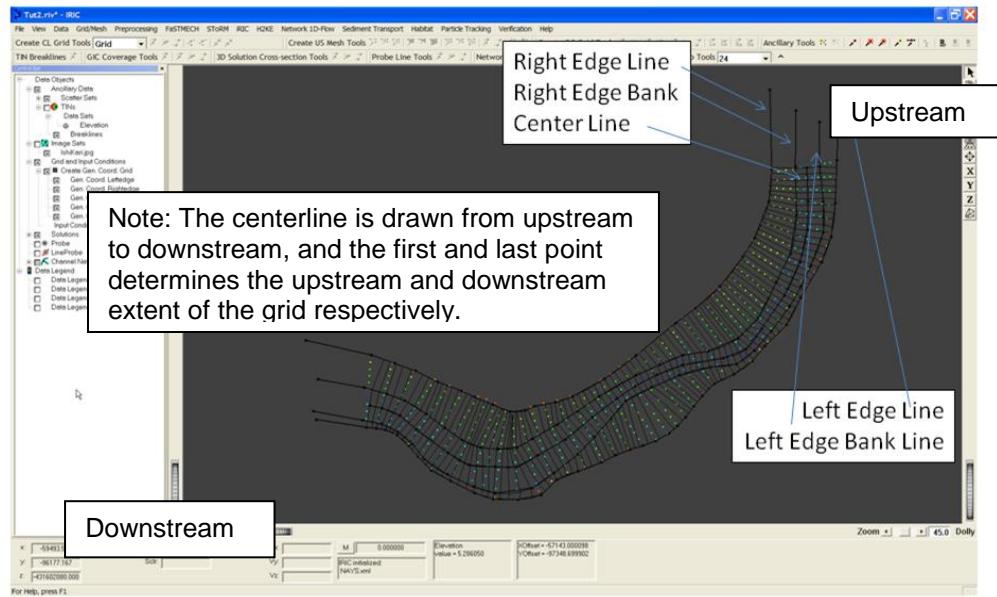


Figure 3.5.2.3 The centerline and the edge and bank lines are shown. Note that the edge and bank lines are drawn such that they extend well beyond the bounds of the measured topography. Note also that the centerline is drawn within the bounds of the measured topography.

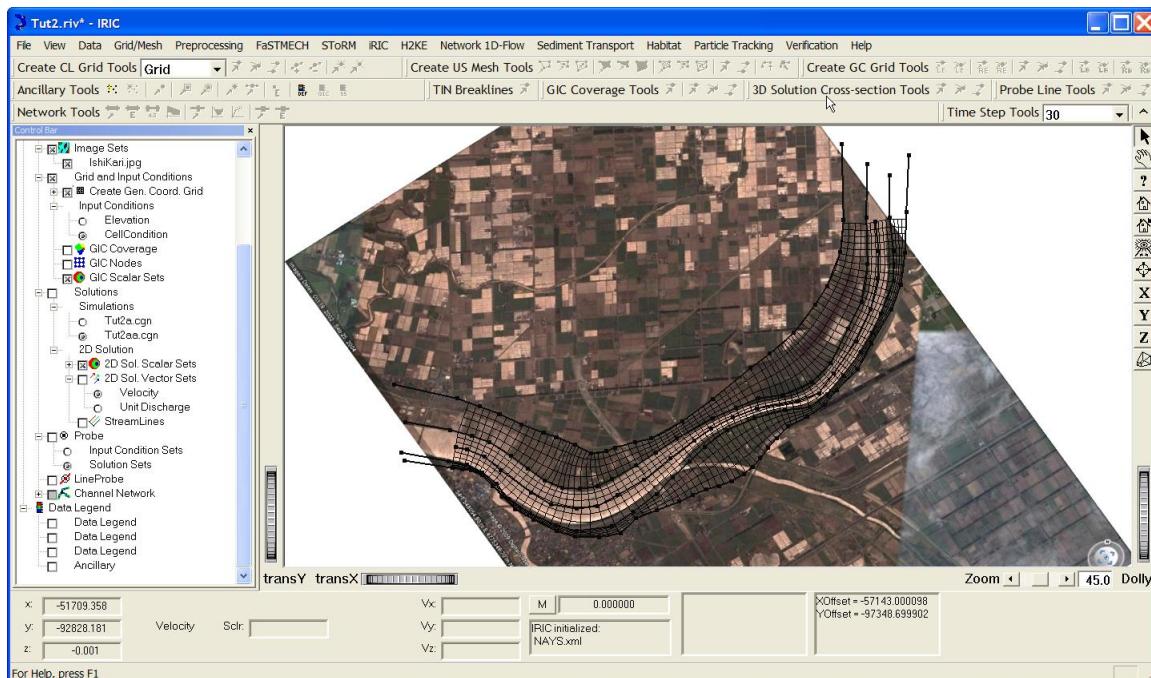


Figure 3.5.2.4 The left and right edge lines define the boundary of the grid. The left and right bank lines define the boundary bankfull or non-flooded channel margins. The centerline follows the flow path of the channel.

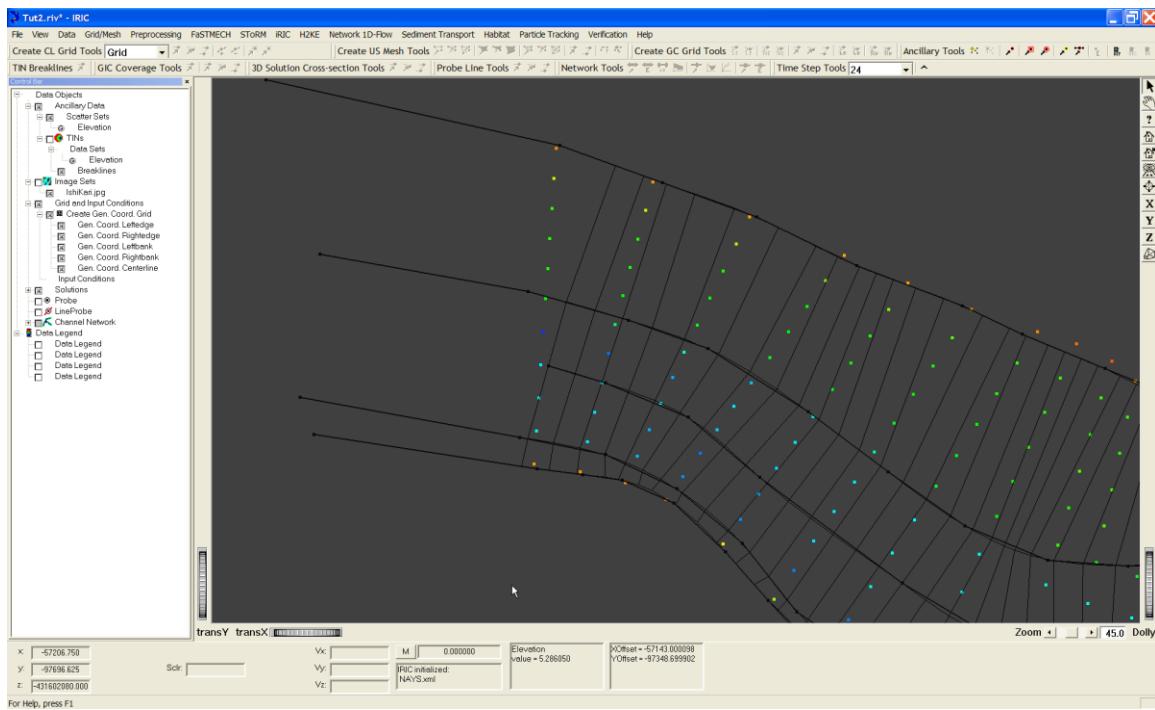


Figure 3.5.2.5 Often the **Gen Coord. Centerline** needs to be edited at the upstream or downstream boundary so that the rows of the grid are perpendicular to the flow direction and approximately parallel to the boundary. Follow the directions in Step 2A above to edit the grid so that the result looks like that in Figure 3.5.2.6.

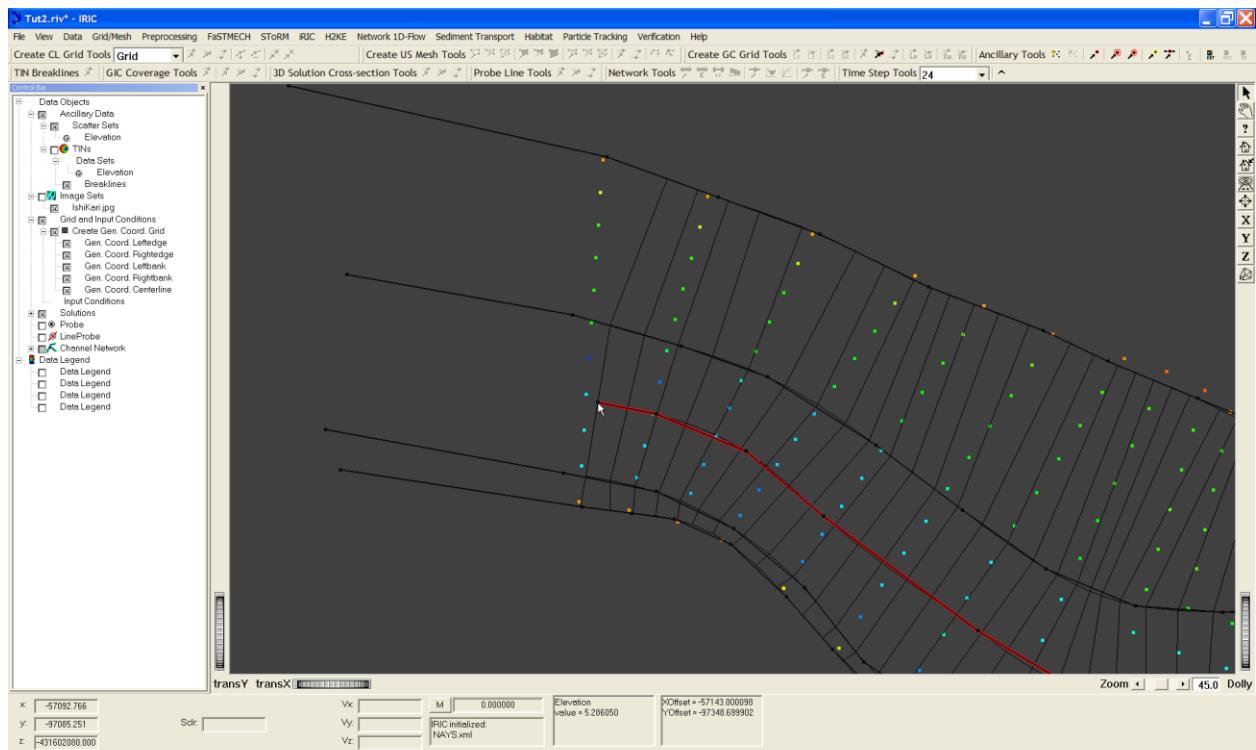
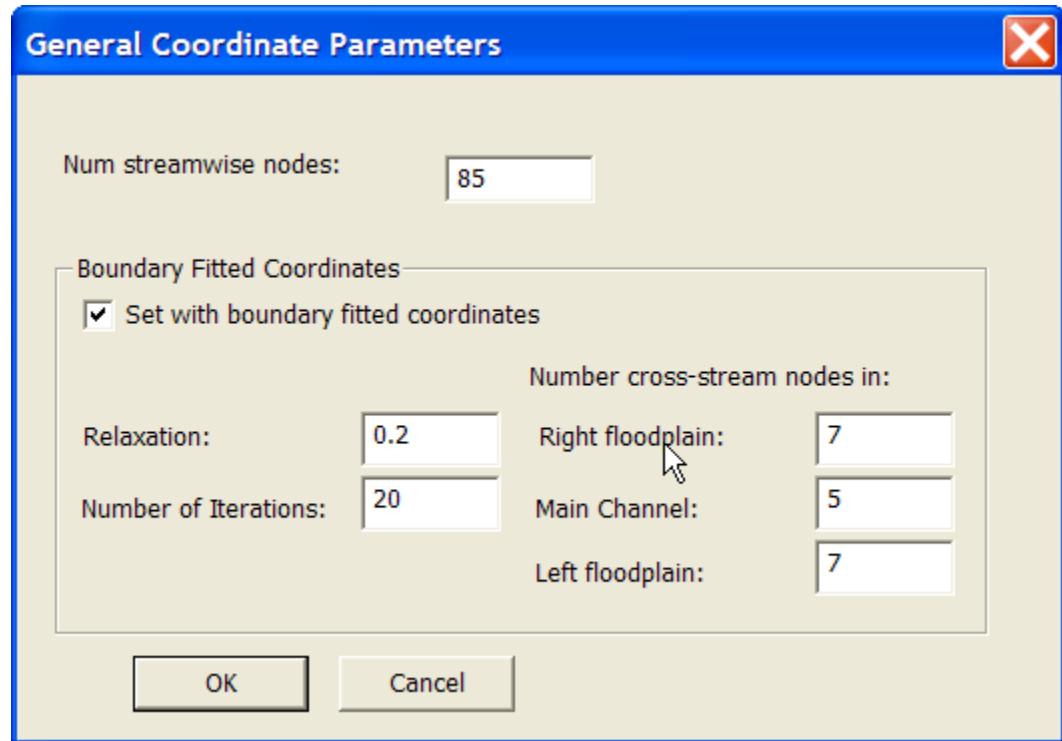
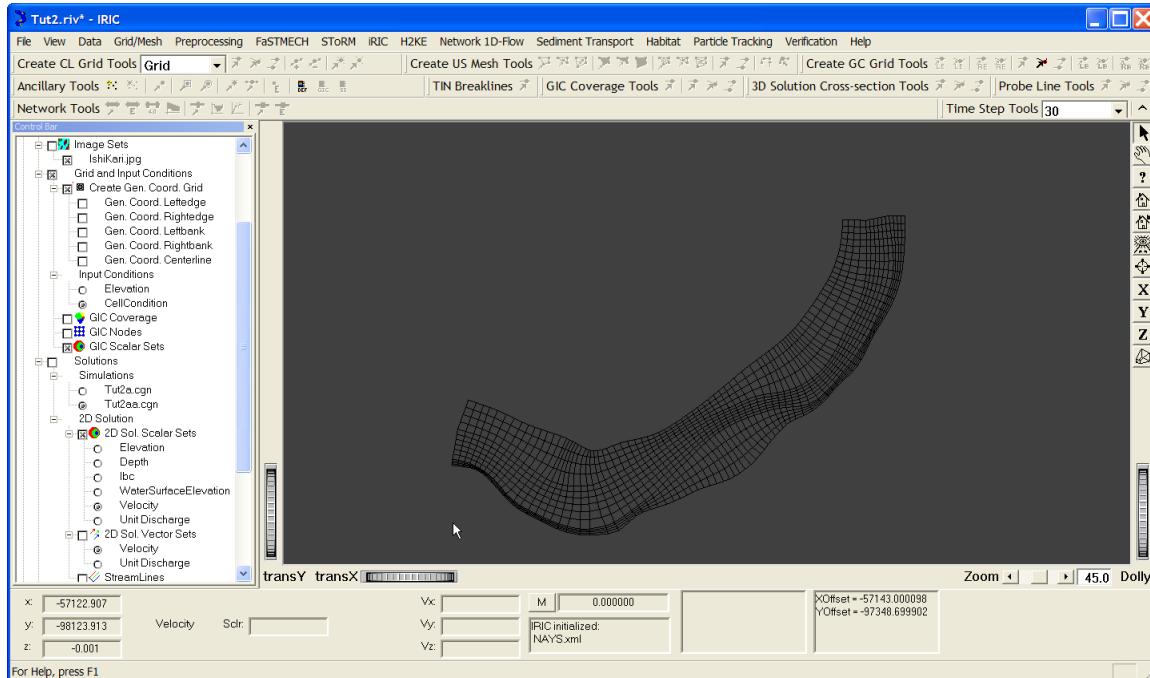


Figure 3.5.2.6 The **Gen Coord. Centerline** has been edited so that the grid rows near the downstream boundary are approximately perpendicular to the flow direction and parallel to the boundary.



A



B

Figure 3.5.2.7 (A) The boundary fitted general coordinate grid parameters and (B) the resulting grid.

3. Map topography to the grid

In the previous step we created a general coordinate grid. This defined the spatial location of the grid, but there are no Input Conditions associated with that grid. The next step is to map the measured topography to the grid. Using the TIN of topography, we can map the topography onto each node of the grid by interpolating the elevation at the location of each grid node from the TIN.

Select **PreProcessing->Set Current Input Condition->Map w/TIN** from the menu. This will do the mapping operation described above. The resulting grid and mapped topography can be viewed as in Figure 3.5.2.10.

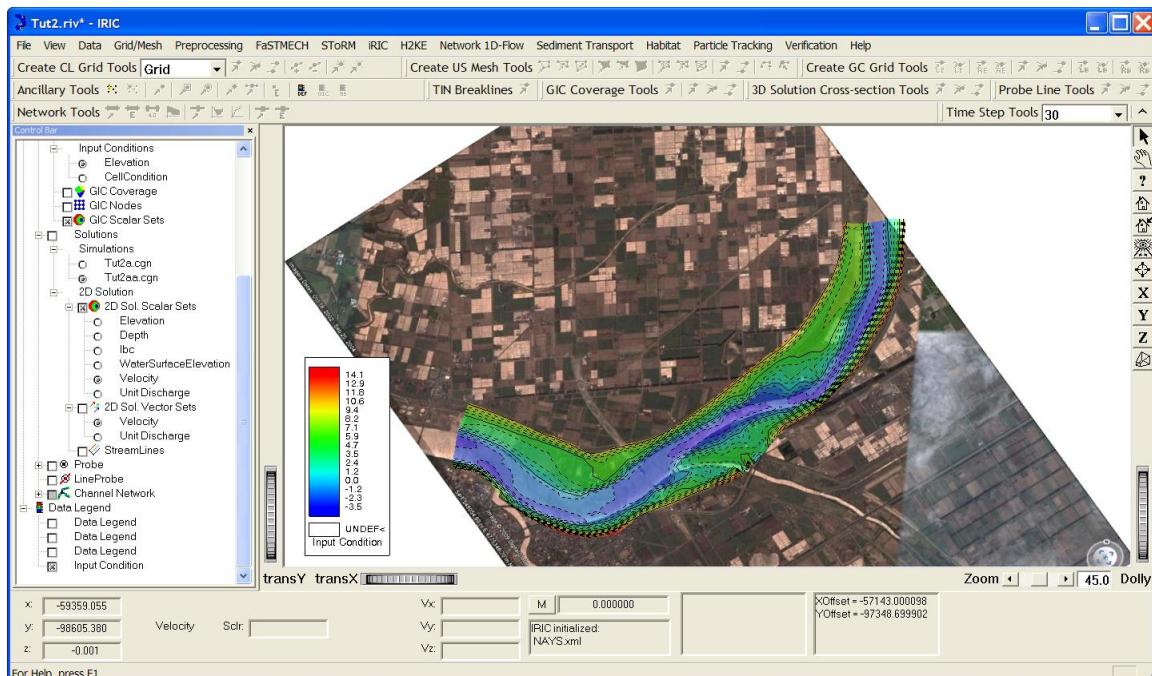
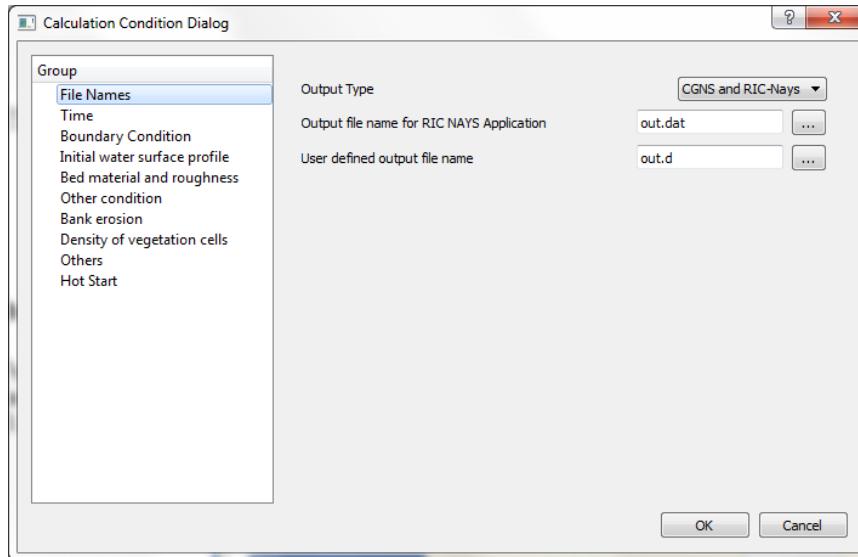


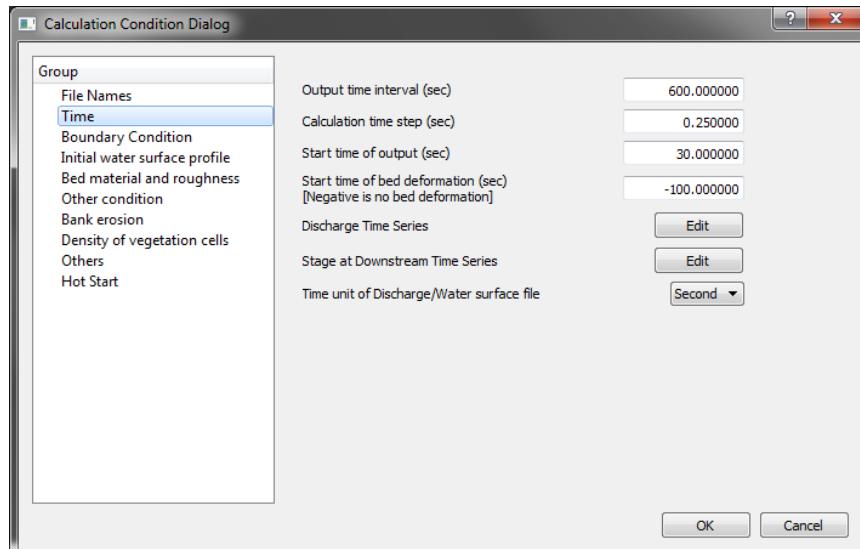
Figure 3.5.2.10 The Elevation mapped to the grid is shown.

4. Create a simulation and edit the calculation conditions

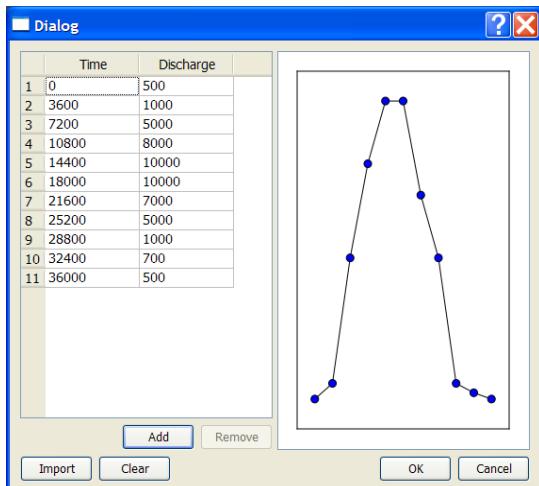
From the menu select **iRIC->Create New Simulation**. In the Save As Dialog type in a name for the simulation (for example, sim1). From the menu select **iRIC->Edit Calculation Conditions**. Enter the values as shown in Figure 3.5.2.11.



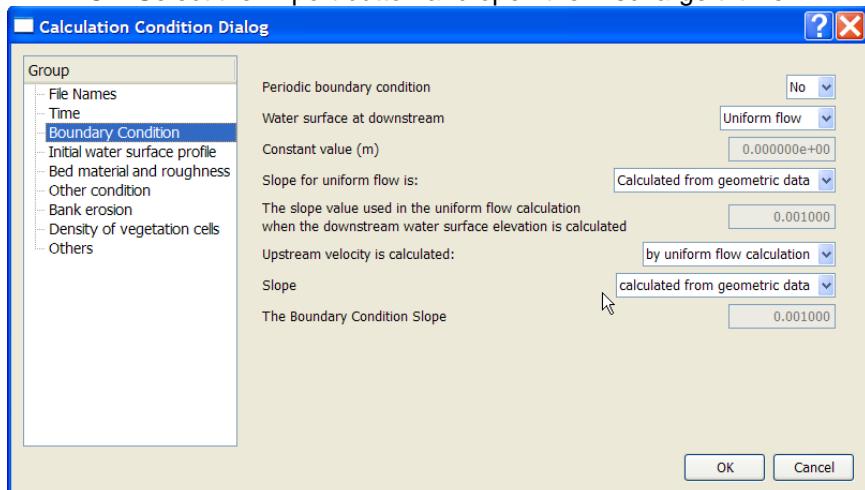
A – Select CGNS and RIC-Nays Output Type



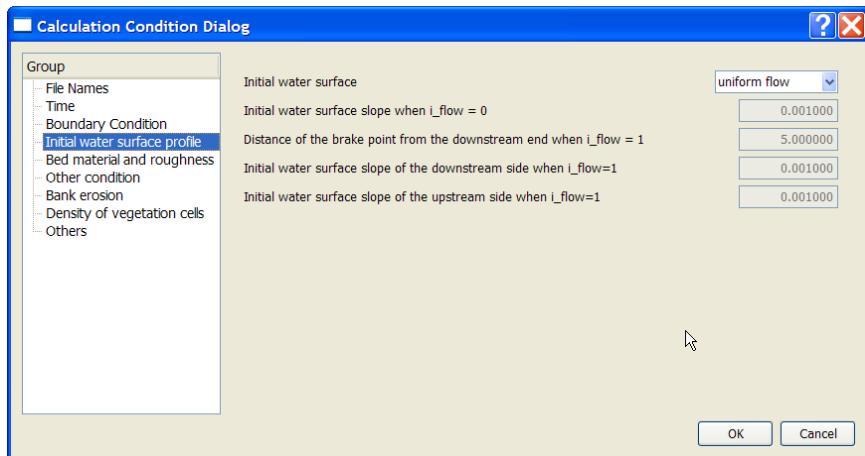
B – Select the Edit button for the Discharge Time Series to input Discharge Data – see figure(C)



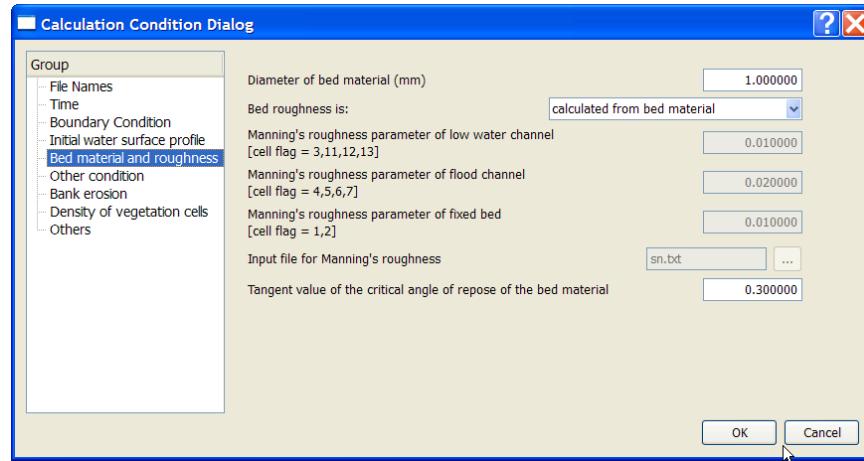
C – Select the Import button and open the Discharge.txt file



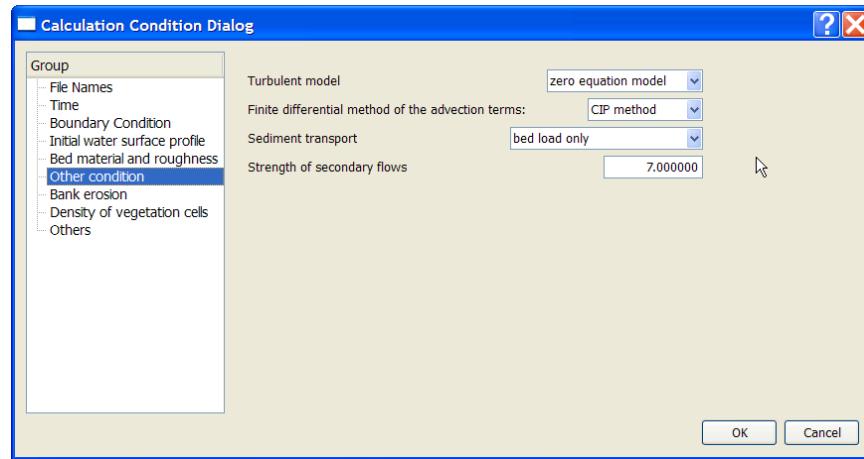
D



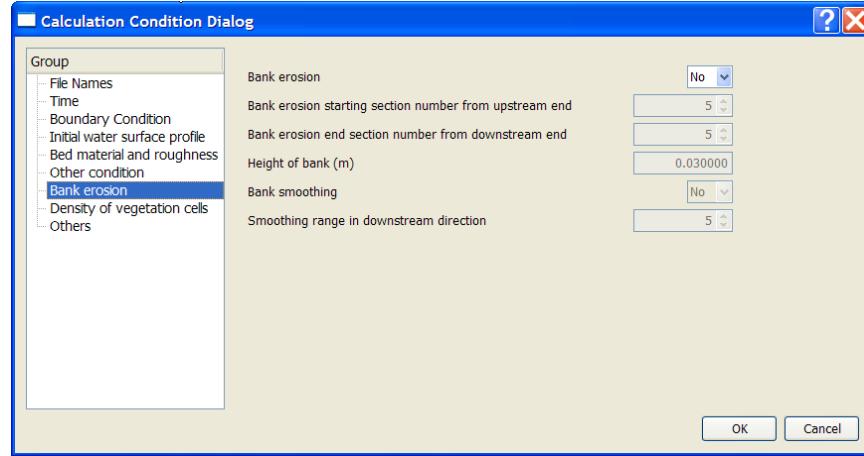
E



F



G



H

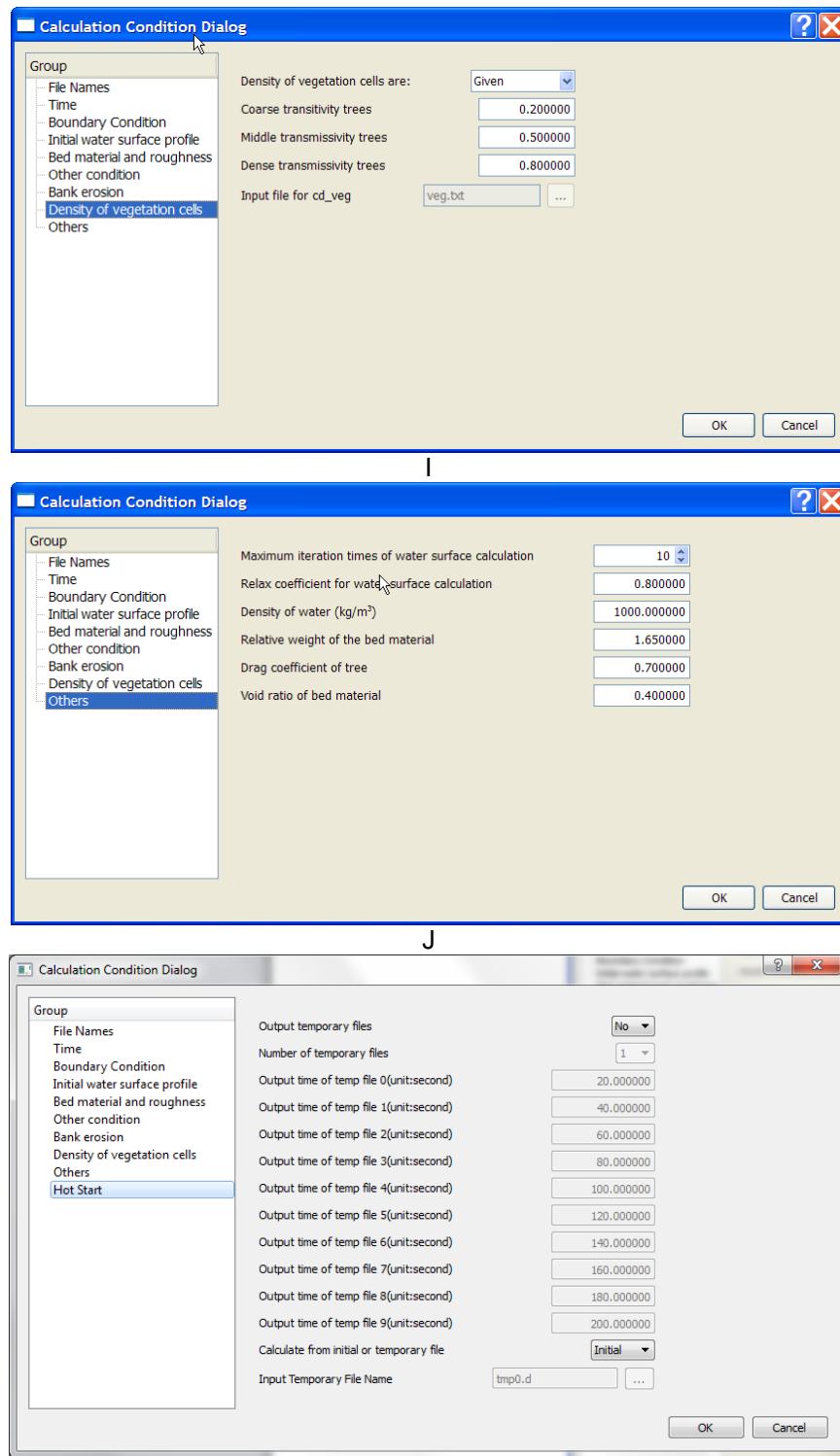


Figure 3.5.2.11 For each of the groups enter the parameters as shown. To enter the Discharge time series, in the Time Group (B), Select Edit for the Discharge Time Series value, and in the resulting dialog (C) select the Import Button and browse to the Discharge.txt file in the iRIC Tutorials\Nays\Tutorial 2 directory.

5. Run the Nays solver and visualize the results

From the menu select **iRIC->Run**. When the solver executes, a new Console window will appear As defined in the Time Group in the Edit Calculation Conditions dialog the Console window will refresh every 600 seconds of computation time so that the progress can be viewed.

When the computation is completed, the Console window will close and the Solutions branch in the Control Bar will display the available Scalar and Vector values available for viewing. **The 2D Solutions | 2D Sol. Scalar Sets** is set to Velocity and the **2D Solutions | 2D Sol. Vector Sets** is set to Velocity. To scroll through the saved solutions use the Time Step Tools toolbar and using the drop-down list select the solution to view, currently the time step is set to 30, the time step of the maximum discharge Figure 3.5.2.11. Save the project using **File->Save**, and give the project a name such as Tut2.

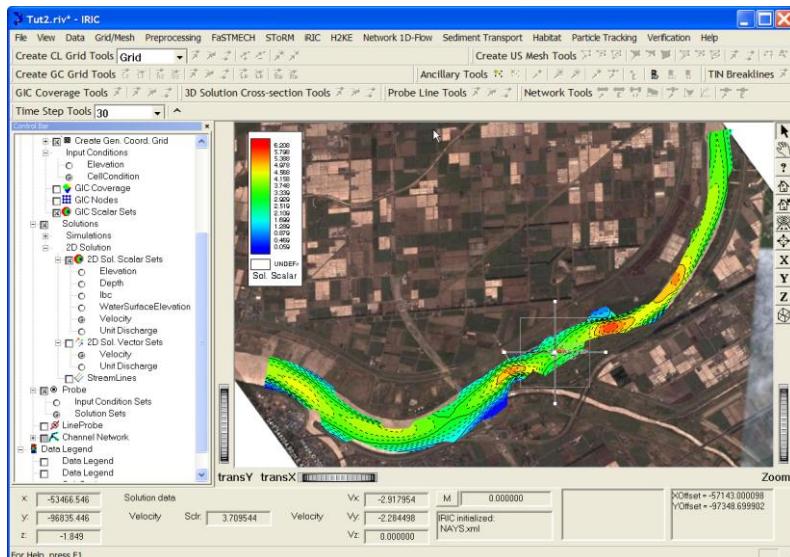


Figure 3.5.2.12 The solution at time step 30. Note that Elevation 2D Sol. Scalar Set is selected and the Velocity 2D Sol. Vector Sets is selected. Also A probe of the solution set has been turn on, and the results of the probe location are show at the bottom of the application.

Second Approach

1. Create a grid using iRIC's structured curvilinear orthogonal grid tools

Select **Grid/Mesh->2D Curvilinear Grid** from the menu. This will activate the general coordinate grid generating tools located in the Create GC Grid Tool toolbar (Figure 3.5.2.8).

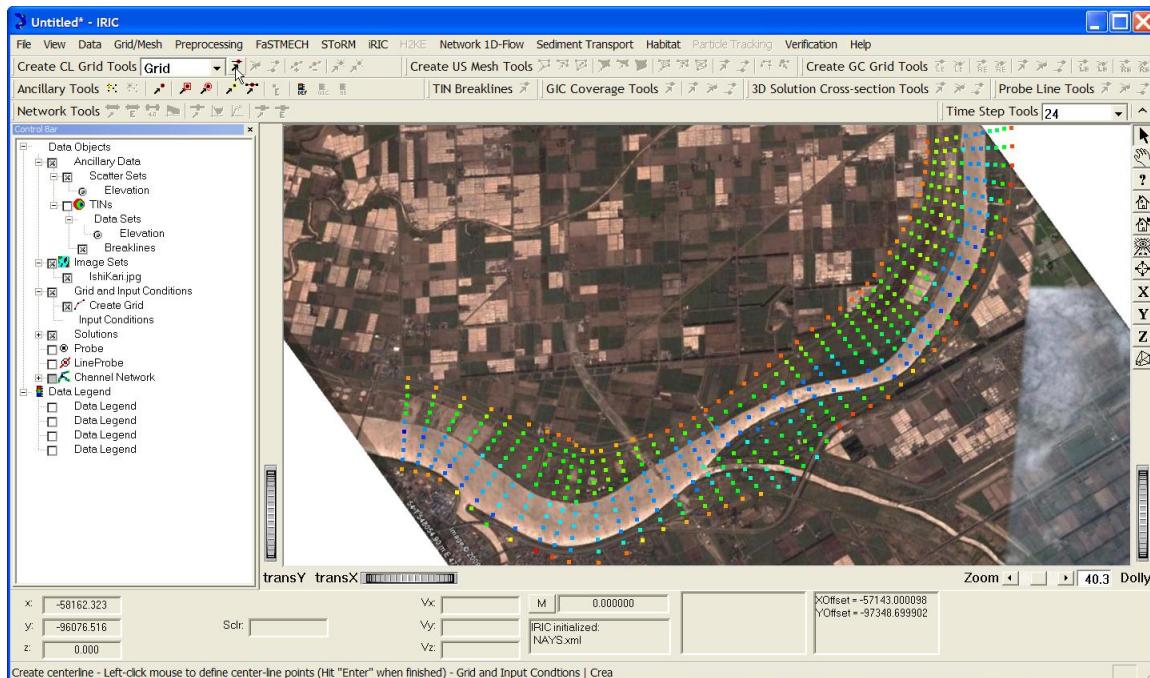


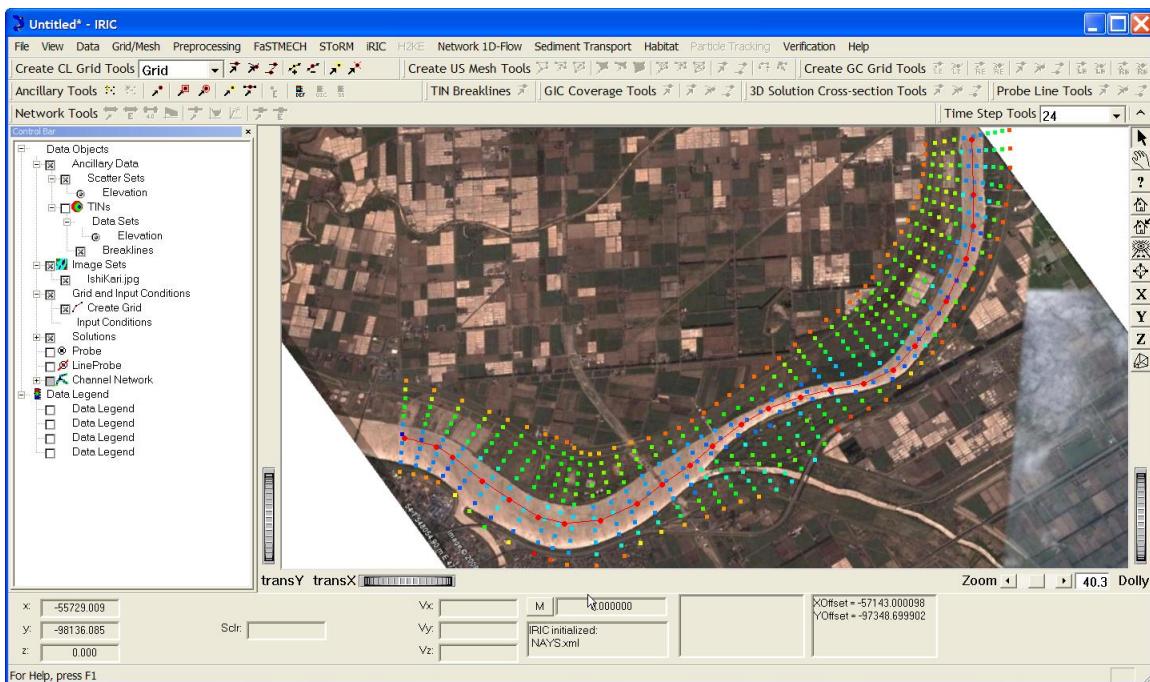
Figure 3.5.2.8 Notice that the **Grid and Input Condition** and the **Create Grid** branches in the Control bar are turned on and in the Create CL Grid Tools the first button has been activated and with the mouse cursor over the button the status bar message at the bottom of the application displays a short help message for the button. . The Create Centerline tool is used to interactively define the centerline of the curvilinear grid.

The following steps will illustrate how to create a general coordinate grid using the tools provided in iRIC. All the tools used will be from the Create GC Grid Tools toolbar, see Section 1.4.1 in the User's guide for a complete description of the tools in the toolbar.

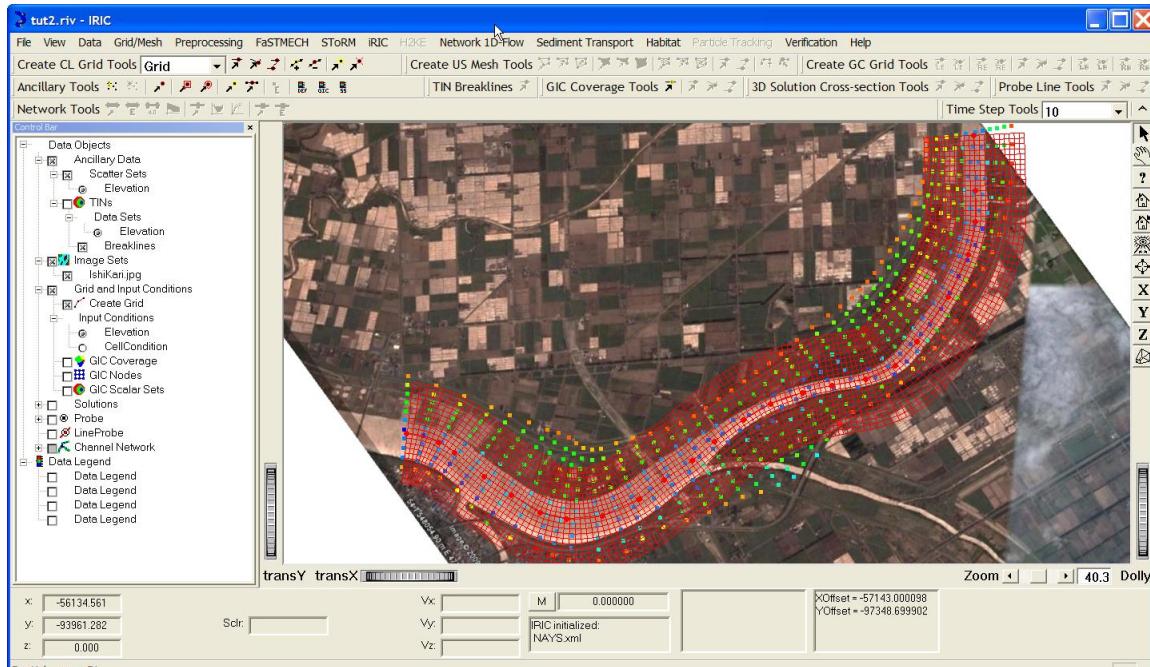
The following steps are required to create a grid. When the mouse is placed over any toolbar button, the status bar at the bottom of the iRIC interface contains information about that button. To build a structured curvilinear grid is relatively simple; first define a centerline following the channel from upstream to downstream and second build the grid by specifying the number of streamwise and cross-stream points and the width of the grid.

Centerline

- Turn on the **Grid and Input Conditions / Create Grid** object in the Control Bar.(i.e., Turn on the Grid and Input Condition and then Turn on Create Grid)
- Draw a centerline by selecting the Create Centerline tool (). The centerline must be drawn in the direction of flow. In this case, flow is from the upper right to lower left. Click the left mouse button to define centerline points and select Enter from the Keyboard when finished drawing (Figure 3.5.2.9 A).



A



B

Figure 3.5.2.9 A) Centerline: Note the spacing of the centerline points roughly one per channel width. B) Curvilinear orthogonal grid with a spacing of approximately 50 m in both the streamwise and cross-stream directions along the centerline

Build the grid

- Select **Preprocessing->Set 2D Curvilinear Grid Parameters** from the menu. In the dialog, enter the grid dimensions and the grid width. Try 161 for the number of points in the streamwise, 21 points in the cross-stream points, 1000 for the width and then press the Apply Button. The Apply button allows the changes to be seen without closing the dialog. The resulting grid will look something like that in Figure 3.5.2.9 B. Save the project and move on to step 3.

Repeat Steps 3 – 5 in the first approach