iRIC Software

Changing River Science

FaSTMECH Tutorial 9

Flood reconstruction and peak discharge estimation

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

Introduction

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

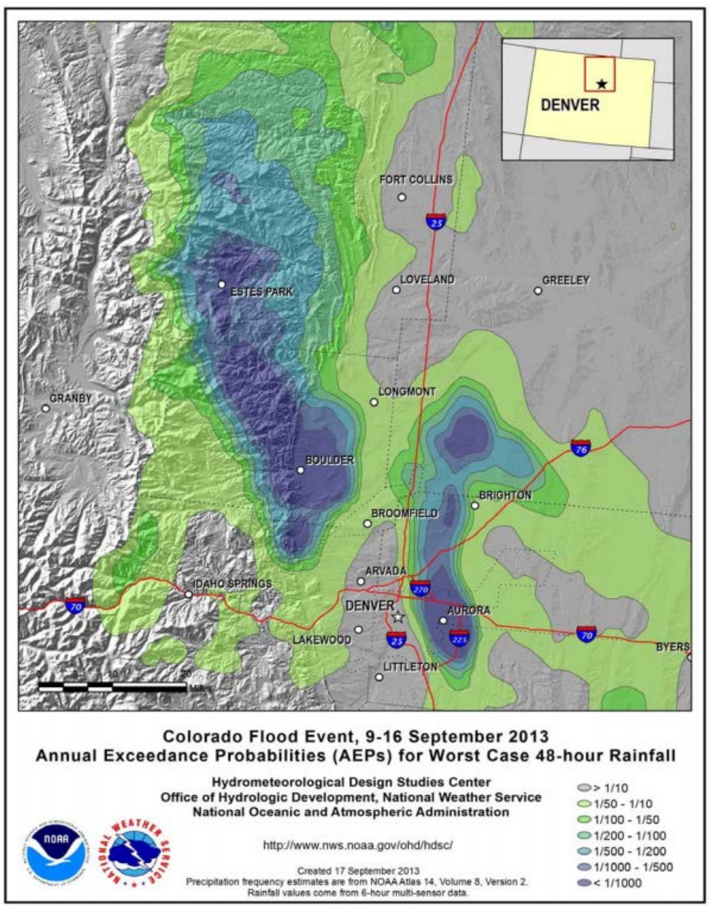


Figure 1

---

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

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

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

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

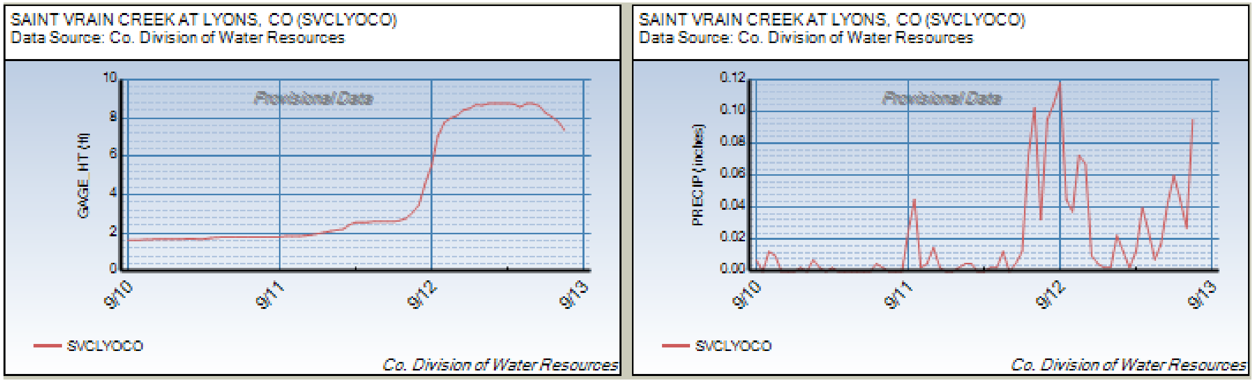


Figure 2. Colorado Division of Water Resources gage record for St. Vrain Creek at Lyons, CO (http://www.dwr.state.co.us/SurfaceWater/data/detail_graph.aspx?ID=SVCLYOCO&MTYPE=DISCHRG)

Following the flood, USGS personnel surveyed High Water Marks (HWMs) in the region of interest, and shortly following the flood LiDAR was flown. We are provided with topography as measured by airborne LiDAR following the flood and the measured HWMs. Our task is to determine the peak discharge during the flood. In this tutorial we will perform the following tasks:

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

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

Menu Bar operation: *File* → *Import* → *Geographic Data* → *Elevation…* The Menu Bar operation is shown in Italic font, and each successive option is separated by an arrow (→).

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

Task 1: Import Data

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

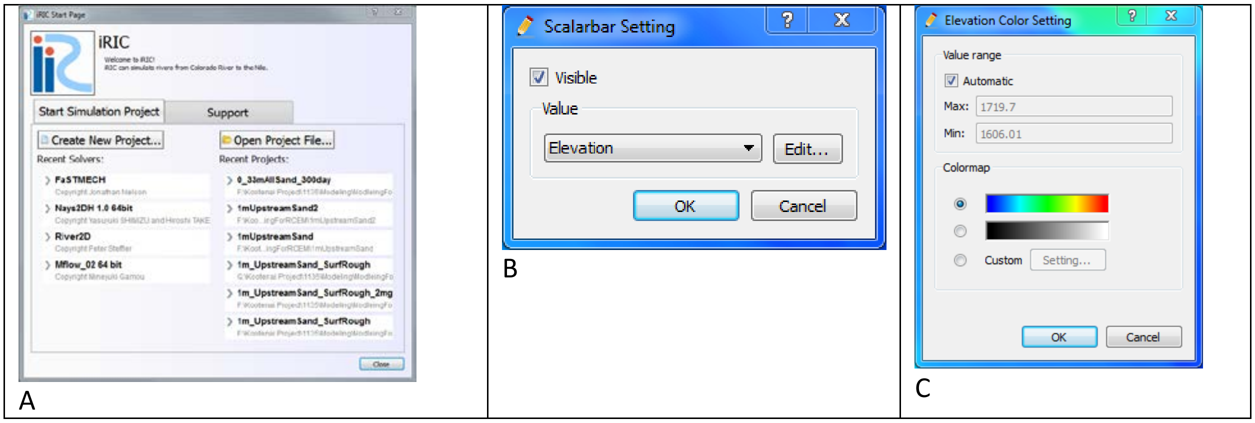


Figure 3. A) Opening dialog to set solver or open previously built project. B) Scalarbar Setting dialog. C) Elevation Color Setting dialog.

2. **<u>Import topography:</u>** Select from the **Menu Bar**, *Import* → *Geographic Data* → *Elevation...* In the folder containing the data for this tutorial select the file *StVrain_1_0m_477700_4452000_Shifted.tpo*. This data is a subset of available LiDAR data for the reach (Figure 4A). It has been decimated to 1-meter resolution, and shifted by 477700 meters in the X-direction and 4452000 meters in the Y-direction.
3. **<u>Add a scalebar:</u>** Select *Geographic Data* → *Set Up Scalarbar…* from the **Menu Bar** (Figure 3B). In the resulting dialog make sure the dropdown list is set to *Elevation* and select OK. You can always change the color setting by selecting *Geographic Data* → *Color Setting…* → *Elevation...* For now keep the default setting by selecting OK (Figure 3C). Your project should now look like Figure 4A.
4. **<u>Import background image:</u>** From the **Menu Bar** select *Import* → *Background Image***…** and select *PostFloodImage_Shifted.jpg*.
5. **<u>Import the measured HWMs:</u>** Select *Import* → *Measured Values***...**, and select the file *HWMarks3_SAC_RIGHTBANK.csv*. By default, iRIC has placed a data legend for the HWMs over the *Elevation* data legend. Move the *Elevation* data legend by first selecting *Geographic Data* | *Elevation* in the **Object Browser**. Next, click your mouse on the legend and drag it to a new location.
6. **<u>Increase point size of HWMs:</u>** Turn off the *Elevation* point set by de-selecting the *Geographic Data* | *Elevation* checkbox in the **Object Browser**. To make the HWMs more visible, first select *Measured Values* | *File path and name* | *Scalar* with your mouse. Then right-click on the same and select *Property…* from the pop-up menu. In the resulting dialog set the Point Size to 7 and select OK. Your project should now look similar to Figure 4B.

We have now imported all the data necessary to begin creating a flow model and in the process, have learned some of the basic functionality of the iRIC application. We can import different data sets, add a data legend for the different data sets, and change the graphic properties and view of imported data.

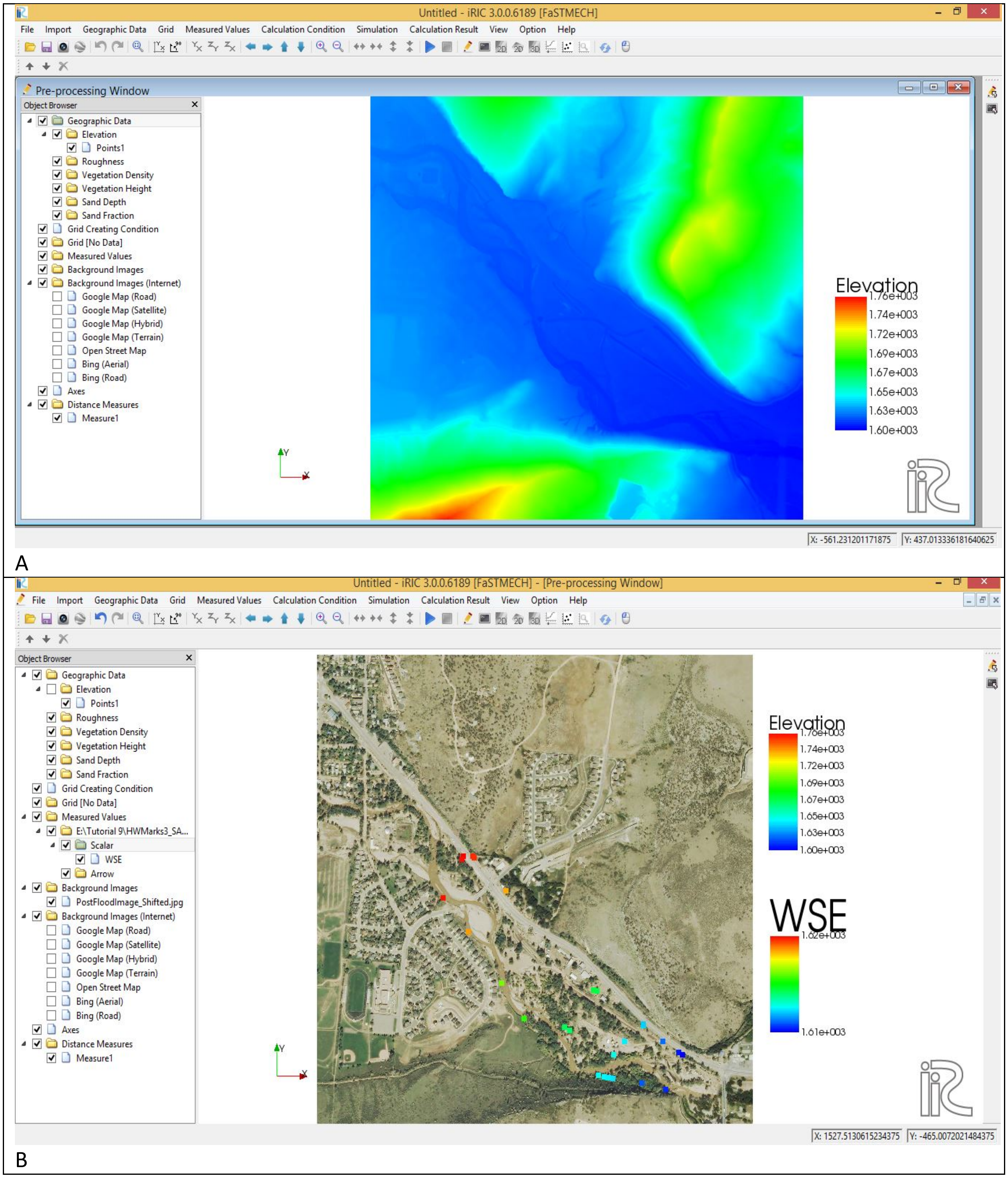


Figure 4. A) graphic view of imported LiDAR data set and B) Background image and measured HWMs.

Task 2: Preprocess the topography

1. **<u>Set the color map of the topography to better visualize the elevations of interest</u>**
    a. Turn off the *Measured Values* by deselecting it in the **Object Browser** and then turn the first *Elevation* back on. We will edit the elevation data to simplify the view. First, we want to delete those points higher in elevation than are necessary to model this reach and second, we will change the color setting.
    b. Select *Geographic Data | Elevation | Points1* in the **Object Browser**. Doing so enables the tool bar  for editing the point set data. Notice that if we select some other branch in the **Object Browser**, the toolbar is no longer visible. In iRIC, whenever you want to operate on a specific data set, you should first select that data set in the **Object Browser**, and any available tools for interacting with that data will be visible in the toolbar near the top of the application. The Select Point button (4th from left) allows you to select a single point, shift-select multiple points, or drag-select a rectangle of points. To see how this works, drag-select a rectangle of points. The point set is very dense, so unless you zoom into a specific area, it will be hard to see the points that are selected. That is not important for now. Following the selection, right-click on the graphic window and select "Edit Selected Points" from the pop-up menu. If we did not effectively select a point then the "Edit Selected Points" option will be greyed out. If so, try again. In the Edit Points dialog notice that we will see the number of points selected and the minimum and maximum value of those points. This provides a quick and easy way to probe the data and we will use this technique again later in the tutorial. Also notice that X-location and Y-location are blank. If you had selected a single point then you could change its x,y position if desired. When multiple points are selected only the "Value" is available to change. In this case we are not interested in editing the data; we only use this tool as a quick probe of the values, specifically the minimum and maximum values. Select the Cancel button. If you select the OK button then all the selected points will be set the "Value" specified in the Edit Points dialog. Don't do that here; again, we are just probing the points to determine their values. If you do this by accident then simply select the Undo button () on the **Main Toolbar** at the top of the application.
    c. We can also delete points greater than a defined value. This will greatly reduce the number of points in the data set to speed up some of the operations we will perform later and will provide more color resolution to the data of interest in the channel. Using the Select Point tool described above, select a point or region of points in the light blue range to determine the elevation value(s) specifically. You should find values near 1634. Zoom out so the entire elevation data set is visible, then using the Select Point tool, drag-select the entire data set. Right-click on the graphics window and select "Delete Selected Points Greater Than Value", and in the resulting dialog enter 1634. Select OK. The results should look similar to Figure 5A.
    d. To change the color map of the data first make sure you have selected *Geographic Data | Elevation | Points1*, since that is the data set you will be operating on. Next select *Geographic Data* → *Color Setting* → *Elevation…* from the **Menu Bar**. In the Elevation Color Setting dialog, de-select the "Automatic" checkbox and then enter 1634 for the Max value. Choose the "Custom" radio button and then select the "Setting…" button and set the Type to "Three Colors" using the drop-down menu, the Medium Value color to white (click the color field to get the *Select Color* dialog) and then enter 1620 for the *Value*. The resulting data and the three dialogs described in this operation are shown in Figure 5B. Select OK and OK again when done.

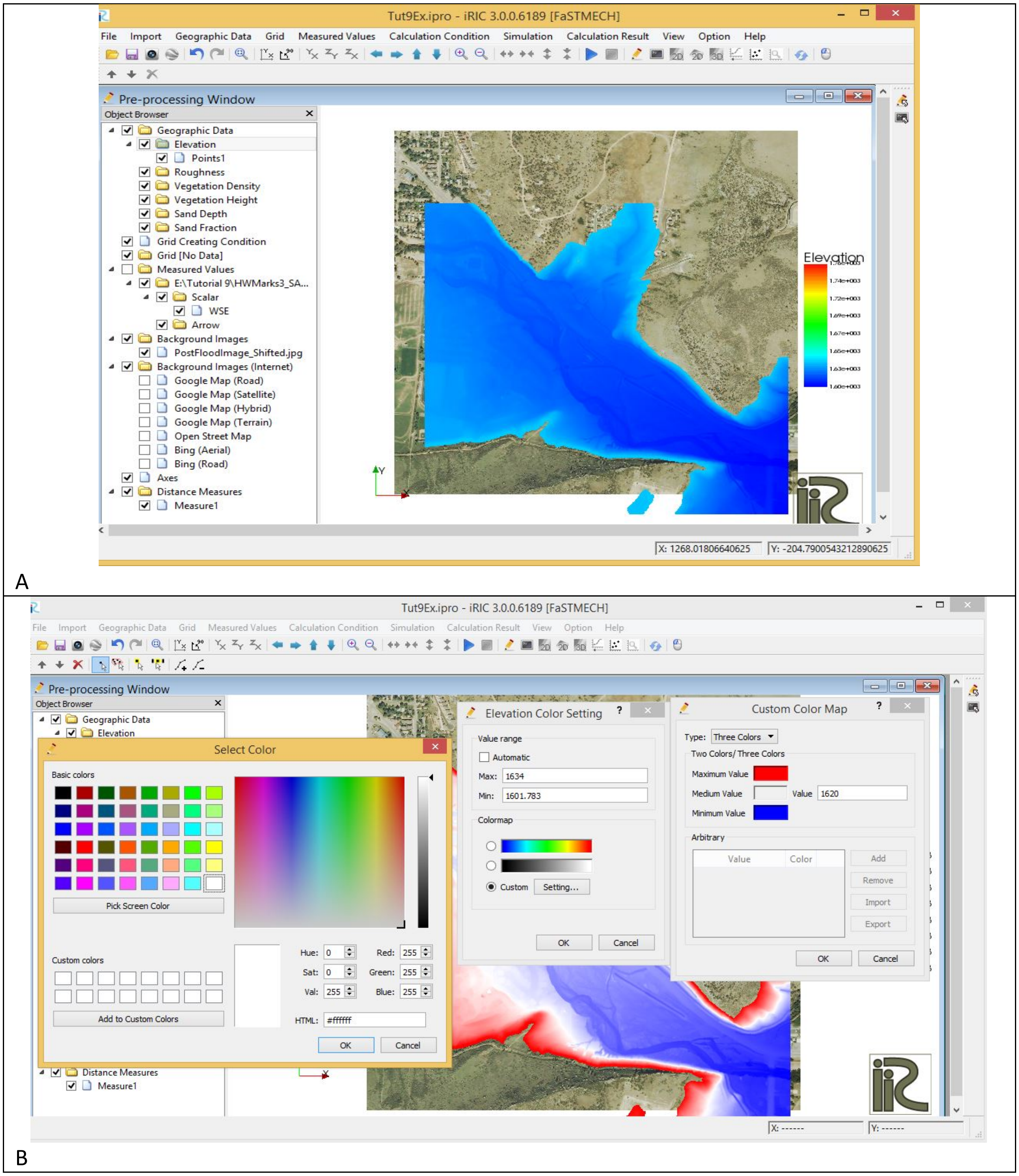


Figure 5. A) Elevation data below 1634 meters. B) Modified Color map of elevation data. Insets are in clockwise direction 1) Elevation Color Setting dialog, 2) Custom Color Map dialog, and 3) Select Color dialog.

2. <u>**Modify the topography to account for flow under the roads.**</u> In iRIC we can modify any of the imported Geographic Data such as elevation by drawing polygons over the point set and defining an elevation value to represent the topography. Then, when the Geographic Data is mapped to the grid we will create later,

wherever a polygon exists it will override the measured values below it. Here, we will add a series of polygons to approximate the location of the channel under the roads.

a. In Figure 6, four locations have been identified that require some modification to the elevation data set. At 1, 2, and 3, a road crosses over the channel, and at 4, there are constrictions in an irrigation ditch likely due to vegetation obscuring the ditch in the LiDAR topography. In the following steps, we will add polygons on top of the elevation data. Each polygon has a user-defined value that will override the underlying topography and approximate the elevation of channel bottom underneath the road. In this way, modifications to the topography can be imposed without losing the original measured data. If you don't like the modification, just delete the polygon.
b. To modify the height of the topography at a road crossing, follow the steps below. You will add a polygon yourself for the road crossing identified by the number 1 in Figure 6, and for the rest of the road crossings and modifications to the irrigation canal you will import the polygons which have been built and saved in a shapefile (.shp) for your use.
    i. Adjust the color setting to add more detail to the topography so it's easier to define the location of the road crossing #1. We learned how to adjust the color setting of legend in section 1d above. Set the color setting to a minimum of 1615 and maximum of 1617.
    ii. Set the size of the elevation point set to 7 which will make the data easier to see when zoomed in. To do this select *Geographic Data | Elevation | Points1* and the right-click and select "Property" in the pop-up menu. Then set the point size to 7 and select the OK.
    iii. Probe the elevation values on either side of the road to determine the best single value to assign to the elevation of the polygon you will create next. We learned how to probe values previously in Section 1b.
    iv. To create a polygon that approximates the spatial location of the channel below the road be sure to first select the *Geographic Data | Elevation* in the **Object Browser** with a left-click, and then right-click the same and select *Add* → *Polygon* in the pop-up menu. Draw a polygon that crosses the road and connects to the topographically low channel on either side. When finished press Enter on the keyboard to finish drawing the polygon and in the resulting Edit Elevation Value dialog, enter the elevation for the polygon based on the value you probed in section iii above (Figure 7A). Select OK to create the polygon.
    v. Import polygons created for you to improve the elevation of the remaining two diversions and the irrigation canal. In the **Menu Bar** select *Import* → *Geographic Data* → *Elevation* and browse to the *IrrigationCanal2.shp* file. Make sure that in the Polygon Import Setting dialog you have selected "Name" from the drop-down menu for the Name field, and you have selected "Value" in the drop-down menu for the Value field (Figure 7B). Select OK.
    vi. Turn off the *Elevation | Points1* in the **Object Browser** and the result of the drawn and imported polygons should look similar to Figure 7C (it's alright if the colors are a bit different).

3.

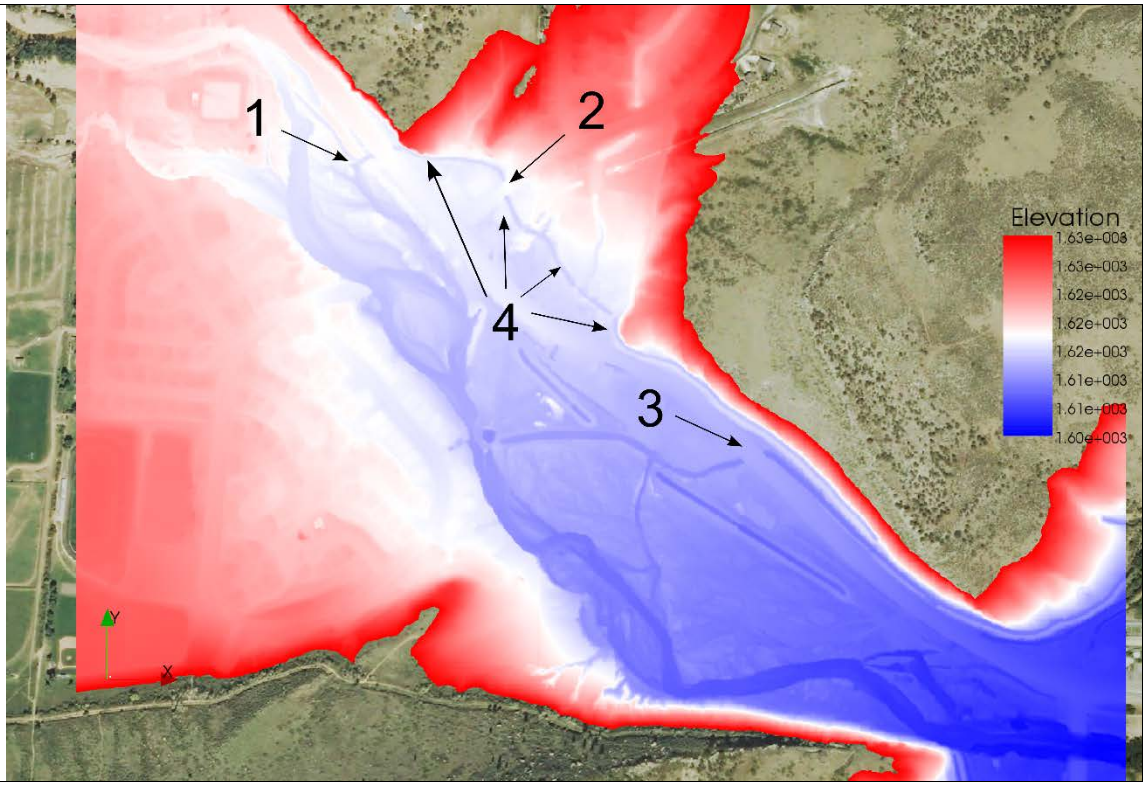


Figure 6. Location of flow diversions under roads and irrigation ditch which require some modification of the existing topography.

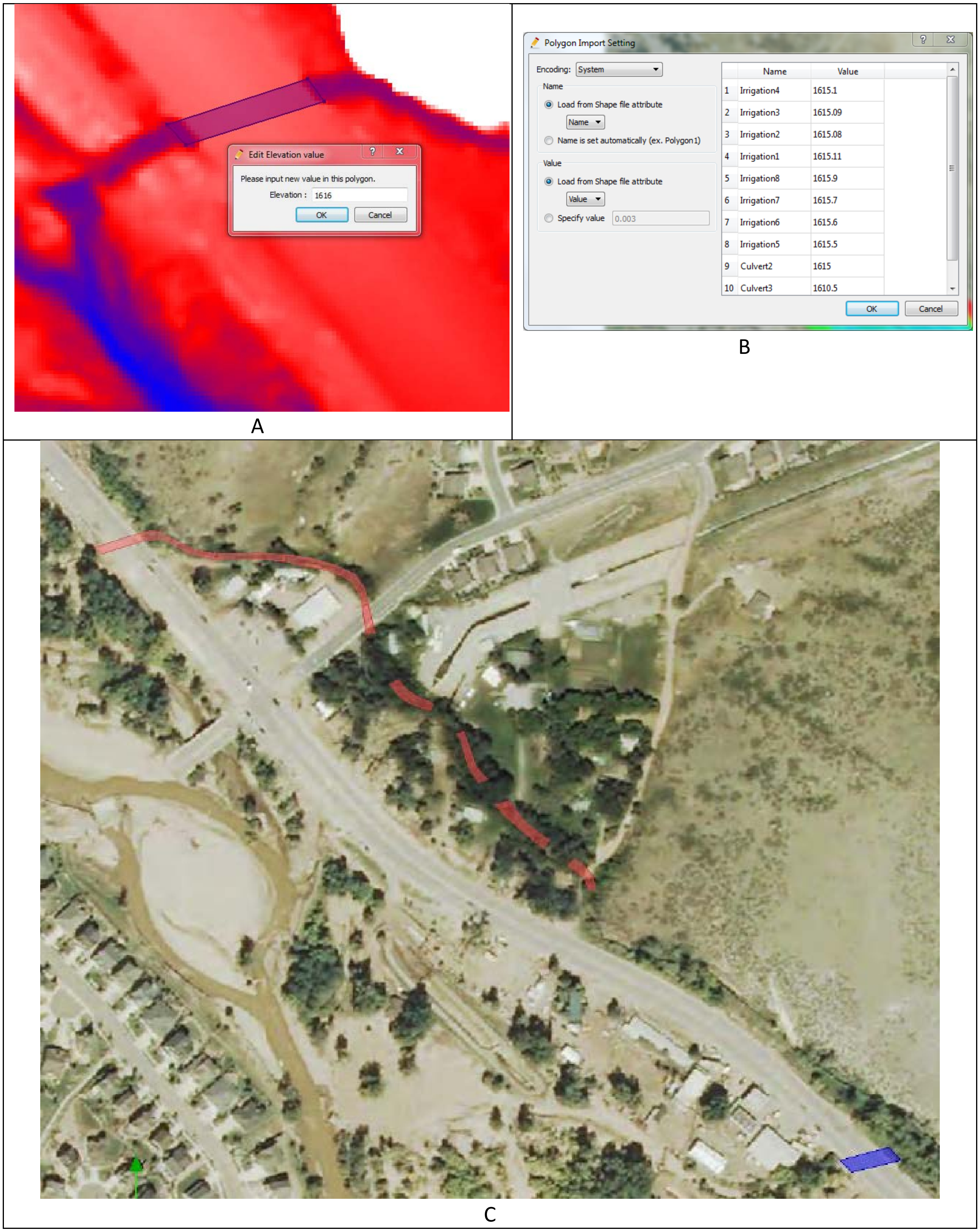


Figure 7. A) Added polygon to approximate diversion 1. B) Polygon Import Setting dialog – when importing a shapefile make sure you have selected the correct values in the drop-down list for the Name and Value fields. C) The complete set of polygons to enhance the topographic representation of diversions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

Task 3: Create Grid

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

1. **Set mapping attributes:** Select *Grid* → *Attributes Mapping* → *Setting* from the **Menu Bar** and in the resulting Grid Attribute Mapping Setting dialog, select "Manual" in the Execute mapping section of the dialog. This will prevent iRIC from mapping data to the grid following every edit to the grid. Select OK.
2. **Considerations:** In step 3 of this section, we will select a grid generator and draw a centerline, but before doing this it's important to consider the following points:
    a. Note that the centerline is drawn from upstream to downstream, in this case from upper left to lower right.
    b. In Figure 8, a simple grid is shown for context only - you won't see this as we draw our centerline. Each of the small black squares on the centerline are the points used to define its path. In this case 6 points are defined. Usually only a few points are desired. You can always add or delete centerline points later if necessary.
    c. The downstream boundary is located such that the furthest downstream measured water-surface elevation is at the boundary because we will use its value as a boundary condition for our simulation.
    d. We will want to adjust the curvature of our grid such that the boundaries are as perpendicular as possible to the main flow direction. See the insets in Figure 8. Here we plot the vectors from a simulation for illustration, but it is generally easy enough to guess the flow direction. It's possible that after an initial solution you may want to go back and adjust the location of the boundaries.
    e. Additionally, it's important to place the upstream boundary such that there is not a narrowing of the channel immediately downstream. The flow direction at the upstream boundary is set perpendicular to the boundary by default (note it is possible to adjust the flow direction at the boundary, but for now it's best and easiest to assume a perpendicular flow direction). If the channel decreases in width immediately downstream, then the flow will be directed into the bank. Looking at the upstream boundary inset in Figure 8, if the boundary was placed further downstream then you might imagine the flow at the upstream boundary would be directed into the bank downstream; the common result is for the model to fail to converge or give a poor solution in this area.
3. **Select grid generator and draw centerline:** Select the grid generator by selecting *Grid* → *Select Algorithm to Create Grid…* from the **Menu Bar** and in the resulting dialog select "Create grid from polygonal line and width" in the list of Algorithms. Select OK. An information dialog will appear. It provides a quick description on how to create the grid using this tool. Select OK. Using your left mouse button define the centerline for the grid. It should look similar to that of Figure 8. It's not important to be precise at this point. As you will see later it's important to get the boundaries of the grid perpendicular to the main flow direction and that is easily done once you have a basic grid. The aim here is to draw your centerline in a location that is close to that of Figure 8. The centerline is completed by pressing "Enter" on your keyboard.
4. **Create a simple grid to help placement of grid and boundary conditions:** In the Grid Creation dialog enter 11 for n_i, 12 for n_j, and 450 for the width (W) and then select OK (Figure 9A). You can create a finer grid once you have edited our initial guess of centerline location.
5. **Adjust your centerline and grid curvature:** Adjust the centerline point locations until you have a grid that encompasses the flow area, provides good boundary angles to the incoming and outgoing flow directions, and has a downstream boundary located at a measured water-surface elevation point which we'll use as a boundary condition.

   a. By adjusting the centerline points of the grid you can adjust the grid curvature, and the orientation of the upstream and downstream boundaries. First, select *Grid Creating Condition* in the **Object Browser**. This will turn on the centerline created in step 3. You can adjust the location of each centerline point by moving the mouse over the point and using a mouse left-click and drag to move it. If you don't like your move select the undo button ( ) in the **Main Toolbar**.

6. **<u>Rebuild grid to final 2-meter resolution:</u>** Once you are satisfied with the location of your grid, rebuild the grid such that it has a 2 X 2 meter spacing between nodes. From the **Menu Bar** select *Grid → Create Grid* and enter values for n_i and n_j, such that the resulting values of d_i and d_j are approximately 2 meters. You can iterate on these values by entering your guess into n_i and n_j, then select the "Apply" button to see the resulting values of d_i and d_j (Figure 9A). Select OK when done.
7. **<u>Map measured values to the grid:</u>** Map or interpolate all geographic data to the grid, in this case just the elevation data. We will map values to the grid using the TIN of the values. This is the default interpolation method. This value is set from the **Menu Bar** by selecting *Grid → Attributes Mapping → Setting*. Note that Geographic data mapping is set to *Mapping with TINs* in the Grid Attribute Mapping Setting dialog (Figure 9B). Select OK to exit the dialog. Next you can execute the mapping. From the **Menu Bar** select *Grid → Attributes Mapping → Execute*. Check Elevation and select OK. If you turn on the *Grid | Node Attributes | Elevation* in the **Object Browser**, your project should look similar to Figure 9C. You may have to experiment with different dialogs and settings.

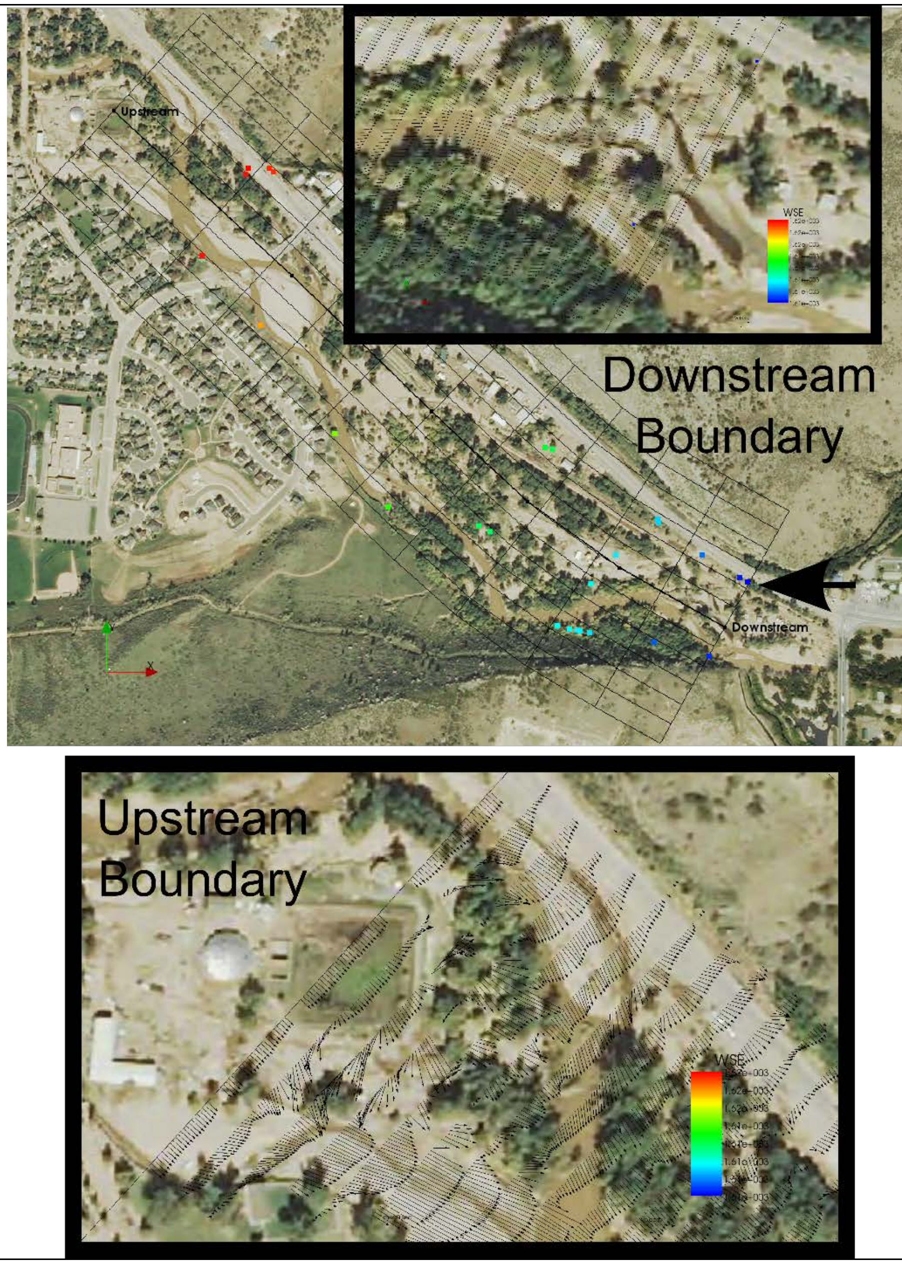


Figure 8. For illustration purposes a simple grid developed from the centerline is shown. Importantly 1) the upstream and downstream grid boundaries are aligned nearly perpendicularly to the incoming and outgoing flow directions, 2) there are no constrictions in flow immediately downstream of the boundary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

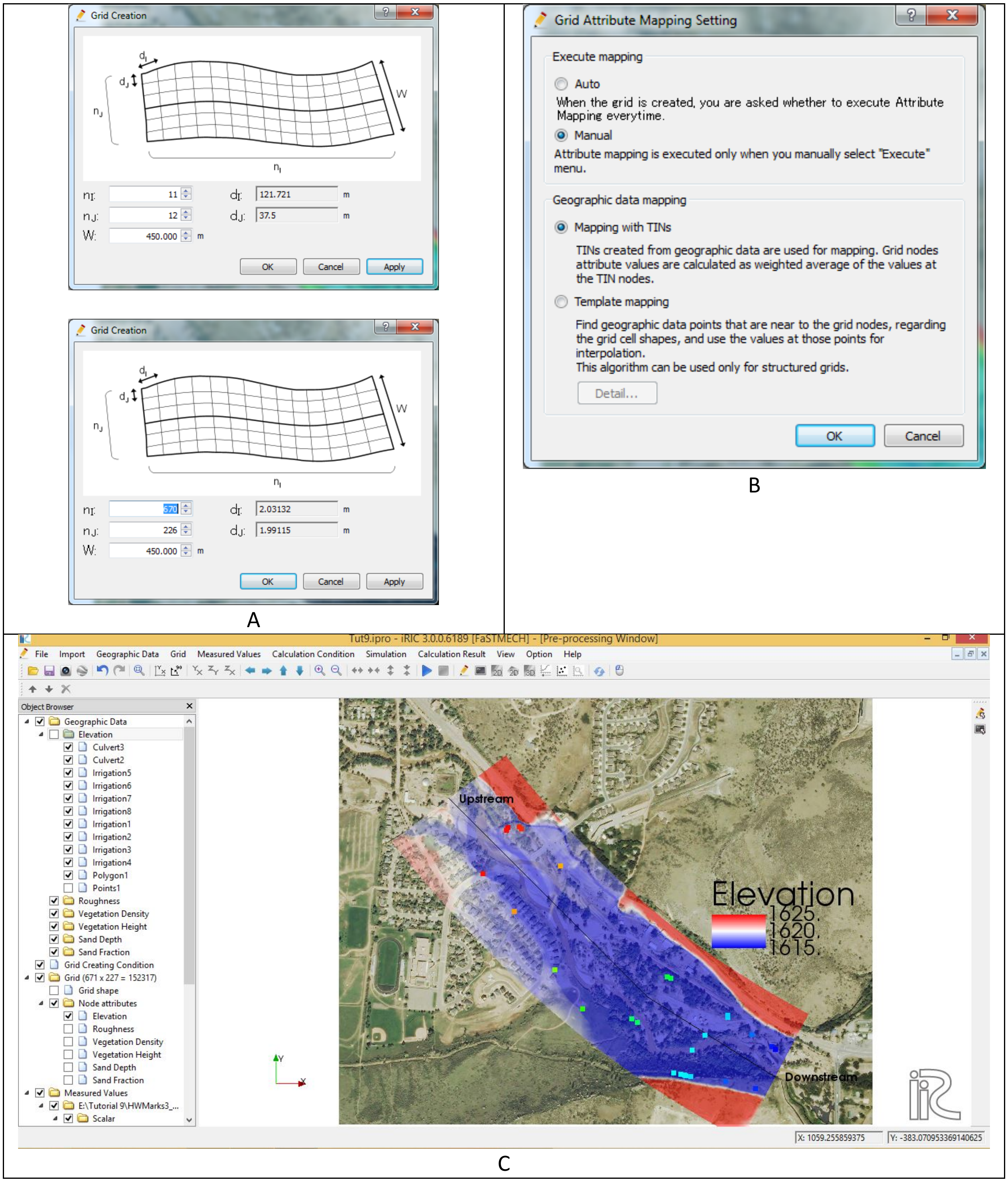


Figure 9. A) Grid Creation parameters for the simple grid in Step 4 (top) and for the final refined grid (bottom). B) The Grid Attribute Mapping dialog with the mapping method set to use the TIN. C) The elevation as mapped to the computational grid. Note that the location of the weirs are approximated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

Task 4: Determine the peak discharge

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

Part A

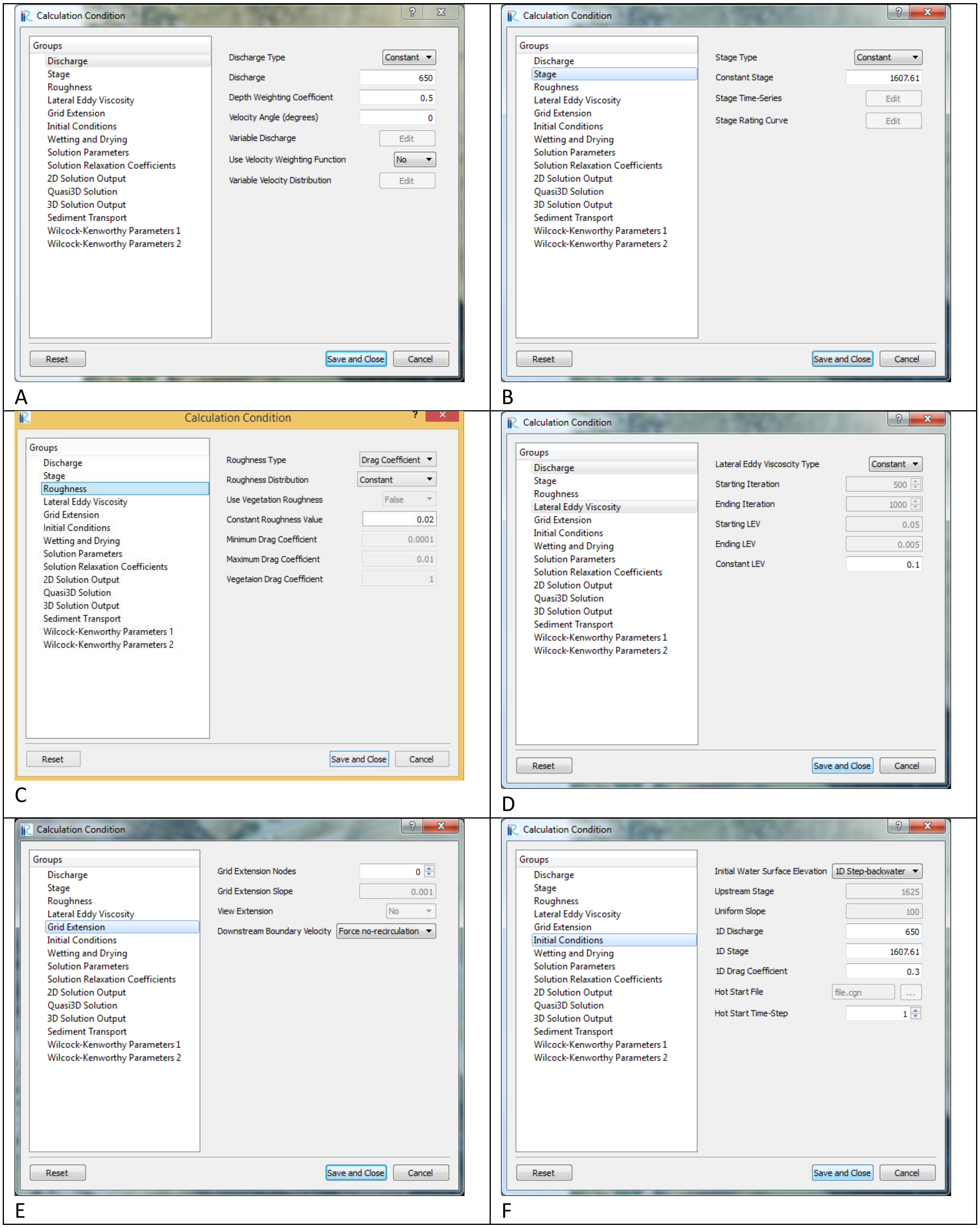
To help you get started the input conditions will be provided for the 650cms flow with a roughness of 0.02. The input conditions are provided below in Figure 10. A few notes before you begin:

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

1. **<u>Enter the calculation conditions for a simulation:</u>** From the **Menu Bar** select *Calculation Condition* → *Setting*, and fill in the values for each tab of the Calculation Condition dialog as those in Figure 10, except as explained above where you will enter the discharge corresponding to the discharge you are modeling in the *Discharge* and *Initial Conditions* tabs.
2. **<u>Run the simulation:</u>** From the **Menu Bar** select *Simulation* → *Run*.
3. **<u>Compare the simulated water surface elevation to the measured HWMs and record the RMSE value:</u>** When the simulation is finished, select *Calculation Result* → *Compare with measured values…* from the **Menu Bar**, which will launch the *Verification* dialog. Initially it starts with the Verification Setting dialog (Figure 11A). Set the Physical Value in the drop-down list to "WaterSurfaceElevation" and select OK.
    a. In the Verification dialog you will see a comparison between the measured data (circles) and simulation (line) as a function of streamwise distance (Figure 11B). Note that in the Verification dialog, the RMSE, a measure of the mean difference between the measured and simulated data is displayed in the dialog below the comparison plot. Record this number down in the appropriate column of Table 1.
    b. You now know how to get a comparison between measured and simulated water-surface elevations in iRIC. (Figure 11 B, C, and D). You will fill your Table 1 with the RMSE value for each discharge and drag coefficient pair.

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

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


Calculation Condition
Groups
Discharge
Stage
Roughness
Lateral Eddy Viscosity
Grid Extension
Initial Conditions
Wetting and Drying
Solution Parameters
Solution Relaxation Coefficients
2D Solution Output
Quasi3D Solution
3D Solution Output
Sediment Transport
Wilcock-Kenworthy Parameters 1
Wilcock-Kenworthy Parameters 2
Discharge Type Constant
Discharge 650
Depth Weighting Coefficient 0.5
Velocity Angle (degrees) 0
Variable Discharge Edit
Use Velocity Weighting Function No
Variable Velocity Distribution Edit
Reset
Save and Close
Cancel
A
Stage Type Constant
Constant Stage 1607.61
Stage Time-Series Edit
Stage Rating Curve Edit
B
Roughness Type Drag Coefficient
Roughness Distribution Constant
Use Vegetation Roughness False
Constant Roughness Value 0.02
Minimum Drag Coefficient 0.0001
Maximum Drag Coefficient 0.01
Vegetaion Drag Coefficient 1
C
Lateral Eddy Viscoscity Type Constant
Starting Iteration 500
Ending Iteration 1000
Starting LEV 0.05
Ending LEV 0.005
Constant LEV 0.1
D
Grid Extension Nodes 0
Grid Extension Slope 0.001
View Extension No
Downstream Boundary Velocity Force no-recirculation
E
Initial Water Surface Elevation 1D Step-backwater
Upstream Stage 1625
Uniform Slope 100
1D Discharge 650
1D Stage 1607.61
1D Drag Coefficient 0.3
Hot Start File file.cgn
Hot Start Time-Step 1
F

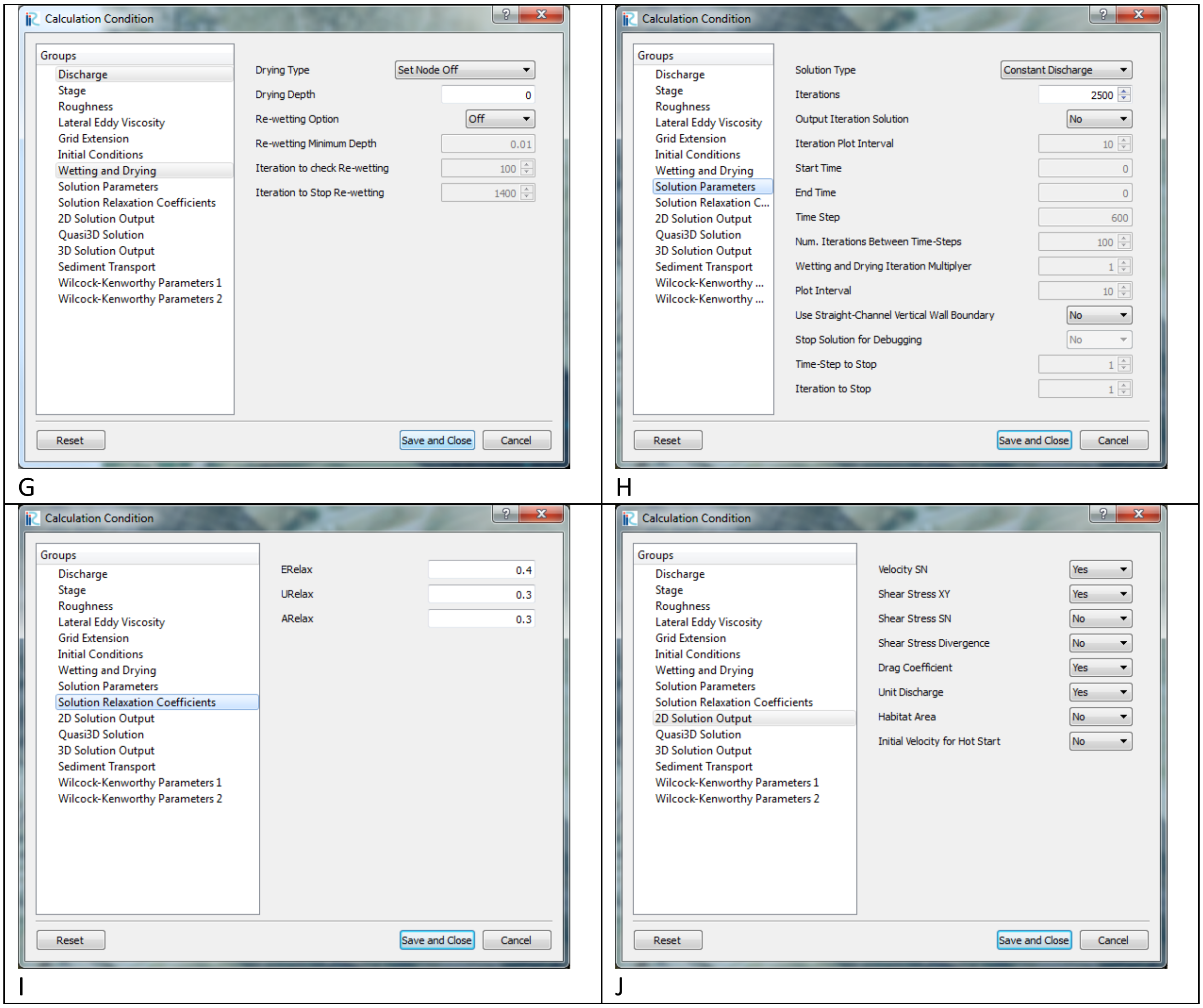


Figure 10. The calculation conditions for the FaSTMECH model accessed by selecting *Calculation Condition* → *Setting* from the menu. Each tab for a flow solution (without sediment transport) is shown including: A) Discharge, B) Stage, C) Roughness, D) Lateral Eddy Viscosity, E) Grid Extension, F) Initial Conditions, G) Wetting and Drying, H) Solution Parameters, I) Solution Relaxation Coefficients, and J) 2D Solution Output .

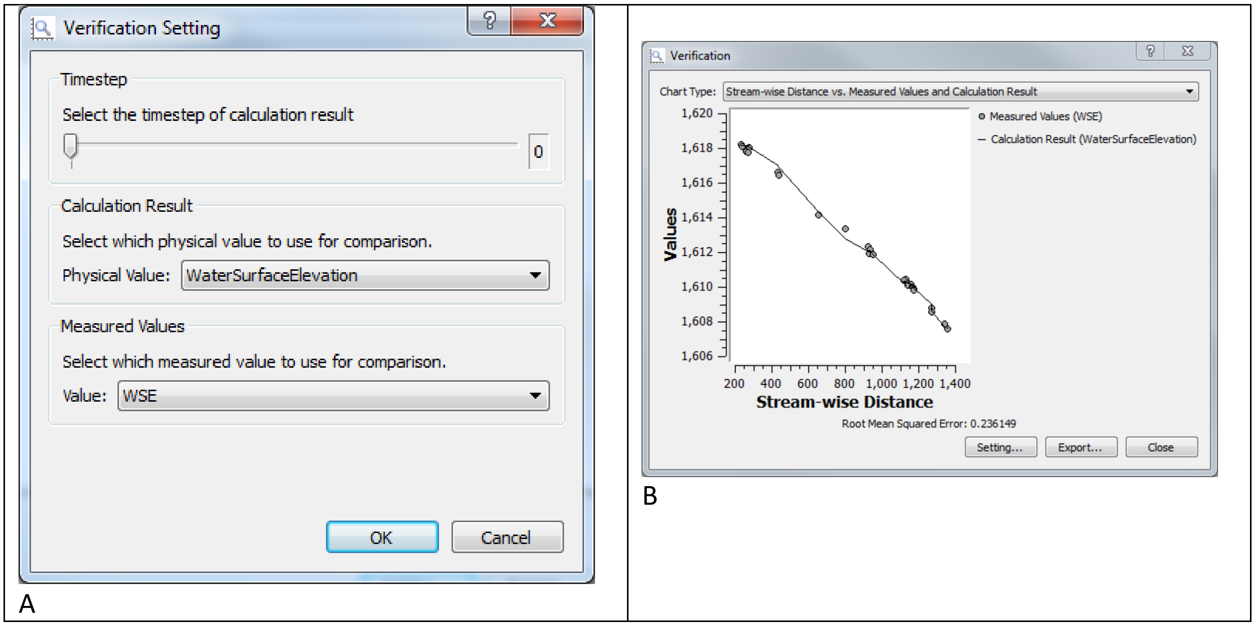


Figure 11. The Verification dialog can be accessed by selecting *Calculation Result* → *Compare* with measured values from the menu. A) The initial dialog provides the setting to choose which simulation scalar set will be used to compare with the measured data- in this case WaterSurfaceElevation is chosen.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

Part B

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

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

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

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

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