iRIC Software
Changing River Science

FaSTMECH Tutorial 3

FaSTMECH Tutorial 3
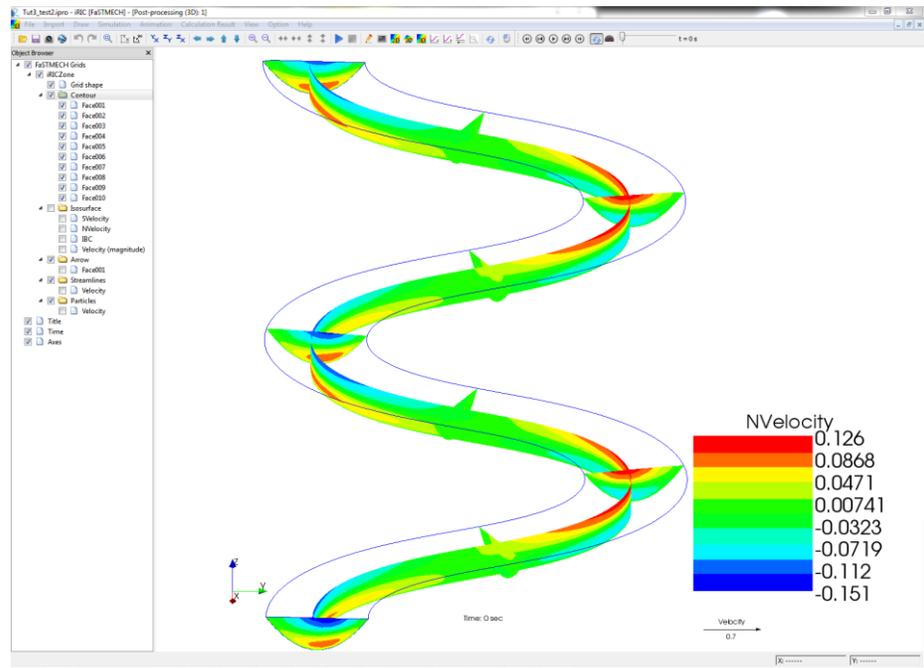
Contents

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 "Multifunction Channel Builder" that builds a model grid 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 and the FaSTMECH solver.

Tutorial 3 steps:
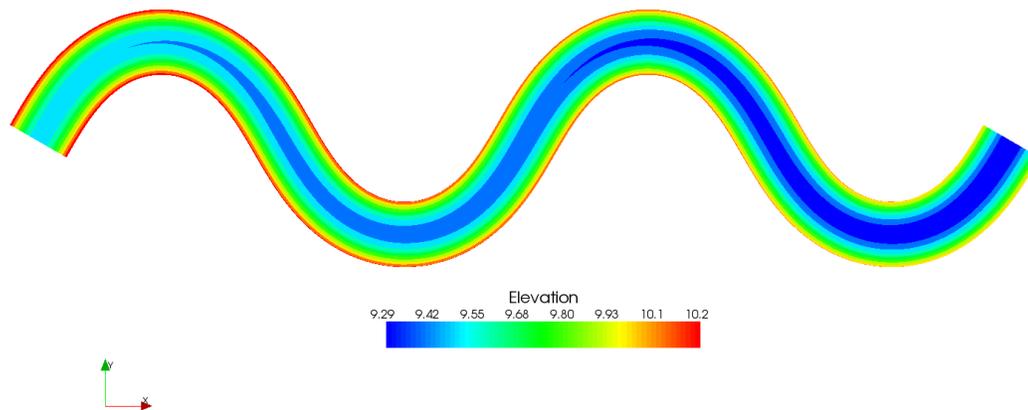
- Construct a meandering channel with the Multifunction Channel Builder
- Run a simulation and calibrate model with constant roughness.
- Visualize two- and three-dimensional solution excluding streamline curvature
- Rerun the simulation including streamline curvature
- Repeat steps above for a meandering channel with point bars
- Repeat steps above for a straight channel with alternate bars

FaSTMECH Tutorial 3

Part A - Construct a Meandering Channel with Multifunction Channel Builder

As in the other FaSTMECH tutorials, launch the iRIC application and in the iRIC Start Page, choose the FaSTMECH solver. In the iRIC Object Browser, right-click on *Grid [No Data] | Grid Creating Condition*, then select **Select Algorithm to Create Grid…** and in the resulting dialog select **Simple Grid Generator** in the list of Algorithms then choose the OK button. The Simple Channel Builder is a tool that constructs simple, idealized channels. Fill in the dialog as shown in Figure 1.
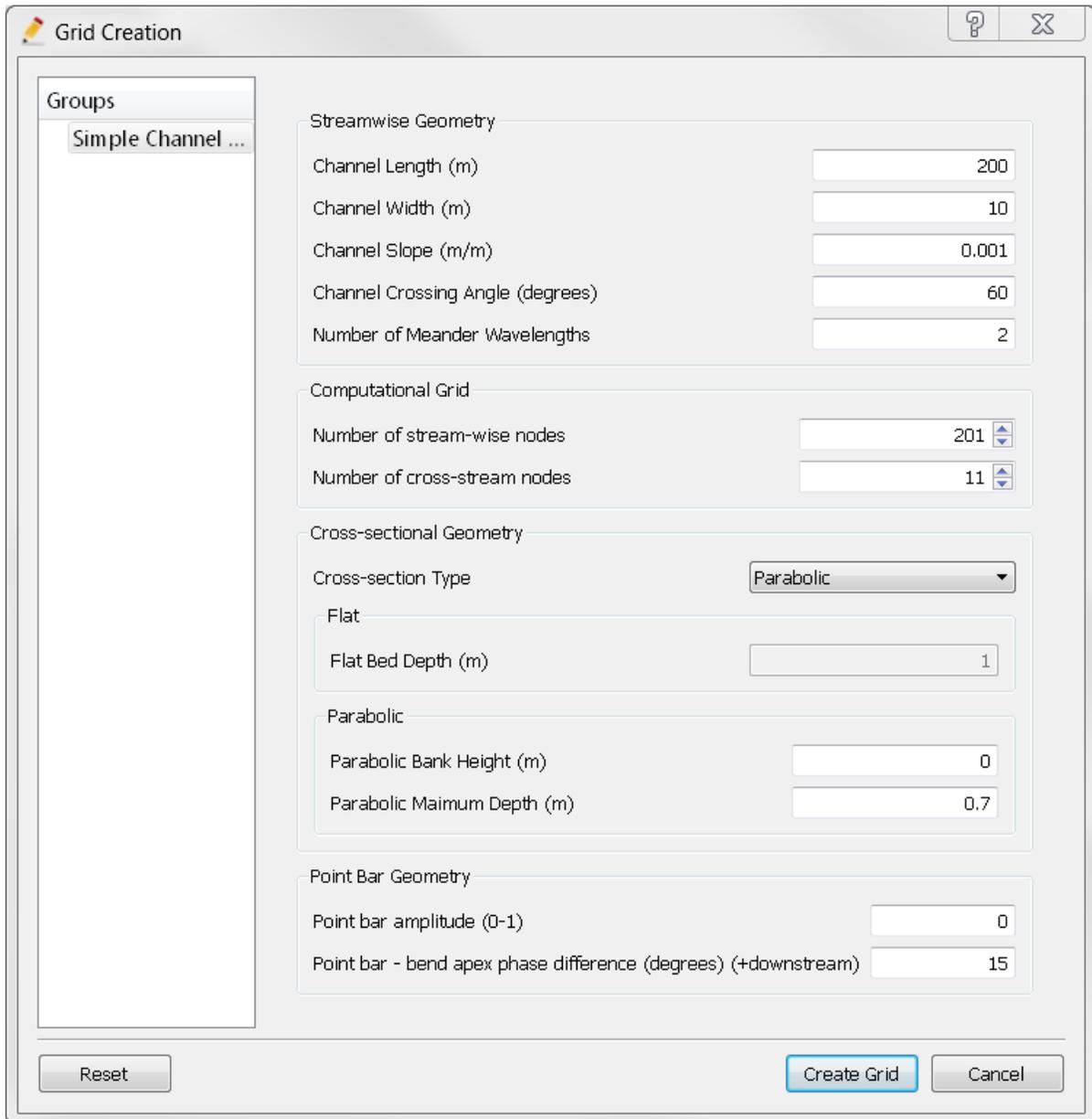
Figure 1. Multifunction Channel Builder dialog

To visualize the resulting topography (Figure 2) turn on *Grid [201 X 11 = 2211] | Node Attributes | Elevation* in the Object Browser. Then right-click on the same object and choose **Property** from the pop-up menu to get the Grid Node Attribute Display Setting dialog (Figure 2). The Contour setting has been changed to the Contour Figure option and the Transparency has been turned off by de-selecting the check-box. The legend is turned on by selecting **Geographic Data -> Setup Scalebar…** from the

3

menu and then selecting Elevation in the drop-down list box. Once you are comfortable with these tools and understand this simple channel form, select **File -> Save** and save as .ipro file under the tutorial 3 folder (renaming it however you like), then move on to the next step.
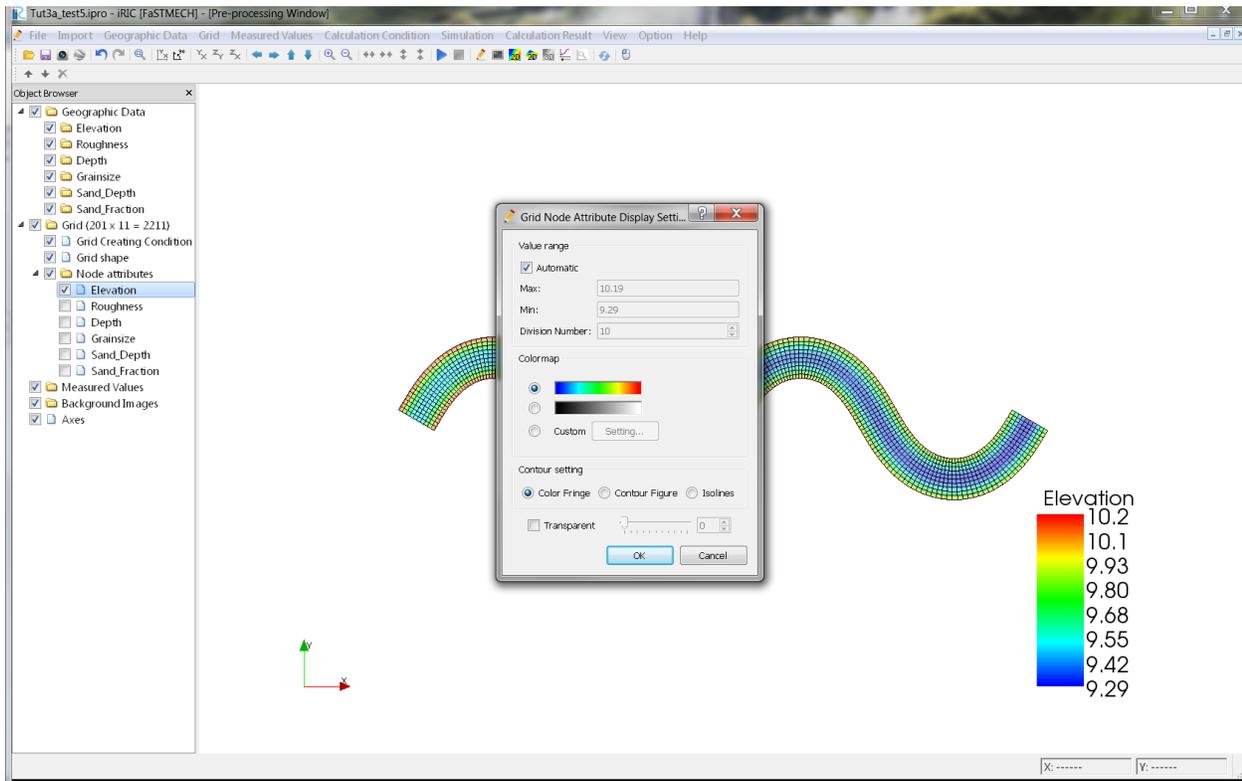

Figure 2. Generated grid and topography

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 making sure you are in the pre-processing window by selecting View->Pre-Processing from the menu. Then from the menu select **Calculation Condition ->Setting** and fill in the input dialogs as shown in Figure 3. Note that streamline curvature effects are turned off in this simulation.

FaSTMECH Tutorial 3

A

B

C

D

E

F

G

H

I

J

K

L

FaSTMECH Tutorial 3
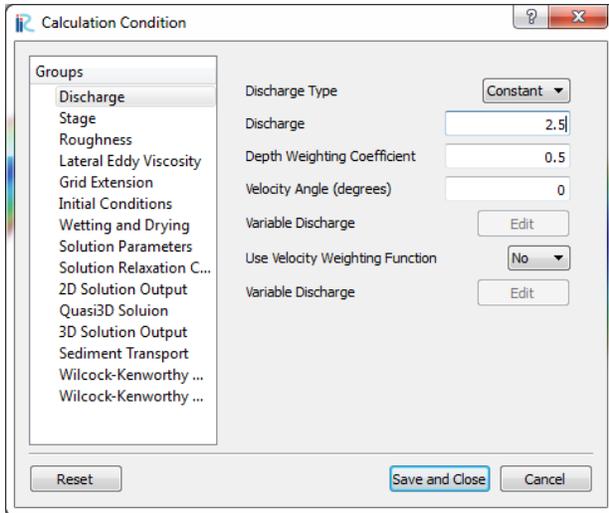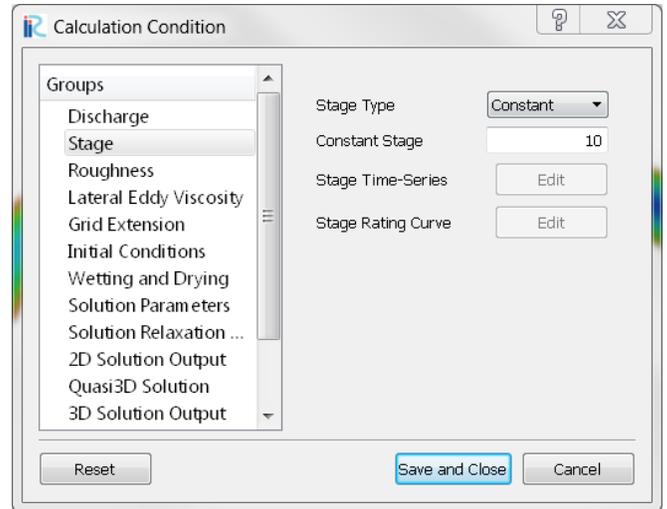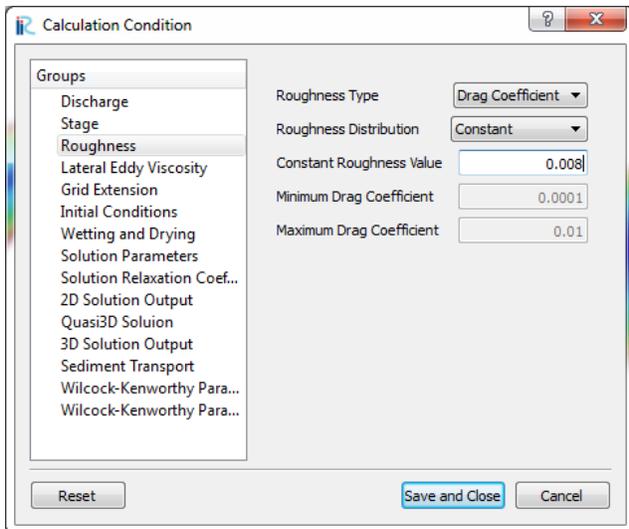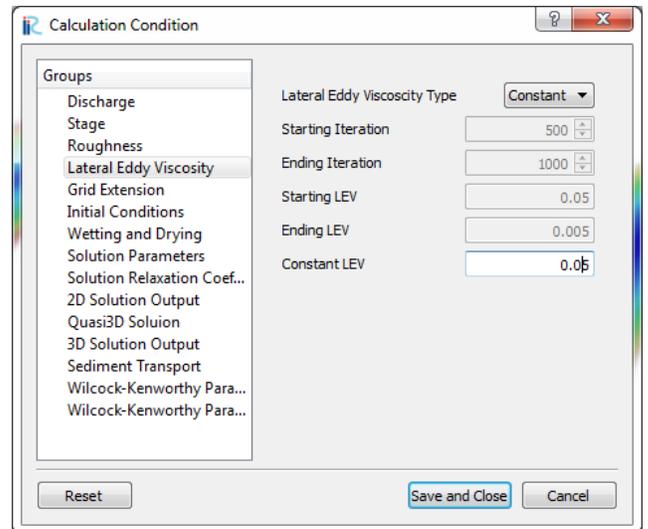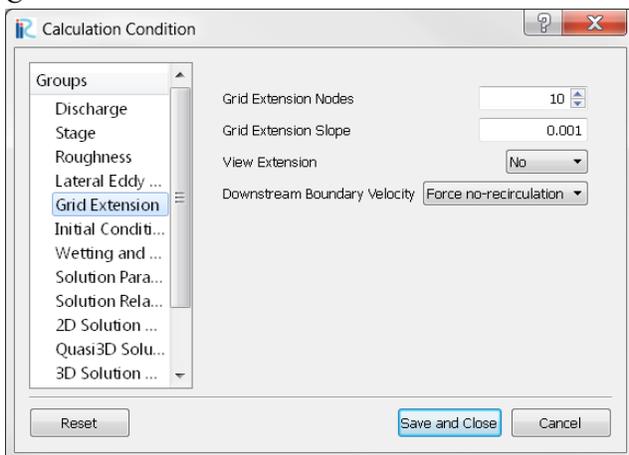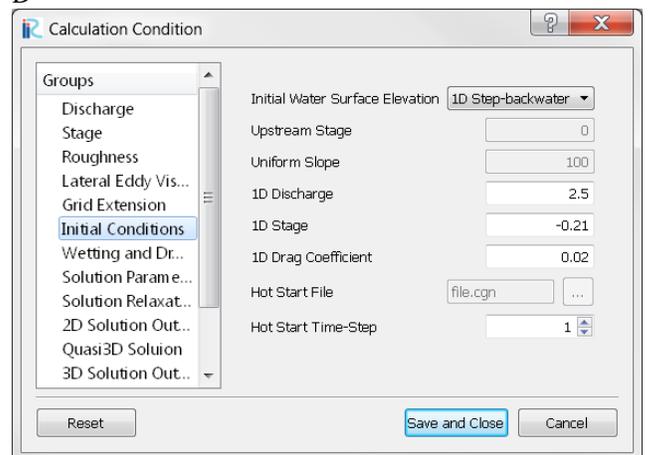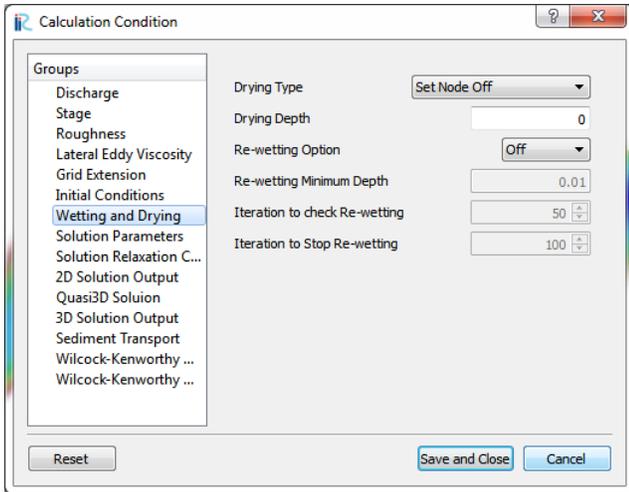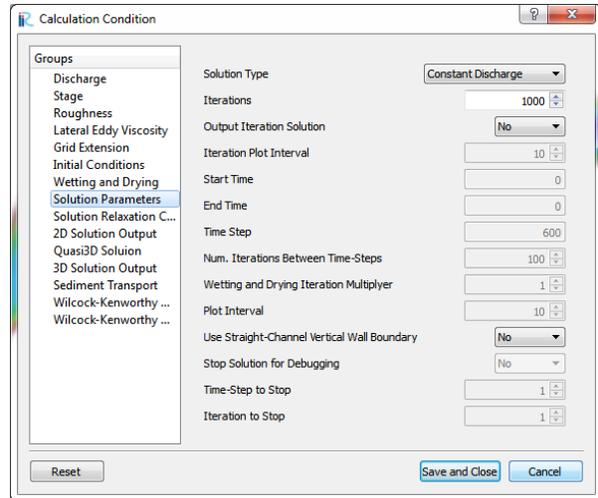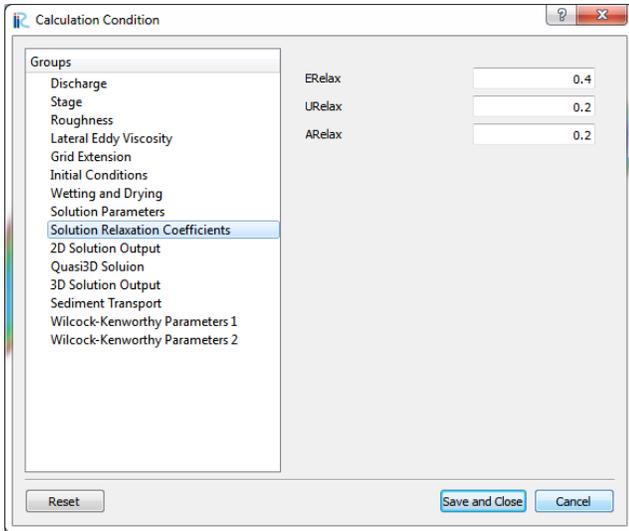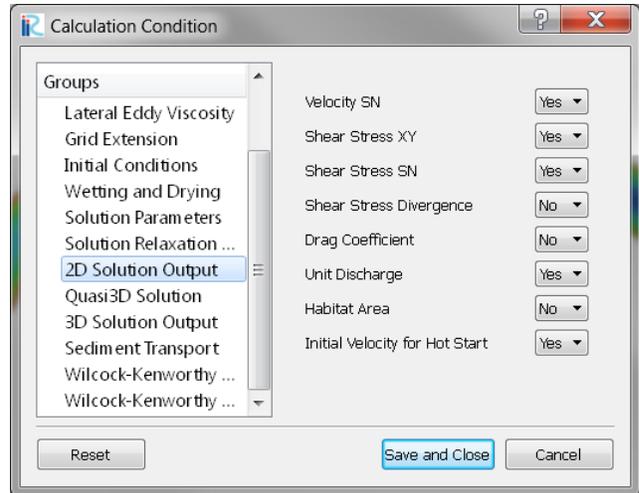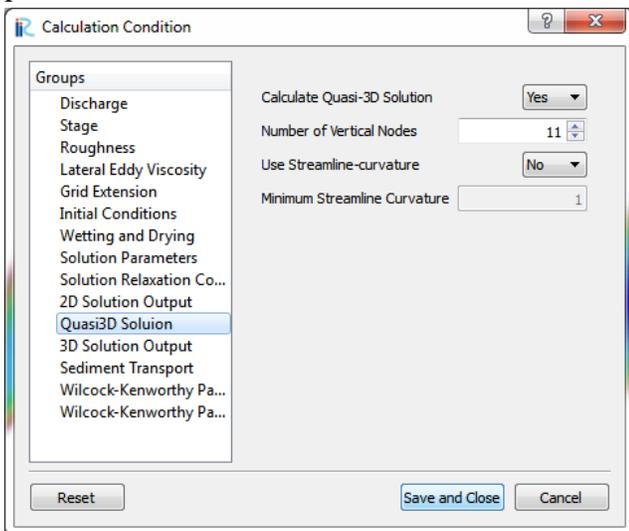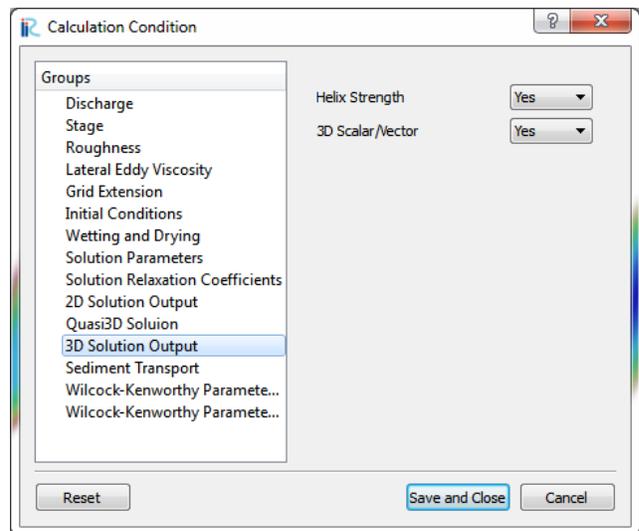
Figure 3. Enter the following parameters into the FaSTMECH Calculation Condition dialog. (A) Discharge, (B) Stage, (C) Roughness, (D) Lateral Eddy Viscosity, (E) Grid Extension, (F) Initial Conditions, (G) Wetting and Drying, (H) Solution Parameters, (I) Solution Relaxation Coefficients, (J) 2D Solution Output, (K)Quasi-3D Solution, (L) 3D Solution Output

Once you've filled in the Calculation Condition dialogs in accordance with figure 3.2.3b, select the Save and Close button. From the menu select **Simulation->Run (Ctrl + R** or ▶ from the toolbar) on the iRIC menu. After the model completes the simulation (should take only a few seconds), select the 2D viewer (2D or from the menu select **View->Post Processing 2D**). 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 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 however, view the value of helix strength and add a suitable legend to determine the maximum and 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 **Calculation Condition->Setting** (Select View->Pre Prodessing 2D if you are currently in the 2D Post Processor), then selecting *Roughness* in the dialog (Figure 3 (C)); change the drag coefficient here and under in the *Initial Conditions* dialog(Figure 3 (E)) and rerun the model simulation. 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-Dimensional Solution Excluding Streamline Curvature

Once you have a calibrated value of drag coefficient you are happy with, examine the solution in detail using both the 2D Post Processing and the 3D Post Processing windows. 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 vector fields as shown in Figure 4 as follows:

1. To create an image as shown in Figure 4:
   - In the 3D Post Processing window right-click *FaSTMECH Grids | iRICZone | Arrow* in the Object Browser and select the Property pop-up menu. Set up the Arrow Setting dialog as shown in Figure 4 (A and B). Where each Face added (Face001, Face002…, Face011) incrementally increases the value of the K index (from 1 – 11). Also, set up the Length, Sampling, and Color fields as shown.
   - By zooming into one of the meander bends you should be able to reproduce the graphics as shown in Figure 4 (C).The velocity from every vertical node is shown illustrating the secondary flow where the surface velocities are oriented towards the outside of the bend and the near bed velocities are oriented towards the inside of the bend.
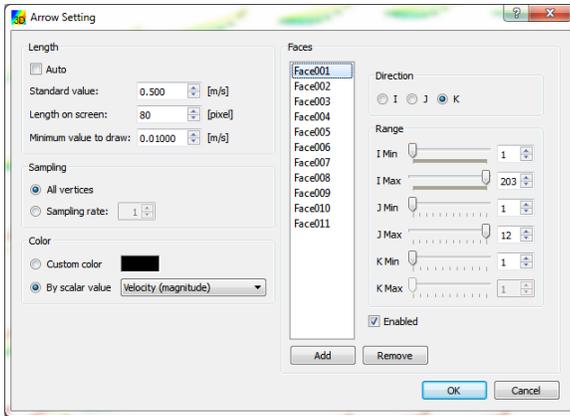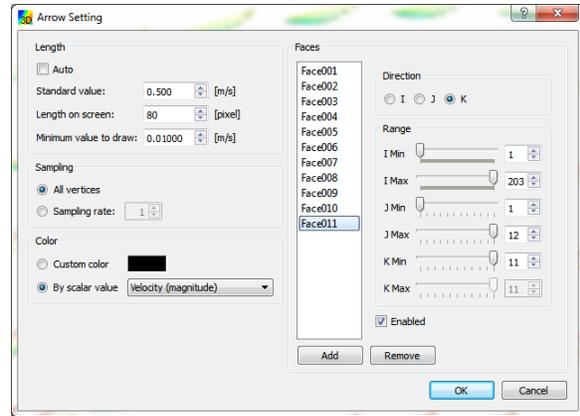
A   B
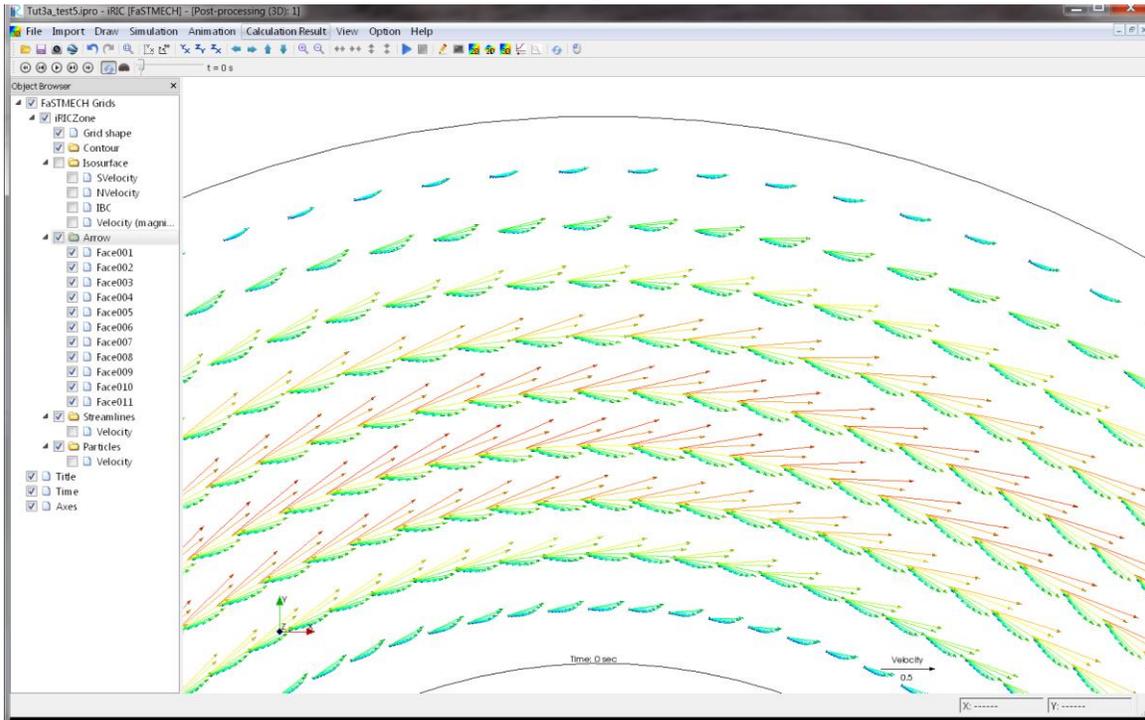
C

Figure 4. The Arrow Setting dialog. Note that in (A) the K index is set to 1 and in (B) the K index is set to 11. The 3-

dimensional vector field where the velocities at the surface are highest and oriented towards the outside of the bend

and the near bed velocities are lowest and oriented towards the inside of the bend.

2. To Create an image as shown in Figure 5:
   - In the 3D Post Processing window right-click ***FaSTMECH Grids | iRICZone | Contour*** in the Object Browser and select the Property pop-up menu. Set up the Contour Setting dialog as shown in Figure 5 (A). Nine Faces are added incrementally with the value of the I index equal to 1,25, 50, 75, 100, 125, 150, 175, 200. One of the faces also goes longitudinally along the J = 6 index as shown in Figure 5 (B).
   - To add some vertical exaggeration so that it's easier to see the cross-sections select View->Z Direction Scale from the Menu and enter 5 in the Z-direction scale. The easiest way to orient the channel is to first select the ⤢Y .button

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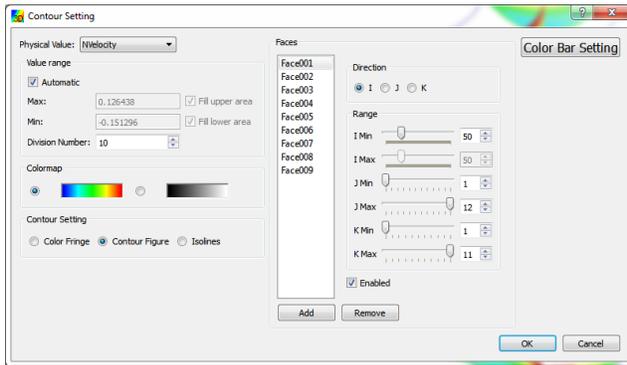
A
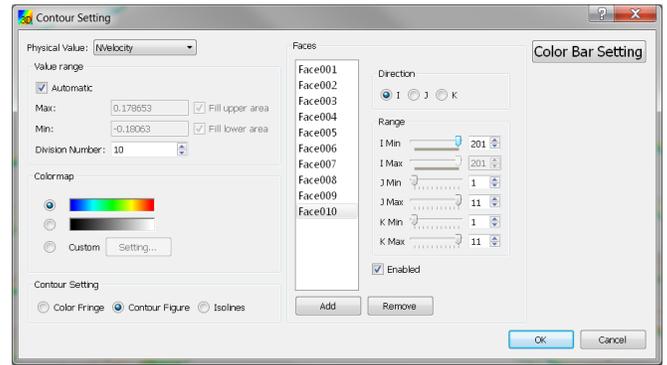
B
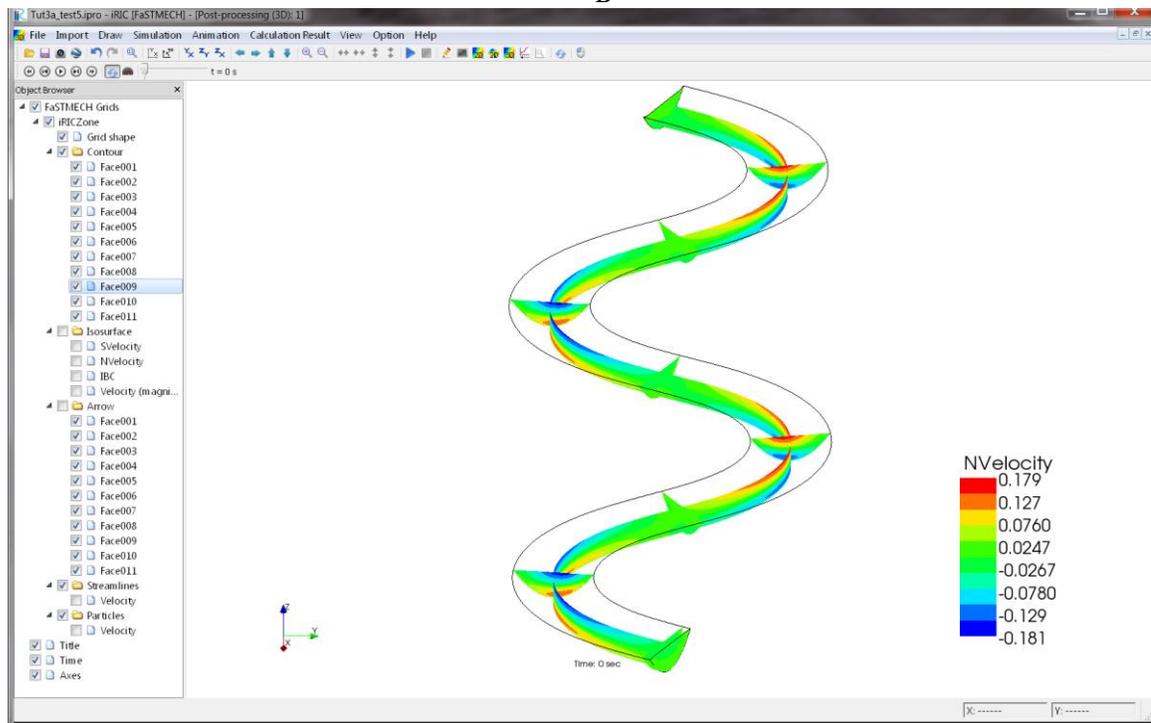
C

Figure 5.   The cross-stream velocity is shown for a number of cross-sections through the reach. NVelocity is the magnitude of the velocity oriented perpendicular to the stream-wise direction. When the grid follows the channel curvature well as is the case here, NVelocity provides a good representation of the secondary flow.

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?

9

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 the Calculation Conditions edit the ***Quasi3D Solution*** dialog to set streamline curvature on and set the minimum radius of curvature to 2 meters. Run the simulation again and visualize the solution results. Create a plot of the helix strength as you did in Part C and Compare the two 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 Bar Topography of channel with point bars.
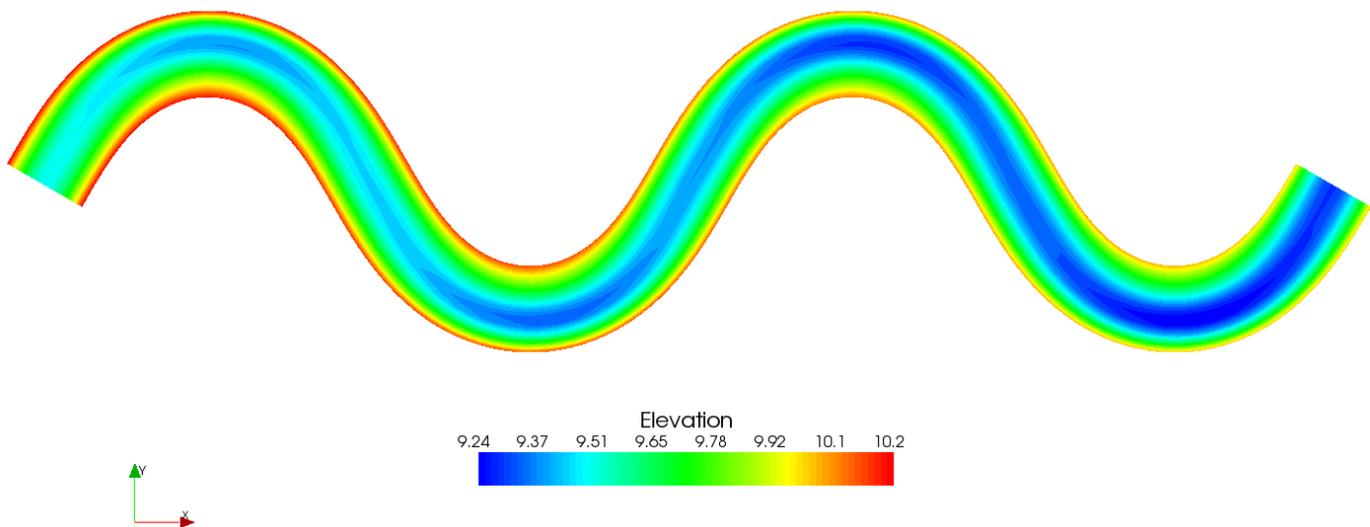
Figure 6. Simple Channel with point bar.

FaSTMECH Tutorial 3

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 (Figure 6), and the effect of that addition will be explored.

Save your existing model run, then select **File -> New Project**, then select FaSTMECH in the Select Solver dialog. As in Part A, to create topography and a grid with point bars right-click on *Grid [No Data] | Grid Creating Condition* in the Object Browser, then select **Select Algorithm to Create Grid…** and in the resulting dialog select **Simple Grid Generator** in the list of Algorithms then choose the OK button. Fill in the dialog as shown in Figure 1 except the Point Bar Amplitude is set to 0.5.

This will add a simple low-amplitude point bar to the curved channel used above (Figure 6). As above, look over the new topography and then prepare an input file using the values given in Figure 3 with the exception of:

- Drag coefficient, which is set to the value determined in Part B above

Run the model and look at the solutions. Is the reach-scale flow uniform? If not, adjust the drag coefficient as in Part 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 Figure 7 A and B). 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?

A   B
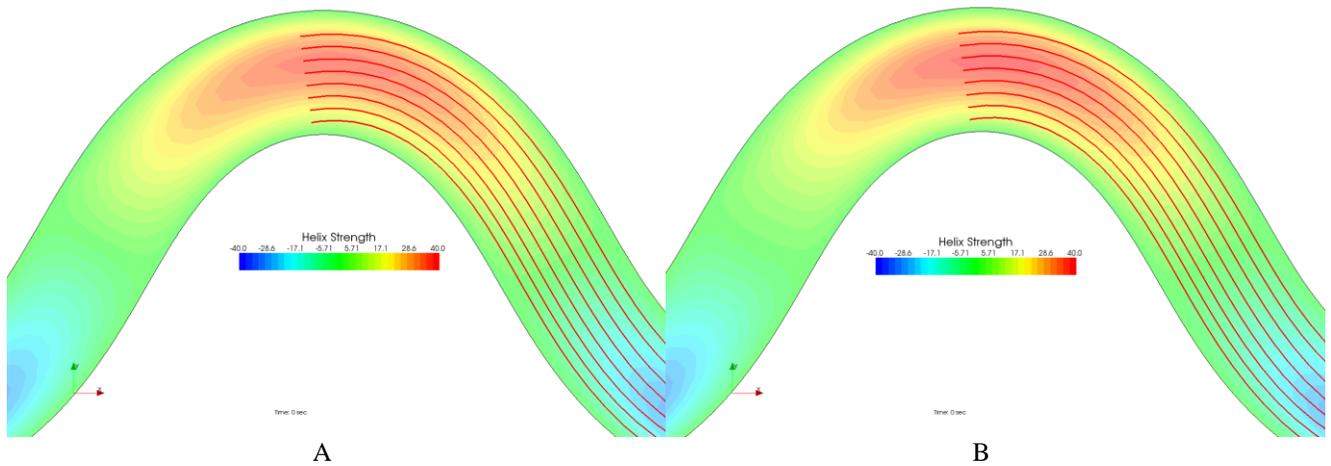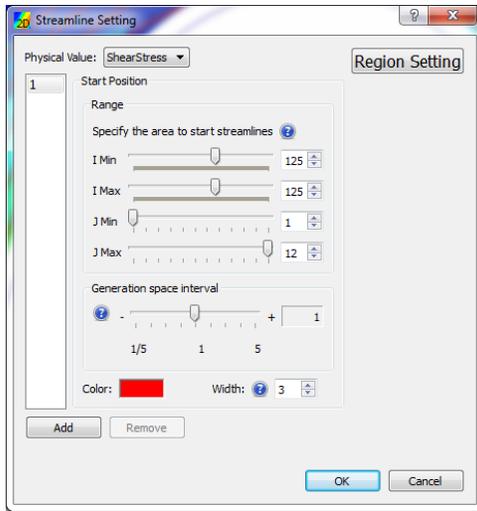
C
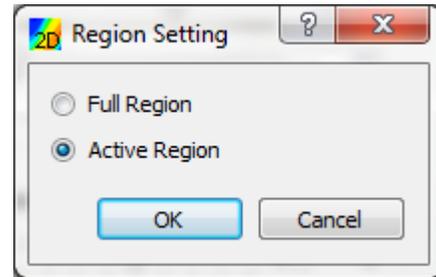
D

Figure 7. Plots of Helix Strength and Streamlines with (A) no streamline curvature and (B) with streamline curvature. To generate (A) and (B), set the Streamline Settings to those in (C) and (D).

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

Elevation

9.24   9.37   9.51   9.65   9.78   9.92   10.1   10.2
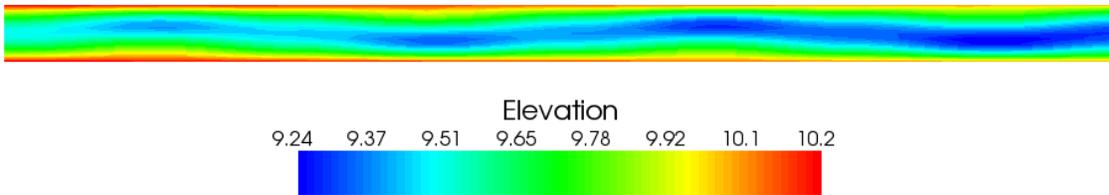
Figure 8. Straight channel with Alternate Bars.

Save your existing model run, then select **File -> New Project**, then select FaSTMECH in the Select Solver dialog. As in Part A, to create topography and a grid with point bars right-click on *Grid [No Data] | Grid Creating Condition* in the Object Browser, then choose **Select Algorithm to Create Grid…** and in the resulting dialog select **Simple Grid Generator** in the list of Algorithms then choose the OK button. Fill in the dialog as shown in Figure 1 with except for the *Crossing Angle*, which is set to 0, and *Point bar amplitude* which is set to 0.5. This will generate a straight channel with simple low-amplitude alternate bars (Figure 8). As above, look over the new topography and then prepare an input file using the values given in Figure 3 with the exception of:

- Drag coefficient, which is set to the value, determined in Part B above.

Run the model and look at the solutions. Is the reach-scale flow uniform? If not, adjust the drag coefficient as in Part 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 9. Try to answer the following questions.

1. How does the amplitude of the secondary flow compare to that in the meandering channels above?

12

FaSTMECH Tutorial 3

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?


A

B
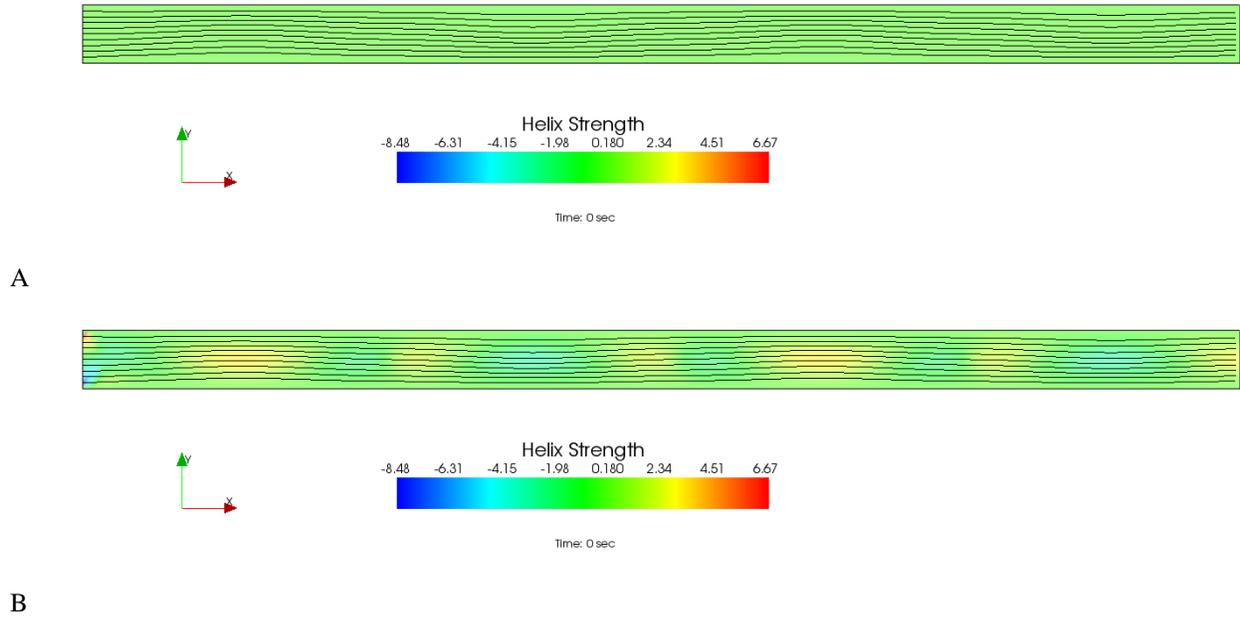
Figure 9. Helix Strength with secondary flow set (A) without streamline curvature and (B) with streamline curvature Streamlines are shown in red.

FaSTMECH Tutorial 3