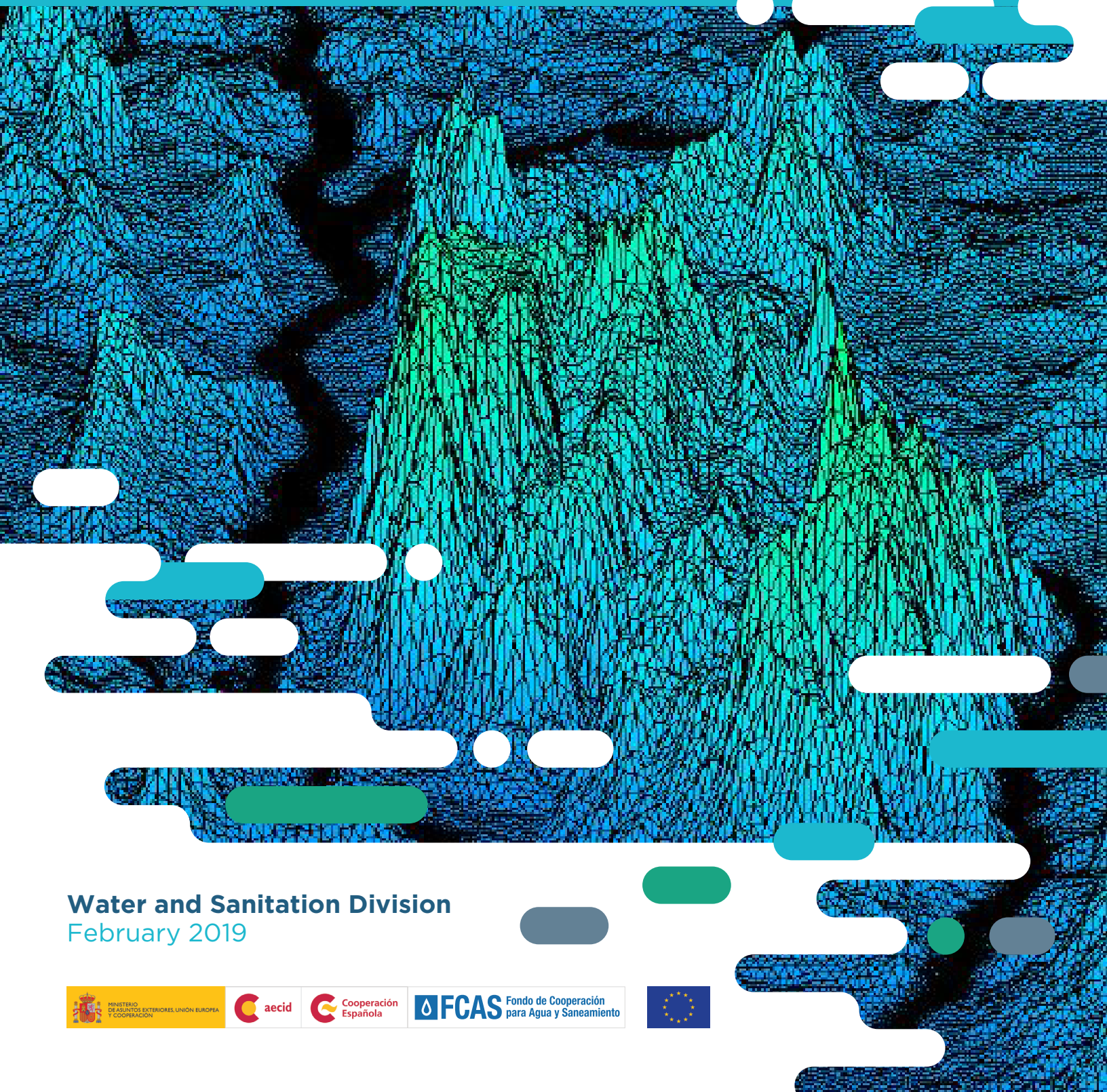
HydroBID

HydroBID Flood Tutorials

Two-dimensional Flood and River Dynamics Model



Water and Sanitation Division
February 2019

MINISTERIO DE ASUNTOS EXTERIORES, UNIÓN EUROPEA Y COOPERACIÓN
aecid
Cooperación Española
FCAS Fondo de Cooperación para Agua y Saneamiento

Content

List of Figures v

1 Introduction 1

2 Creating your First HydroBID Flood Project 2
2.1 Starting a new project . . . 2
2.1.1 Starting QGIS . . . 3
2.1.2 Start a new HydroBID Flood project . . . 3
2.1.3 Load elevation data . . . 6
2.1.4 Create the limits of the modeling area . . . 11
2.1.5 Generating the triangular-cell mesh . . . 14
2.1.6 Setting the boundary conditions . . . 16
2.1.7 Assigning Manning's n . . . 19
2.2 Exporting the files . . . 21

3 Simulating Levees using Weirs 24
3.1 Open an existing project . . . 25
3.2 Create a Weir layer and the weir polyline . . . 25
3.3 Generate the mesh . . . 29
3.4 Exporting files to RiverFlow2D . . . 30
3.5 Running the model . . . 31
3.6 Review the output files . . . 33

4 Simulating bridges 35
4.1 Create a bridge geometry file . . . 35
4.2 Open an existing project . . . 38
4.3 Enter the bridge polyline in the *Bridges* layer . . . 39
4.4 Generate the mesh . . . 42
4.5 Exporting files to HydroBID Flood . . . 43
4.6 Running the Model . . . 44

5 Simulating culverts 47
5.1 Open an existing project . . . 48
5.2 Create Culverts layer and draw the Culvert . . . 49

5.3 Generate the mesh . . . 52
5.4 Exporting files to HydroBID Flood . . . 53
5.5 Running the model . . . 54
5.6 Review culvert output file . . . 56

6 Simulating dam breaches 58
6.1 Open an existing project . . . 59
6.2 Create the DamBreach layer and draw the line that defines the dam . . . 59
6.3 Generate the mesh . . . 62
6.4 Exporting files to HydroBID Flood . . . 63
6.5 Running the model . . . 64
6.6 Review the output files . . . 66

7 Using Manning's n ESRI shape files 68
7.1 Open an existing project . . . 68
7.2 Load the shape file with the Manning's n polygons . . . 69
7.3 Import the Mannings n geometry and values to the Manning N layer . . . 70

8 Simulating bed load sediment transport with limited erosion bed areas 75
8.1 Open an existing project . . . 75
8.2 Add template of the MaximumErosionDepth layer and draw the polygon that define the area of limited erosion . . . 76
8.3 Generate the mesh . . . 79
8.4 Exporting files to HydroBID Flood . . . 80
8.5 Running the model . . . 80
8.6 Check the output files . . . 82

9 Hydrologic simulations 84
9.1 Create the rainfall and evaporation time series data file . . . 85
9.2 Create the infiltration parameters data file . . . 86
9.3 Open an existing project . . . 86
9.4 Add the *RainEvap* component layer, and the rainfall/evaporation polygons . . . 87
9.5 Add the *Infiltration* component layer, and the Infiltration polygons . . . 89
9.6 Generate the mesh . . . 93
9.7 Exporting files to RiverFlow2D . . . 94
9.8 Running the model . . . 94
9.9 Review the output files . . . 97

10 Post-processing calculations 98
10.1 Open an existing project . . . 98
10.2 Create a template of the layers *ObservationPoints*, *CrossSections* and *Profiles* and draw the output controls . . . 99

10.3 Generate the mesh . . . 106
10.3.1 Exporting files to HydroBID Flood . . . 107
10.4 Running the Model . . . 108
10.5 Review output files . . . 110

11 Advanced digitization/Snapping 114
11.1 Open an existing project . . . 115
11.2 Activate the Snapping tool . . . 116
11.3 Configure the snapping tool . . . 117
11.4 Draw contiguous or adjacent polygons using the Snapping tool . . . 118
11.5 Delete a polygon . . . 125

12 Creating raster elevations from X Y Z data sets 127

List of Figures

2.1 Files with data required for the example. . . . 2
2.2 File containing terrain elevation points. . . . 3
2.3 QGIS interface indicating window areas. . . . 3
2.4 Coordinate Reference System Selector dialog window. . . . 4
2.5 Create New HydroBID Flood Project. . . . 5
2.6 Layers created for the project. . . . 5
2.7 Dialog to create a layer from delimited data text file. . . . 7
2.8 How to zoom to layer extent. . . . 8
2.9 Layer of points created from delimited data file. . . . 8
2.10 Loading to the QGIS interpolation plugin. . . . 9
2.11 Interpolation plugin window. . . . 9
2.12 Digital elevation model in raster format created by interpolation. . . . 10
2.13 Window to change the raster layer render style. . . . 10
2.14 Digital elevation model with color render. . . . 11
2.15 Menu buttons for digitalization toolbar. . . . 11
2.16 Modeling area delimitation on the *Domain Outline* layer. . . . 12
2.17 *CellSize* defined for the *Domain Outline* layer. . . . 13
2.18 Domain Outline layer. . . . 14
2.19 Resulting mesh. . . . 15
2.20 Message panel of the registry with GMSH messages. . . . 15
2.21 Polygon that covers the nodes defining the Inflow boundary condition segment. . . . 16
2.22 Inflow Boundary Condition data. . . . 17
2.23 Polygon that includes the Outflow boundary condition nodes. . . . 17
2.24 Attributes for the polygon layer that defines the outflow boundary condition. . . . 18
2.25 Polygons that define the inflow and outflow boundary conditions. . . . 19
2.26 Editing the *Manning N* layer. . . . 20
2.27 Dialog to input *Manning N* Feature Attributes. . . . 20
2.28 Export HydroBID Flood dialog. . . . 21
2.29 HydroBID Flood project screen after exporting the files. . . . 22
2.30 Hydronia Data Input Program window. . . . 22
2.31 The graph generated by HydroBID Flood. . . . 23

3.1 The weir polyline representing the levee is shown in red. . . . 24
3.2 Example of the tutorial loaded in QGIS. . . . 25
3.3 Text file with weir alignment data. . . . 26
3.4 Plugin to add a new template layer. . . . 27
3.5 Window to input Weir parameters. . . . 28
3.6 Window for weir geometry data. . . . 28
3.7 Weir alignment loaded from the file. . . . 29
3.8 The mesh generated. . . . 30
3.9 Plugin window to export the files to RiverFlow2D. . . . 31
3.10 Control data panel. . . . 31
3.11 Weirs component data panel. . . . 32
3.12 HydroBID Flood output graphics. . . . 32
3.13 Extract of the output file of the Weir1. . . . 34

4.1 Bridge geometries. . . . 36
4.2 Front view of the bridge. . . . 36
4.3 Data of the bridge geometry. . . . 37
4.4 Bridges panel. . . . 38
4.5 Project screen loaded in QGIS. . . . 39
4.6 HydroBID Flood dialog to add new layer. . . . 40
4.7 Bridge drawing. . . . 41
4.8 Attribute dialog window of the *Bridges* layer. . . . 42
4.9 Mesh aligned with bridge polyline. . . . 43
4.10 Export HydroBID Flood tool. . . . 44
4.11 Control Data panel. . . . 44
4.12 Bridges component data panel. . . . 45
4.13 RiverFlow2D output graphics. . . . 45
4.14 Bridge hydrograph file `.bridgeh`. . . . 46

5.1 Culvert scheme. . . . 47
5.2 Example of the tutorial loaded in QGIS. . . . 49
5.3 Plugin to add a New Template Layer. . . . 50
5.4 Culvert alignment. . . . 51
5.5 Window to input Culvert parameters. . . . 52
5.6 Final mesh. . . . 53
5.7 Plugin window to export the files. . . . 54
5.8 Control data panel. . . . 54
5.9 Culverts component data panel. . . . 55
5.10 Runtime graphics. . . . 56
5.11 Culvert1 output file. . . . 57

6.1 Final dimensions of the dam breach. . . . 58

6.2 Project loaded in QGIS. 59
6.3 Dialog to add a new layers. 60
6.4 Dam axis. 61
6.5 Dialog to input the dam breach parameters. 61
6.6 Evolution of the breach of the dam. 62
6.7 Dam axis. 62
6.8 The resulting dam breach mesh. Detail show mesh along the dam axis. 63
6.9 Export dialog. 64
6.10 Hydronia Data Input Program. 64
6.11 *Dam Breach* component. 65
6.12 HydroBID Flood output graphics. 66
6.13 Extract of the `DamBreach.dambreachh` file 67

7.1 Project loaded in QGIS. 69
7.2 Window to find and open the shape file. 69
7.3 SaltRiver_ManningsN Layer. 70
7.4 Attribute table of the *SaltRiver_ManningsN* layer. 71
7.5 Properties the *SaltRiver_ManningsN* layer. 72
7.6 Field properties of the edited *SaltRiver_ManningsN* layer. 72
7.7 Attribute table of the *Manning N* layer. 74

8.1 Project screen loaded in QGIS. 76
8.2 Plugin to add a New Template Layer. 77
8.3 Polygon of the area with limited erosion. 78
8.4 Window for input the polygon parameters of *MaximumErosionDepth* layer. 78
8.5 The mesh generated. 79
8.6 Plugin window to export the files to HydroBID Flood. 80
8.7 Control data panel. 81
8.8 HydroBID Flood Sediment Transport Module. 81
8.9 HydroBID Flood output graphics. 82
8.10 Maps of elevation difference of the river bed between the initial time and at the end of the run. 83

9.1 Rain intensity time series. 86
9.2 Rainfall and infiltration tutorial loaded in QGIS. 87
9.3 Dialog to add a new layer. 88
9.4 Polygon with the spatial distribution of Rainfall/evaporation data. 89
9.5 Plugin to add a new template layer. 90
9.6 Infiltration areas of the watershed. 91
9.7 Polygon of the infiltration2 area of the watershed. 91
9.8 Window to input the parameters for the infiltration2 polygon. 92
9.9 Polygons of the infiltration areas of the watershed. 92

9.10 Dialog to enter the parameters for the infiltration1 polygon. 93
9.11 Final mesh. 93
9.12 Export dialog. 94
9.13 Control data panel. 95
9.14 Rainfall/Evaporation component data panel. 95
9.15 Infiltration component data panel. 96
9.16 RiverFlow2D output graphics. 96
9.17 Output hydrograph of output. 97

10.1 Project screen loaded in QGIS. 99
10.2 Plugin to add the new templates layer. 100
10.3 Attribute windows for the three cross sections. 101
10.4 Transversal sections incorporated into the model. 102
10.5 Attribute window for the profile. 103
10.6 Profile (green line) incorporated into the model. 104
10.7 Attribute windows for the two observation points. 105
10.8 Observation points incorporated into the model. 106
10.9 The mesh generated. 107
10.10 Plugin to export the files to HydroBID Flood. 108
10.11 Control data panel. 109
10.12 River Flow2D output graphics. 110
10.13 OutControl.XSECE file. 111
10.14 OutControl.PRFE file. 112
10.15 RESvsT_0004938.oute file. 113

11.1 Areas with overlap and empty spaces in manual digitization of adjacent polygons. 114
11.2 Project screen loaded in QGIS. 115
11.3 QGIS Options / Digitizing Window. 116
11.4 QGIS with snapping panel. 117
11.5 Snapping configuration panel. 117
11.6 The first polygon sketch. 119
11.7 Drawn of the second polygon. 119
11.8 Final draw of the second polygon. 120
11.9 Preliminary draw of the third polygon. 120
11.10 Final draw of the third polygon. 121
11.11 Preliminary draw of the fourth polygon. 122
11.12 Final draw of the fourth polygon. 122
11.13 Preliminary draw of the fifth polygon. 123
11.14 Final draw of the fifth polygon. 124
11.15 Final draw of the *Manning N* layer. 124
11.16 Polygon to be deleted selected. 125

11.17 Manning N layer with the polygon removed. 126

12.1 File containing terrain elevation points. 127
12.2 Dialog to create a layer from delimited data text file. 128
12.3 How to zoom a layer. 129
12.4 Layer of points created from delimited data file. 129
12.5 Loading to the QGIS interpolation plugin. 130
12.6 Interpolation plugin window. 130
12.7 Digital elevation model in raster format created by interpolation. 131
12.8 Window to change the render style of a raster layer. 132
12.9 Digital elevation model with color render. 133

1 — Introduction

This document presents HydroBID Flood model instructional material and tutorials to facilitate the application of several model components using QGIS. HydroBID Flood is a software system based on the RiverFlow2D model develped by Hydronia LLC.

RiverFlow2D is a combined hydrologic and hydrodynamic, mobile bed, and pollutant transport finite-volume model for rivers, estuaries, coastal areas, and floodplains. It can simulate floods and inundation over complex terrain at high resolution and with remarkable stability, accuracy and speed. The use of adaptive triangular-cell meshes enables the flow field to be resolved around key features in difficult river environments.

Hydronia computational engines use an accurate, fast and stable finite-volume solution method that eliminates the boundary and hot start difficulties of some two-dimensional flexible mesh models. RiverFlow2D also includes capability to represent dam or levee breaches and can integrate hydraulic structures such as culverts, weirs, bridges, and gates. The hydrologic capabilities include spatially distributed rainfall, evaporation and infiltration. This current version can integrate the effect of spatially distributed wind stresses on the water surface.

OilFlow2D is a two-dimensional model that simulates spreading of oil and viscous fluids on the land surface. It can simulate oil depths and velocity over irregular surfaces considerint fluid properties such as density, viscosity that can vary in time. The model can use 3 different rheogological formulations for Newtonian and non-Newtonian fluids that can have Yield Stress. The OilFlow2D model also accounts for oil evaporation and that process affects density and viscosity and its subsequent impact on fluid flow.

The user interface of Hydronia models is based on QGIS, a powerful software for the analysis of geographic information systems of free distribution. This software system integrated with GMSH mesh generation plugin provides interactive functions to generate and refine the finite-volume flexible mesh, and uses familiar GIS objects to construct a high level representation of the model. QGIS also offers a complete set of visualization tools that include rendering, for the representation of the results generated by the model.

2 — Creating your First HydroBID Flood Project

This section provides a step-by-step guide to help you get started with a HydroBID Flood project using the QGIS interface. This procedure will include instructions to enter the terrain elevation data, create the mesh, prepare the layers with the input information, and run HydroBID Flood. The boundary conditions include an inflow hydrograph and uniform outflow.

2.1 Starting a new project

Before starting the project with QGIS. It is recommended to create a folder that would contain all the necessary data such as the boundary conditions files, the elevation data files, either in ASCII or in raster format, etc. In this section we will use the data of the example included in the `Hoh_QGIS` folder under `My Documents\RiverFLow2D_QGIS\ExampleProjects`. The folder contains the files shown:

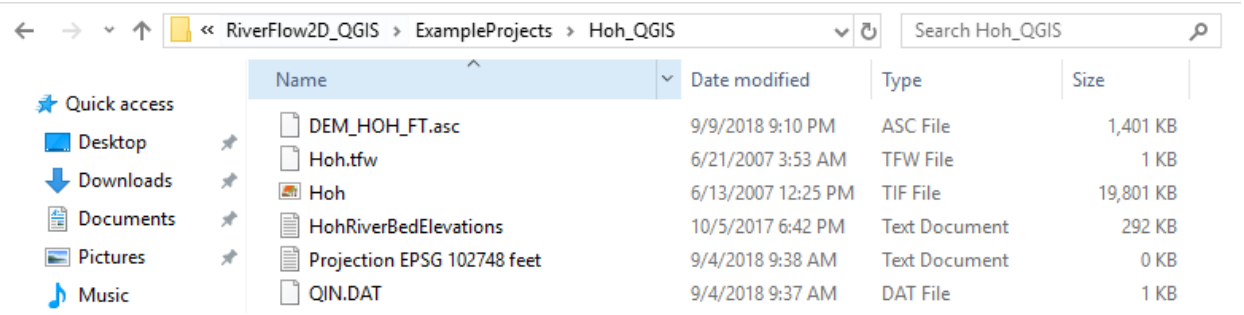


Figure 2.1 – Files with data required for the example.

The file `HohRiverBedElevations.txt` is formatted to comma separated data, where the first two columns correspond to point X and Y coordinates and the third column is the point elevation as shown in the following figure.

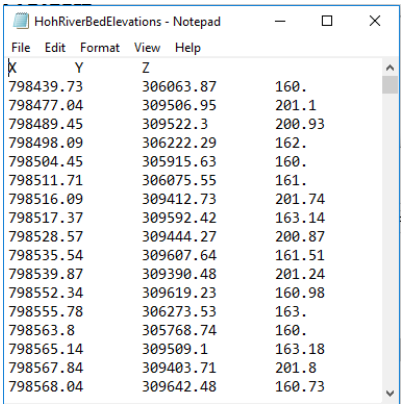


Figure 2.2 – File containing terrain elevation points.

2.1.1 Starting QGIS

The first step is to start the QGIS software. After loading we will have a window similar to the one shown below:

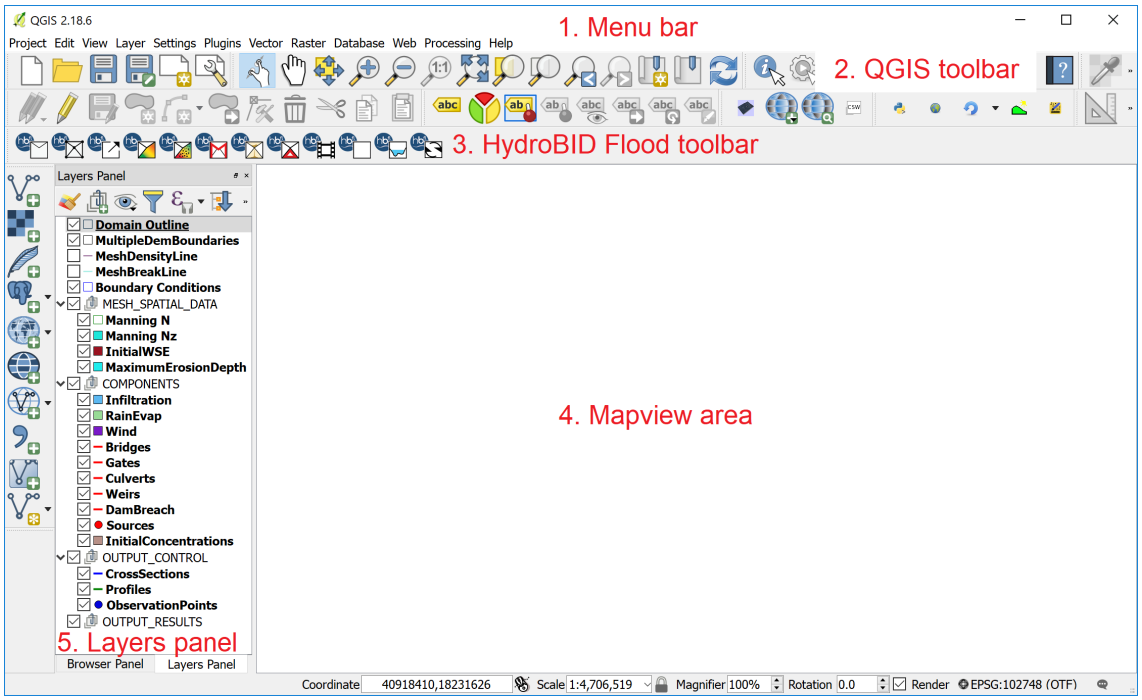


Figure 2.3 – QGIS interface indicating window areas.

2.1.2 Start a new HydroBID Flood project

1. In the HydroBID Flood toolbar, click on the *New HydroBID Flood Project* button

to start a new HydroBID Flood project. A dialog window appears where you select the layers that will be created, the Coordinate Reference System (CRS), and the directory path where the layers will be saved. This example will use the basic layers: *Domain Outline*, *Manning N*, and *BoundaryConditions*

2. Select *None* in the Layers dropdown menu.

3. Select the *Projection* button. Note that it is EPSG:4326 is set as default, but should be changed!.

4. In the Filter textbox, type 102748 and select the *Coordinate Reference System* as shown:

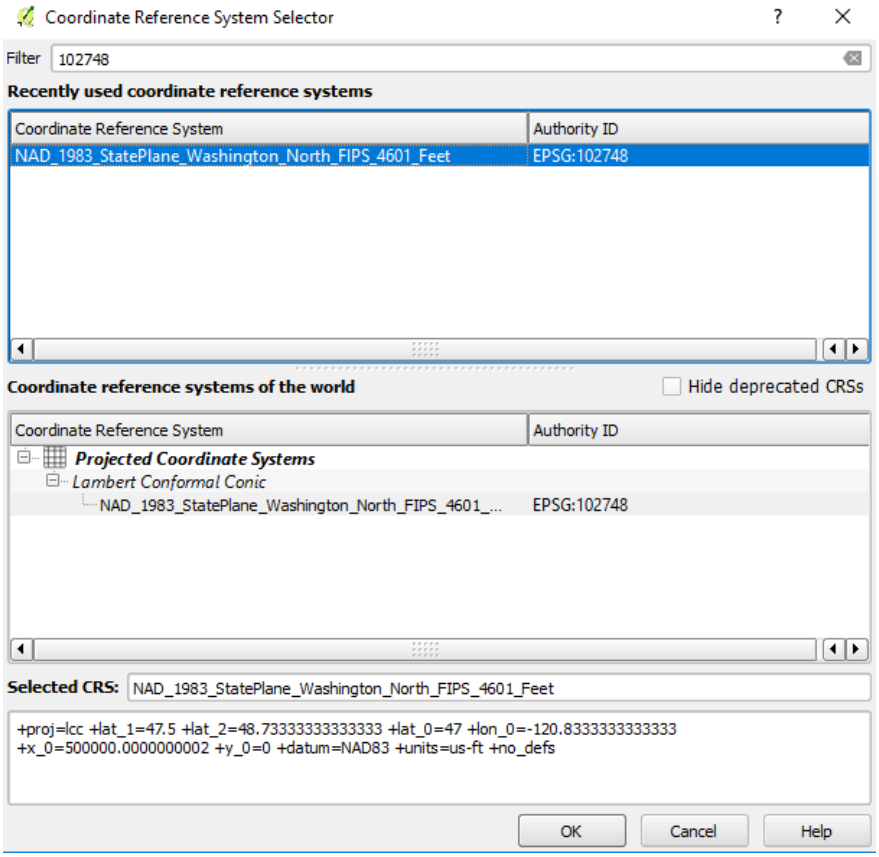


Figure 2.4 – Coordinate Reference System Selector dialog window.

5. Click OK.

6. The Coordinate Reference System (CRS) EPSG code: 102748 should be selected, and the dialog window will look like this:

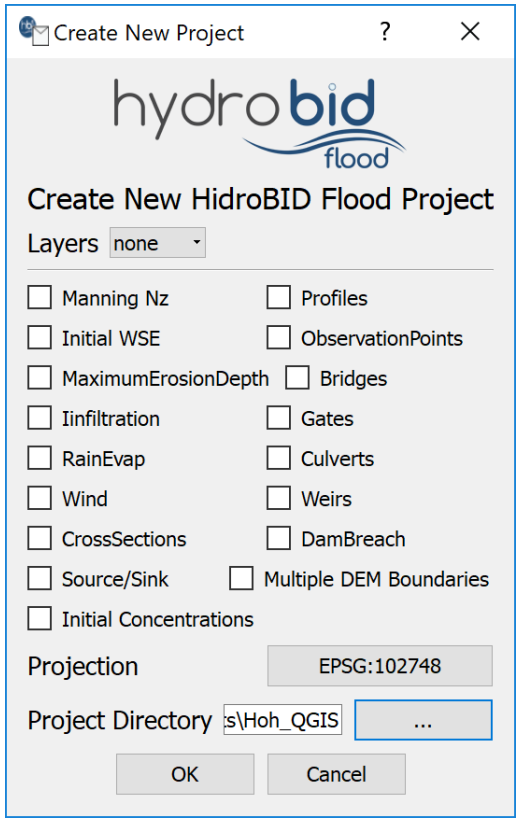


Figure 2.5 – Create New HydroBID Flood Project.

7. Click the "..." button to provide a path to store the project files in the *Project Directory* textbox. This will be the folder where the model will write all results and output files.

8. After clicking OK, the layer templates are created, and displayed on the *Layers Panel*:

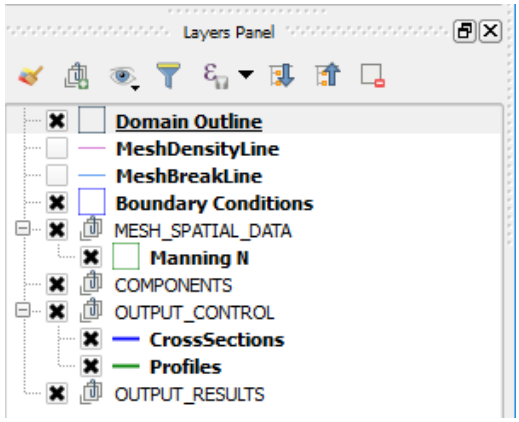


Figure 2.6 – Layers created for the project.

HydroBID Flood will use the unit system as that defined in the projection you selected. If the projection has coordinates in feet, units will be set to English. If the projection coordinates are in meters, units will be set to Metric or SI.

9. On the QGIS *Project* menu, click *Save*, to save the project in the same directory that you previously selected in the *Create New Project* dialog above.

2.1.3 Load elevation data

HydroBID Flood uses elevation data in raster format. For instance, to load an ASCII grid file, from the *Layer* menu, click *Add Layer*, and then click *Add Raster Layer...* You may also click the *Add Raster Layer* button:

In this case we will use the elevation data in a tab delimited file. First, the file is loaded to create an event layer. This can be accomplished as follows:

1. From the *Layer* menu, click *Add Layer*, and then click *Add Delimited Text Layer...* Alternatively, you may also click the *Add Delimited Text Layer* button:

the following dialog is presented:

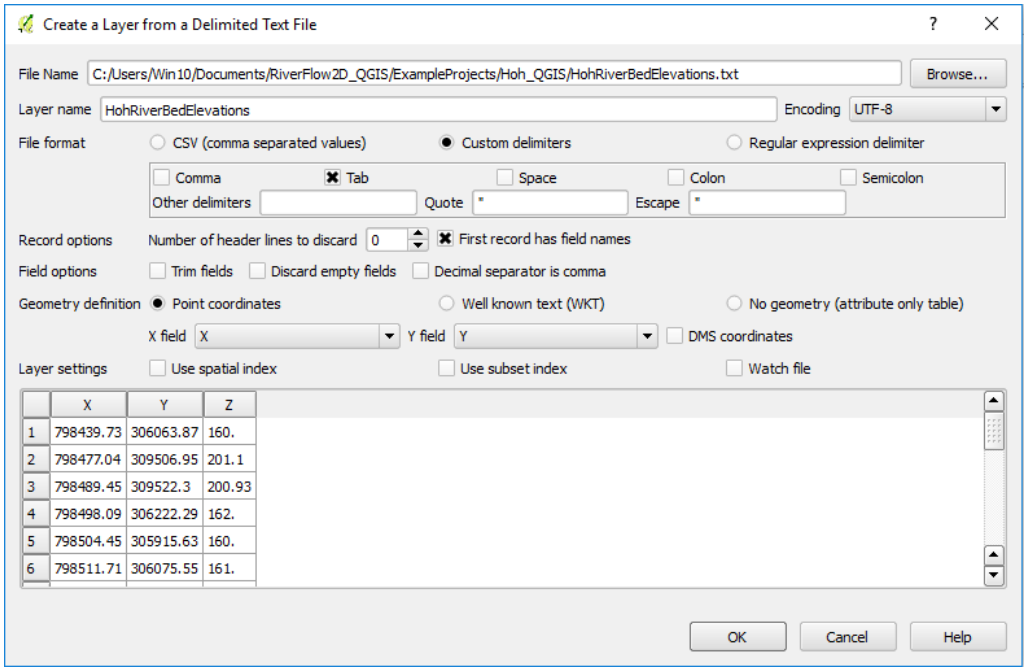


Figure 2.7 – Dialog to create a layer from delimited data text file.

2. Here you must indicate the location and name of the file with the data in this case: Click the Browse button and go to `C:\"username"\Documents\RiverFLow2D_QGIS\ExampleProjects\HohRiverBedElevations.txt`. Under *File format* select *Custom delimiters* and *Tab*.

3. Make sure that the first row has the names of the fields, and this is assigned to *X field*, *field_1* and *Y field*, *field_2*, then click OK to load the points in the project window. The Coordinate Reference System Selector dialog box appears again and we can click OK to continue, the correct CRS 102748 should already be present.

4. As surely the points are not within the extension of the window, you will have to click with the right mouse button on the label of the created layer and select the option Zoom to the layer.

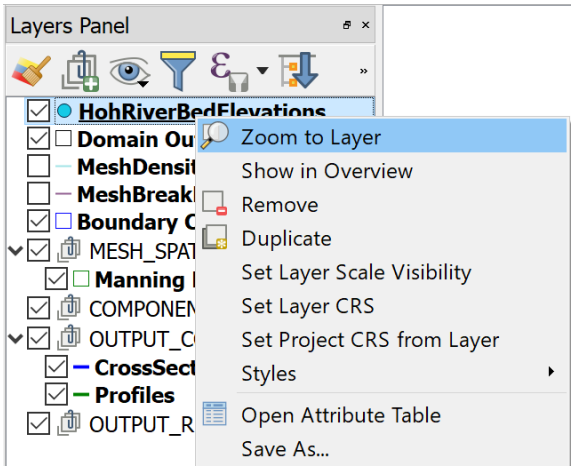


Figure 2.8 – How to zoom to layer extent.

5. The points should appear in the project window:

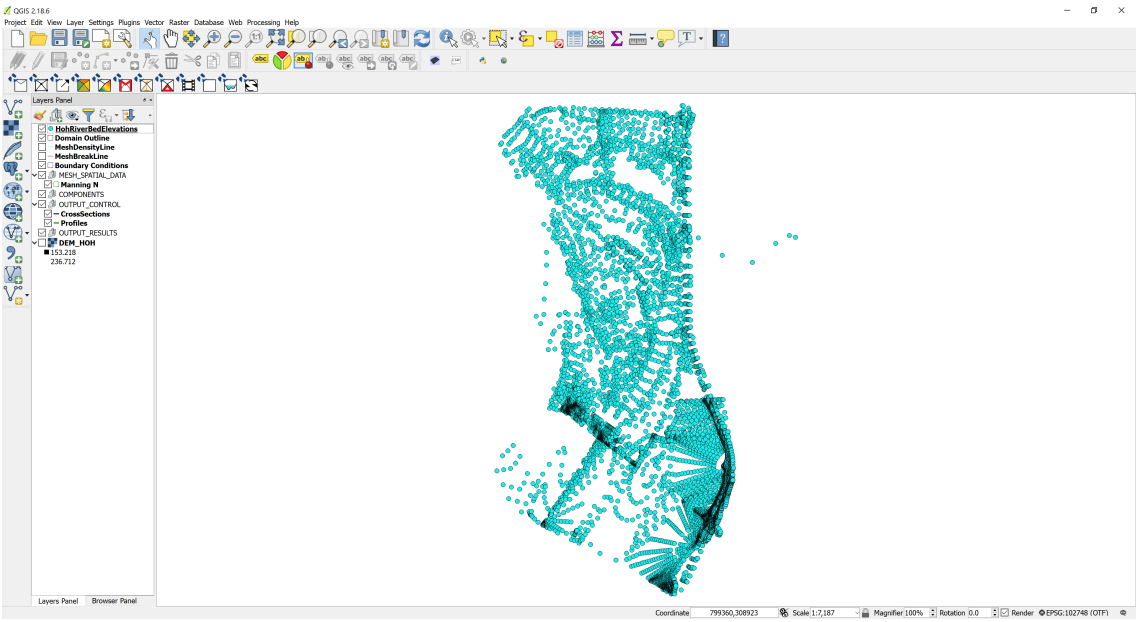


Figure 2.9 – Layer of points created from delimited data file.

6. The raster is created interpolating from the data in the point layer. For that we will use the *Interpolation* command which is found on the *Raster* menu under *Interpolation*. This command is a plugin that is installed by default in QGIS. If for any reason it does not appear, on the *Plugins* menu, click *Manage and Install Plugins...*, type *Interpolation* on the search field, and then then select the *Interpolation Plugin* checkbox to enable it.

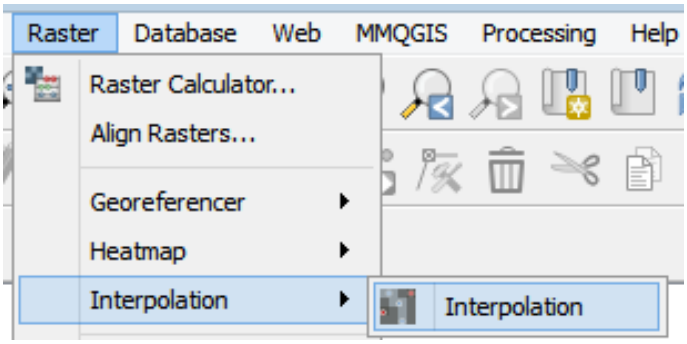


Figure 2.10 – Loading to the QGIS interpolation plugin.

When executing it, the following dialog window appears:

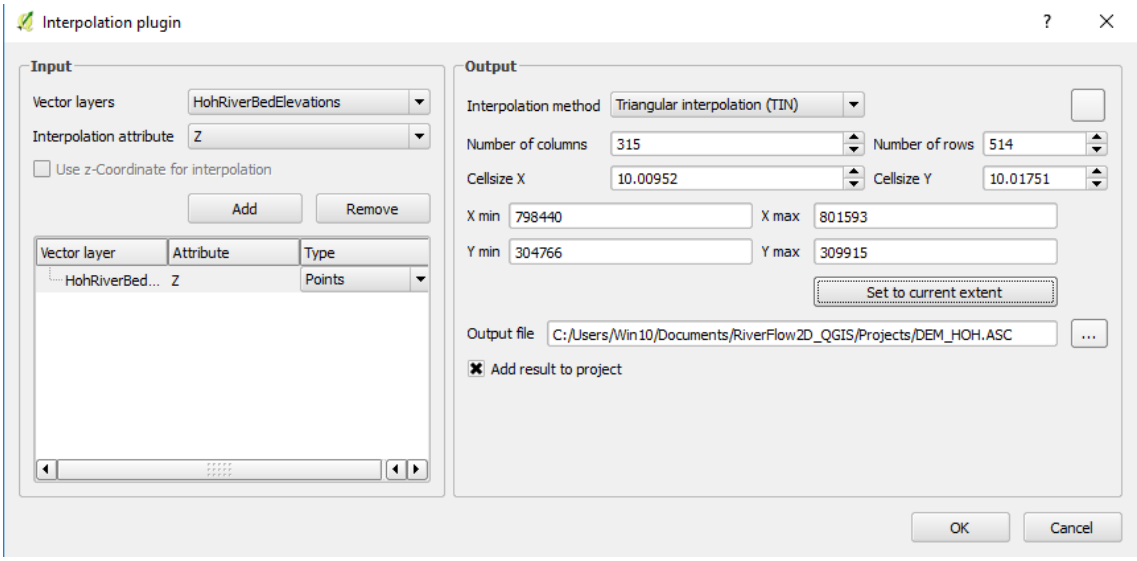


Figure 2.11 – Interpolation plugin window.

7. In this window select the layer to be interpolated to create the raster, in this example, it is the point layer that we just created *HohRiverBedElevations*. Change it by selecting the dropdown for Vector Layers. We select the field that has the attribute that was used as the value to interpolate, in this case the z field. This is done by clicking the dropdown for Interpolation Attribute, selecting Z, then clicking the Add button.

8. Once added, go to select the *interpolation method*. There are 2 options: *triangular interpolation (TIN)*, or *Inverse Distance Weighting (IDW)*. In this case the TIN method is selected, then the resolution of the raster is defined, which is fixed to cells of $10ft \times 10ft$, which generates a raster of 315 columns x 514 rows.

9. Select the path to the folder where the raster layer is to be created. It is recommended to point to the project folder directory chosen earlier in the tutorial.

10. Click OK.

Once the process is completed, the raster resulting from the interpolation will be displayed on the screen, by default it is rendered in gray gradient as shown in Figure 12.6.

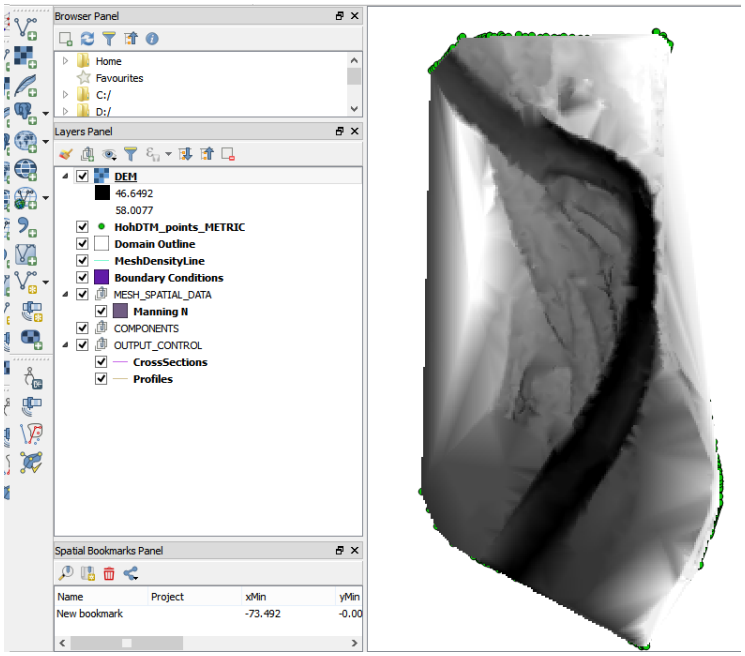


Figure 2.12 – Digital elevation model in raster format created by interpolation.

Right-clicking on the label of the created layer and selecting *Properties* allows you to change the rendering style for a more informative color palette.

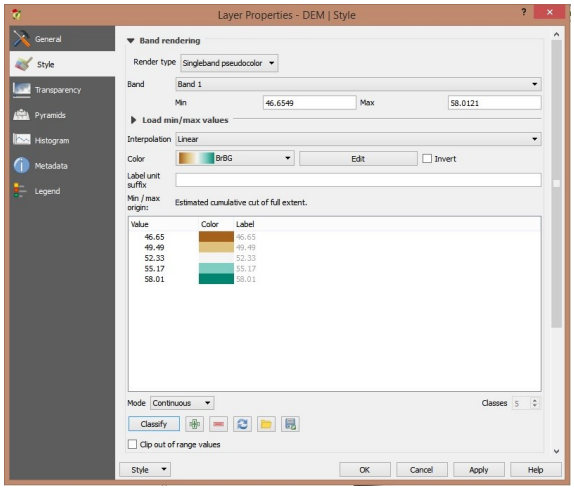


Figure 2.13 – Window to change the raster layer render style.

And now the raster layer is displayed with the new color palette selected:

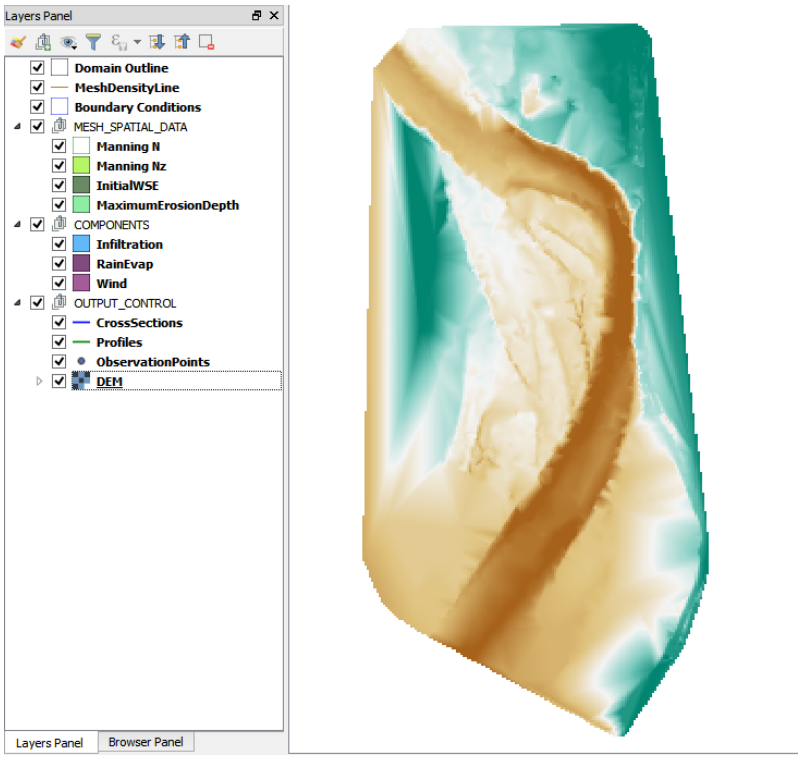


Figure 2.14 – Digital elevation model with color render.

It is convenient to move the raster layer created to the end of the list of layers, thus it does not interfere with the display of other layers.

2.1.4 Create the limits of the modeling area

The limits of the modeling area are defined using a polygon on the *Domain Outline* layer. To create it do as follows:

1. Click the *Domain Outline* layer to activate it and then click *Toggle Editing* (pencil) in the toolbar:

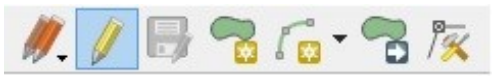


Figure 2.15 – Menu buttons for digitalization toolbar.

2. This activates the rest of the editing buttons. Now click the *Add Feature* tool which is the bean looking polygon.

Proceed to delineate the outline of the polygon by marking the vertices clicking with the left mouse button:

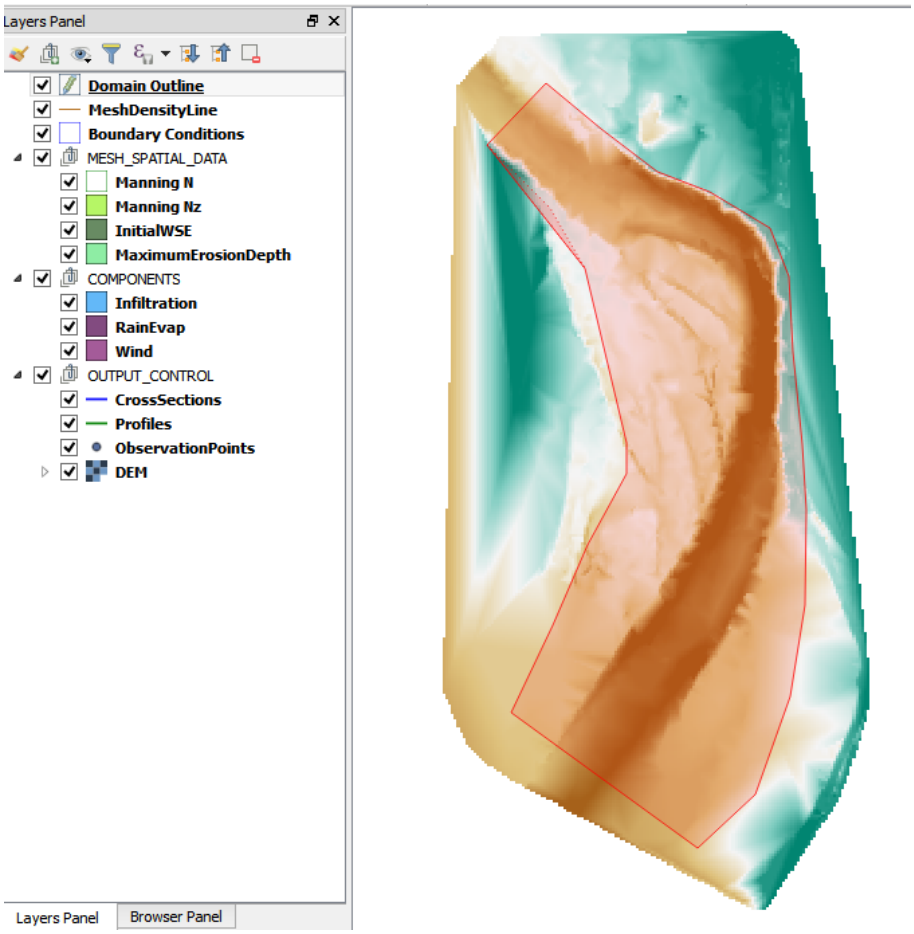


Figure 2.16 – Modeling area delimitation on the *Domain Outline* layer.

Make sure that the polygon is contained within the limits of the raster layer since HydroBID Flood will not extrapolate elevations to areas that are outside of the available data on the raster layer.

3. To finalize and close the polygon, right-click anywhere on the map view area. A dialog window to input the cell size attribute of the newly created polygon. The value for the reference size of the mesh cell is indicated. Enter a value of 100 ft.

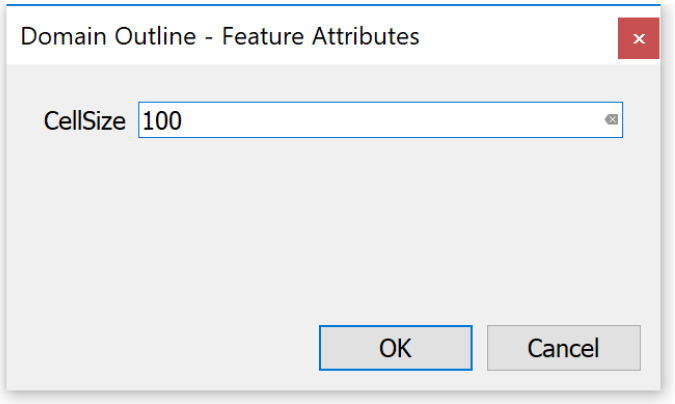


Figure 2.17 – *CellSize* defined for the *Domain Outline* layer.

If you want to make any correction in the outline of the created polygon, use the *Node* Tool:

You can move, add or remove nodes as necessary.

4. Save the polygon by clicking the Save button

and click on *Toggle Editing* button to deactivate the layer Edit mode.

The *Domain Outline* is now complete.

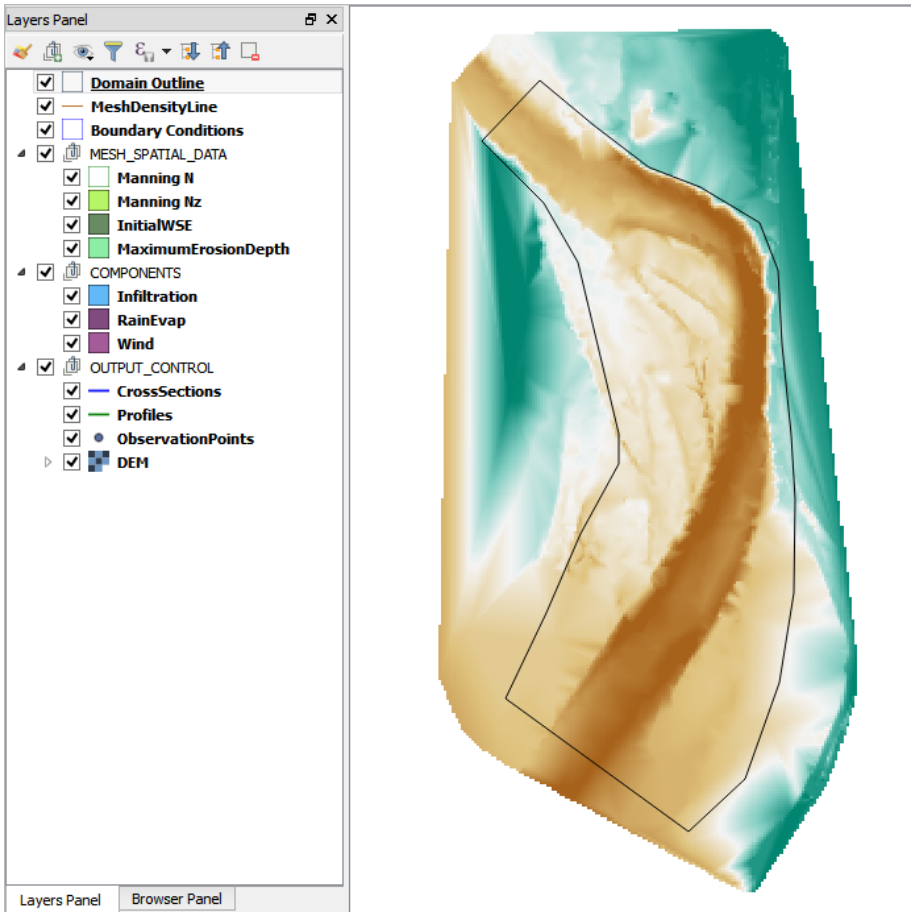


Figure 2.18 – Domain Outline layer.

2.1.5 Generating the triangular-cell mesh

Now that the *Domain Outline* layer has been created, proceed to create the mesh using the GMSH program. There are 2 plugins to access this program. For this example we will use the plugin for non-expert users which does not require the user to define any parameter.

To run the plugin, on the the *Plugins* menu, click *Generate TriMesh*, or click on the icon:

The following figure shows the generated mesh. You will also see in the Layers panel two new layers: *Trimesh* and *Trimesh_point*:

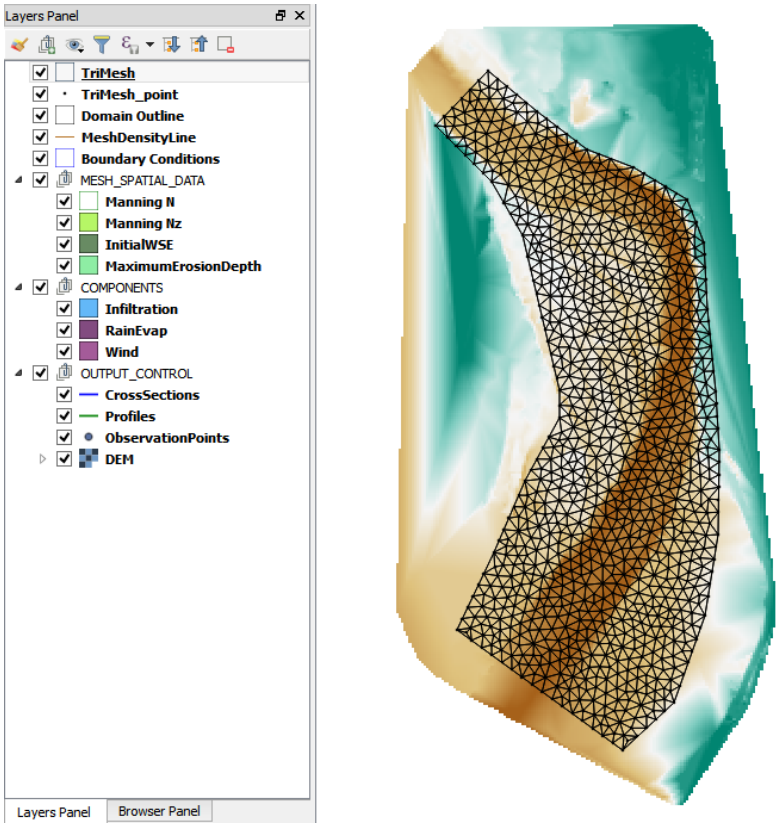


Figure 2.19 – Resulting mesh.

If you want to know the mesh generation statistics, and other messages produced by the GMSH program while creating the mesh, you can see it in the Log messages panel. This window is accessed from the *View* menu, clicking *Panels*.

Log Messages Panel

General | Plugins | Processing | Python warning | Gmsh

```
2018-09-04T11:14:31 0    Info    : Meshing curve 105 (Line)
2018-09-04T11:14:31 0    Info    : Meshing curve 106 (Line)
2018-09-04T11:14:31 0    Info    : Meshing curve 107 (Line)
2018-09-04T11:14:31 0    Info    : Meshing curve 108 (Line)
2018-09-04T11:14:31 0    Info    : Meshing curve 109 (Line)
2018-09-04T11:14:31 0    Info    : Meshing curve 110 (Line)
2018-09-04T11:14:31 0    Info    : Meshing curve 111 (Line)
2018-09-04T11:14:31 0    Info    : Meshing curve 112 (Line)
2018-09-04T11:14:31 0    Info    : Meshing curve 113 (Line)
2018-09-04T11:14:31 0    Info    : Meshing curve 114 (Line)
2018-09-04T11:14:31 0    Info    : Meshing curve 115 (Line)
2018-09-04T11:14:31 0    Info    : Meshing curve 116 (Line)
2018-09-04T11:14:31 0    Info    : Done meshing 1D (0.015625 s)
2018-09-04T11:14:31 0    Info    : Meshing 2D...
2018-09-04T11:14:31 0    Info    : Meshing surface 1 (Plane, Delaunay)
2018-09-04T11:14:31 0    Info    : Done meshing 2D (0.046875 s)
2018-09-04T11:14:31 0    Info    : 1186 vertices 2486 elements
2018-09-04T11:14:31 0    Info    : Writing 'C:\Users\Hydronia Dell\Documents\RiverFlow2D_QGIS\ExampleProjects\Hoh_QGIS\TriMesh.msh'...
2018-09-04T11:14:31 0    Info    : Done writing 'C:\Users\Hydronia Dell\Documents\RiverFlow2D_QGIS\ExampleProjects\Hoh_QGIS\TriMesh.msh'
2018-09-04T11:14:31 0    Info    : Stopped on Tue Sep 04 11:14:31 2018
2018-09-04T11:14:31 0    Gmsh has been executed correctly
```

Figure 2.20 – Message panel of the registry with GMSH messages.

2.1.6 Setting the boundary conditions

To define the boundary conditions:

Inflow boundary conditions. Draw a polygon that includes the nodes or vertices at the upper end of the mesh as indicated in the figure:

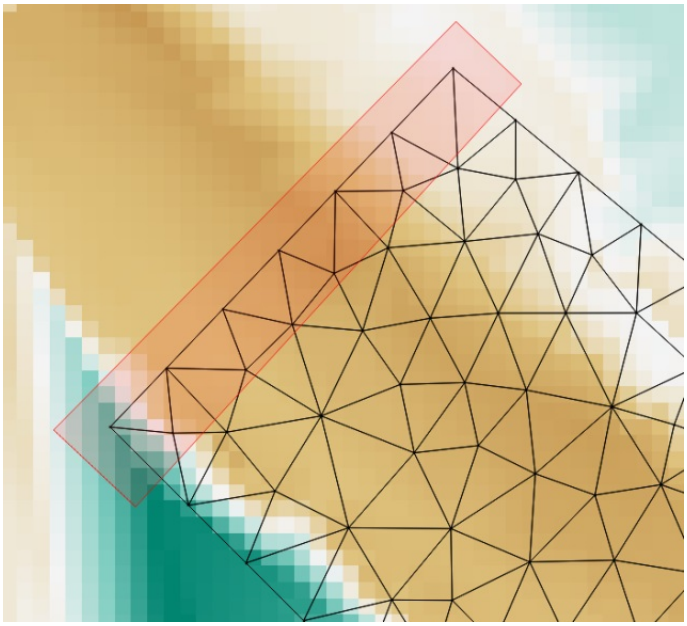


Figure 2.21 – Polygon that covers the nodes defining the Inflow boundary condition segment.

1. Select the *BoundaryConditions* layer in the Layers panel.
2. Click the Toggle Editing button to add the polygons that are going to indicate the nodes on which the inflow and outflow conditions are established.

3. To finish the polygon, right-click on desired location. A window to enter the attributes of the newly created polygon is displayed.
4. The window contains a list to select the name of ID of this BC (*Boundary Cond. ID*), if it is INFLOW or OUTFLOW condition, and a list to indicate the boundary condition Type. In the *File Conditions* border field, click the button next to it and a dialog window will open to select the file with the values of the contour condition if this is the case.

   Enter the data as shown in Figure 2.22 below:

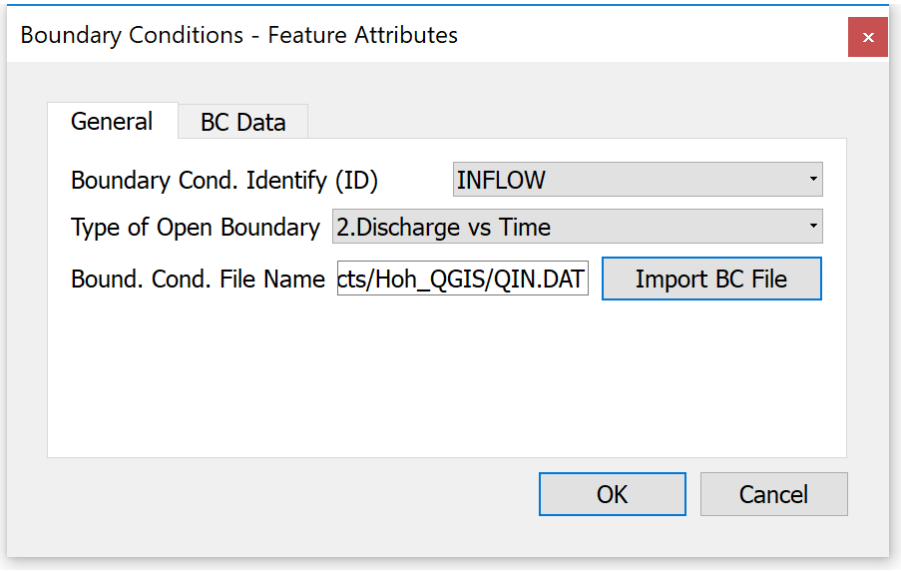


Figure 2.22 – Inflow Boundary Condition data.

5. Click the Save button:

Outflow boundary conditions. We will draw the polygon as done with the *Inflow Boundary Condition* but in the lower section of the mesh at the end of the channel. Below, the polygon that includes the nodes that indicate the outflow segment:

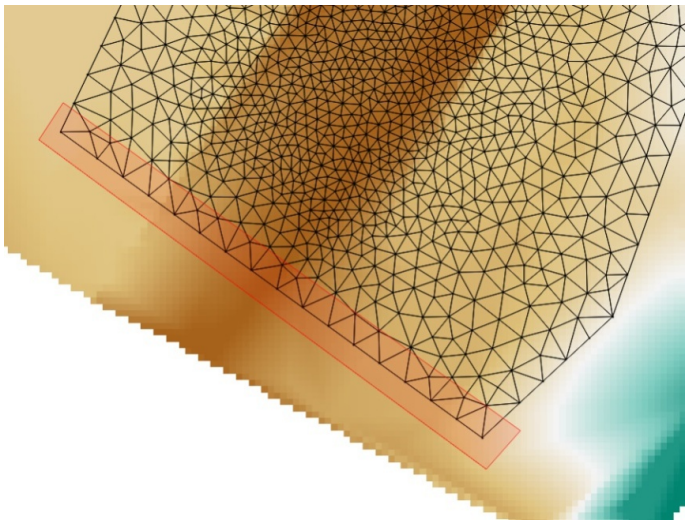


Figure 2.23 – Polygon that includes the Outflow boundary condition nodes.

1. Draw the polygon defining the Outflow Boundary condition at the end of the channel.

2. Right click to close the polygon, and a dialog window will appear to define the feature attributes. In this case, the condition type *Uniform flow conditions* is selected, so that a file is not selected, but instead the value of the slope is entered in So field. Enter the values as shown:

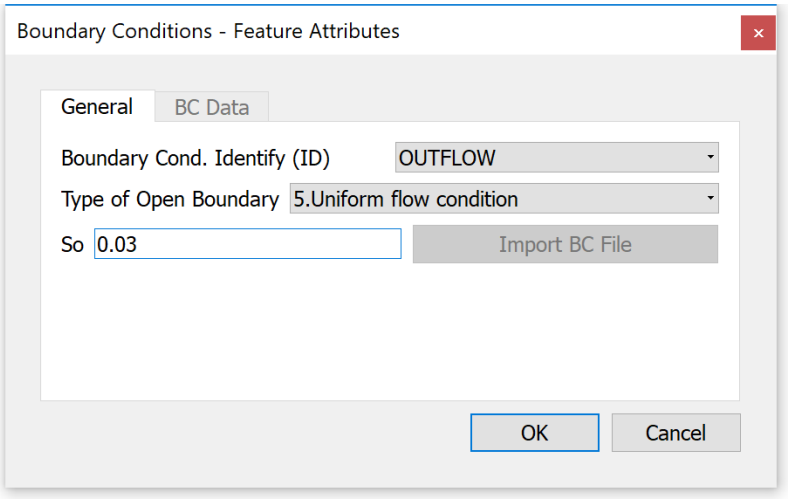


Figure 2.24 – Attributes for the polygon layer that defines the outflow boundary condition.

3. Save the changes made to the layer:

4. Deactivate editing mode by clicking on the *Toggle Editing* button

The figure below shows how the *BoundaryConditions* layer looks:

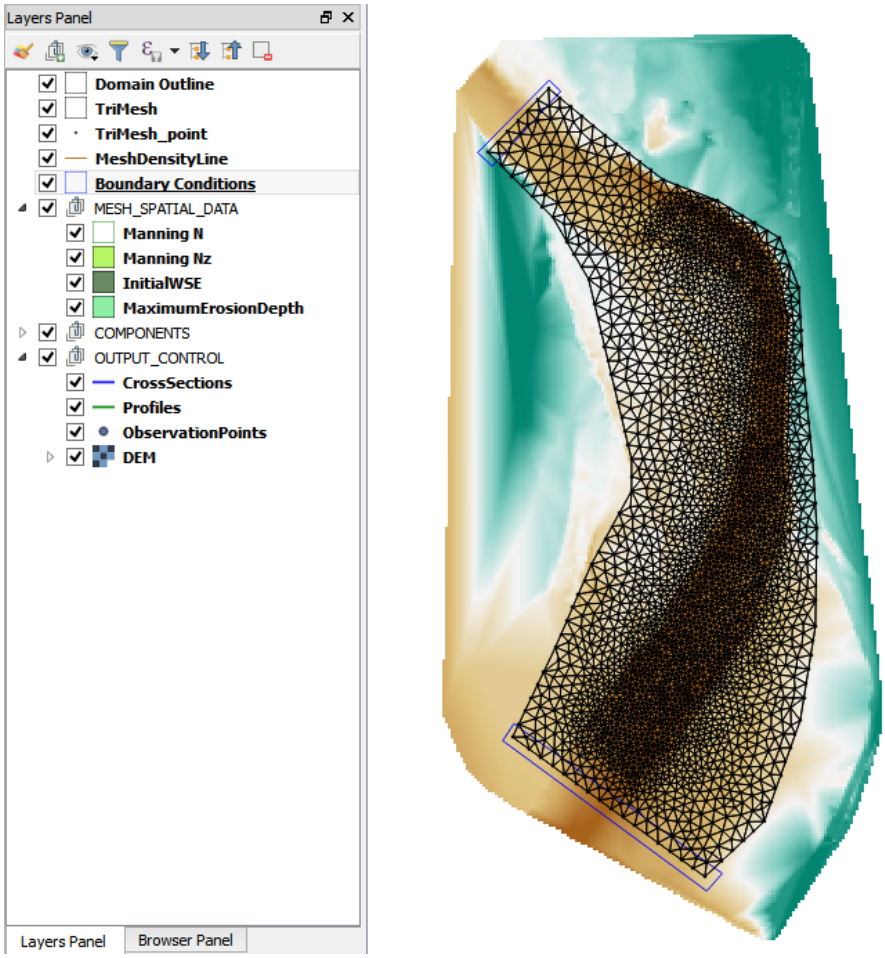


Figure 2.25 – Polygons that define the inflow and outflow boundary conditions.

2.1.7 Assigning Manning's n

To assign Manning's n values, we will enter polygons with particular n's. There can be as many polygons as those required to reproduce the spatial variability of this parameter. In this example, a single polygon will be drawn for the entire area.

1. Select the *Manning N* layer and click the *Toggle Editing* button:

2. Draw the polygon around the entire domain taking care that it covers all the cells.

   You should have an image like the one shown below:

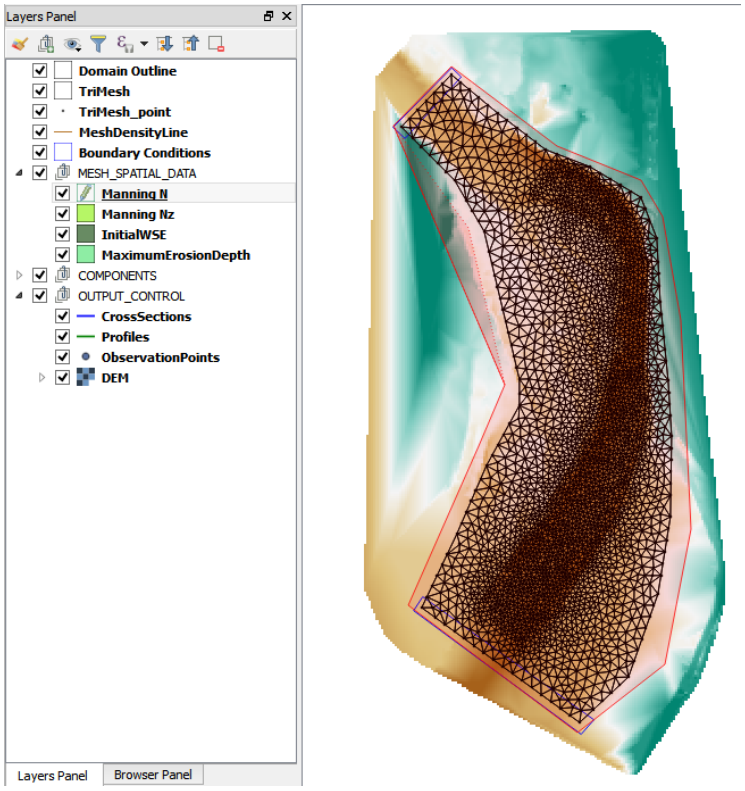


Figure 2.26 – Editing the *Manning N* layer.

3. Close the last vertices on the polygon by right-clicking on the desired position. The following dialog window is presented where you must input the Manning's n value associated to the polygon. For this case, enter 0.035:

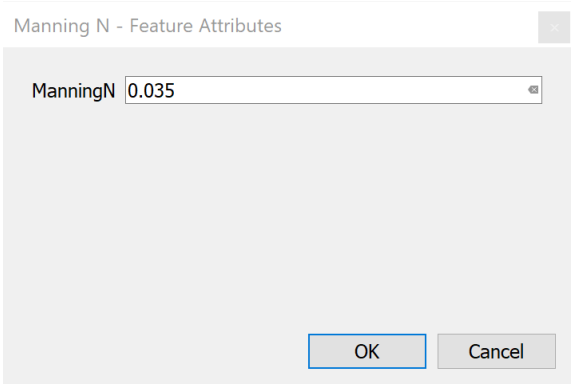


Figure 2.27 – Dialog to input *Manning N* Feature Attributes.

4. Click the Save button

and then click the Editing Tool button

to deactivate editing mode.

Before using the *Export* plugin, save the QGIS project. To accomplish this, from the *Project* menu, click *Save*. Name the project file `Hoh.qgs`.

2.2 Exporting the files

Once the layers with the input information to the model have been created, the next step is to run the second plugin, which creates the data files required by the RiverFlow2D model.

1. Run the *Export HydroBID Flood* plugin

2. A dialog window is presented. We must indicate the raster layer of the Digital Elevation Model (DEM), as this layer is not created by the plugin and its name may be different.
3. Indicate the path and file name. The path should be the same as that previously selected and the same of the `.qgs`. Also, it should not be in the sub-directory *shape* where the shape files corresponding to each layer are located.
4. Click OK.

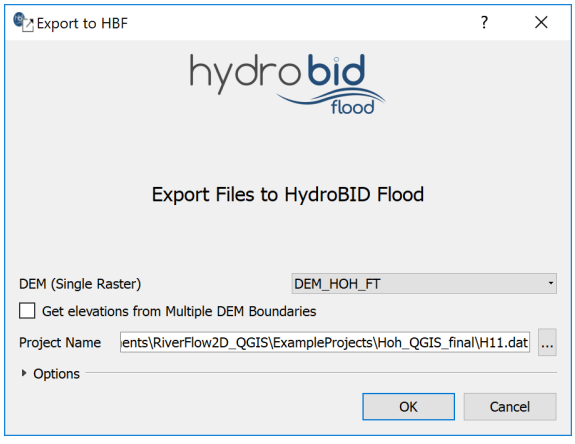


Figure 2.28 – Export HydroBID Flood dialog.

The plugin will begin to process the information, in the message bar at the top it will indicate the approximate progress of the process.

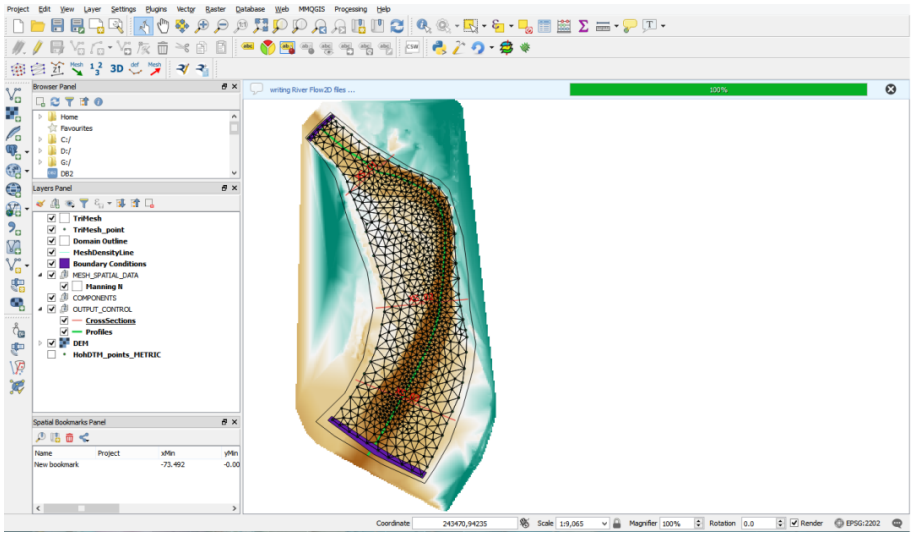


Figure 2.29 – HydroBID Flood project screen after exporting the files.

Once the process of creating the files with the input data is finished, the Hydronia Data Input Programprogram is opened automatically and a dialog window is presented with the model project to run. In this case: `Hoh.DAT` should already be set.

Then the window with the input parameters of HydroBID Flood is presented, as shown in the image below:

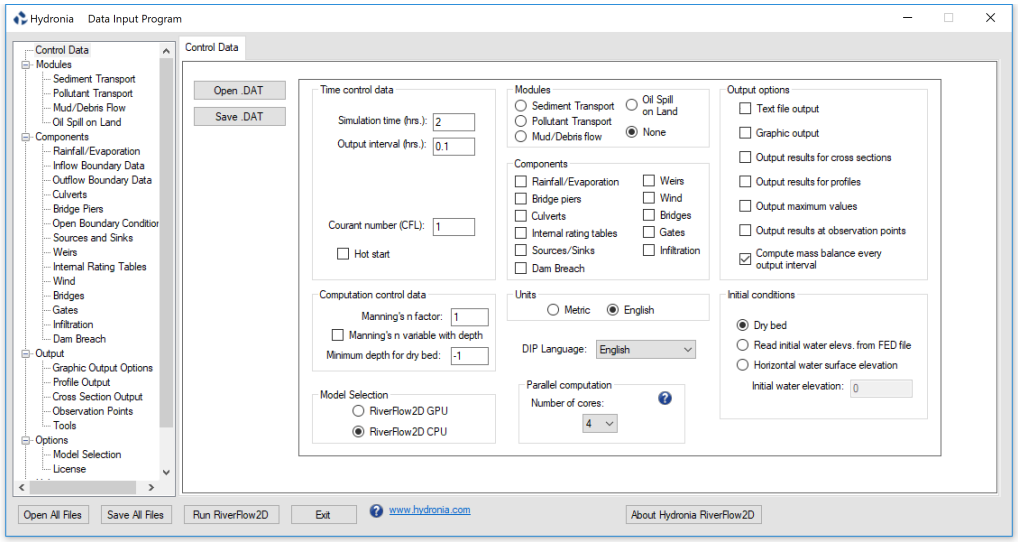


Figure 2.30 – Hydronia Data Input Program window.

5. Click the *Run RiverFlow2D* button to run the model. An image similar to the one shown below should appear:

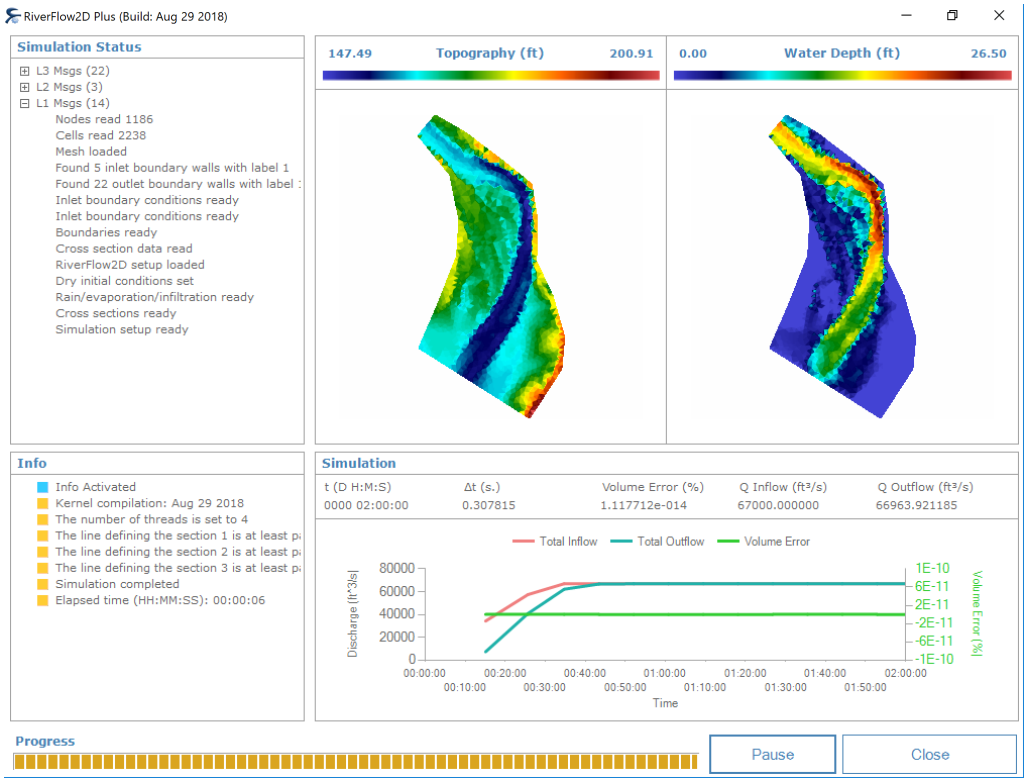


Figure 2.31 – The graph generated by HydroBID Flood.

Take some time to explore the information included in this window.

This concludes the *Creating your First HydroBID Flood Project using QGIS* tutorial.

3 — Simulating Levees using Weirs

This tutorial illustrates how to incorporate a levee to an existing HydroBID Flood project using the Weirs Component in the QGIS interface. The problem consists in modeling a lateral weir along the right margin of a river, as can be seen in the following figure.

Figure 3.1 – The weir polyline representing the levee is shown in red.

The procedure to incorporate the weir in a HydroBID Flood simulation involves the following steps:

1. Open an existing HydroBID Flood project.
2. Add *Weirs* layer
3. Draw the weir polylines
4. Input the weir parameters or attributes
5. Export the files to the HydroBID Flood
6. Run the HydroBID Flood model

7. Review output files.

The files required to follow this tutorial can be found in the following directory:

`...MyDocuments\RiverFlow2D_QGIS\ExampleProjects\WeirsTutorial`

3.1 Open an existing project

1. Open QGIS.
2. On the *Project* menu click *Open...* and browse to select the existing project: `WeirsTutorial.qgs`. This project contains the following layers: domain polygon, Digital Elevation Model (DEM) of the river bed in raster format, aerial photography, polygon with the Manning coefficient and the polygons with the boundary conditions where the flow entrance is in the upper left corner and exit in the lower left corner. The boundary conditions are a hydrograph with a peak discharge of 220,000 ft^3/s, and outflow condition is set to *uniform flow*. When you open the project you will have a project image loaded in QGIS as shown in Figure 3.2:

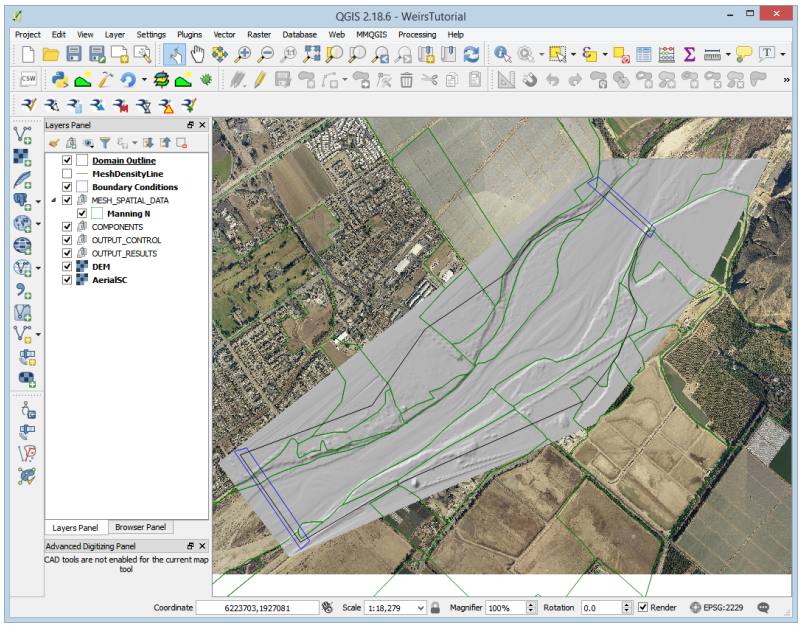


Figure 3.2 – Example of the tutorial loaded in QGIS.

3.2 Create a Weir layer and the weir polyline

The weir location can be drawn on the *Weirs* layer or it can be imported from an existing file. In this tutorial, the polyline that indicates the location of the weir will be imported from a text file. The structure of the file should be as follows:

- The first line must indicate the number n of nodes that contains the polyline.
- The *n* successive lines will contain three columns with the X, Y coordinates.
- The elevation of the weir crest separated by space as shown in Figure 3.3.

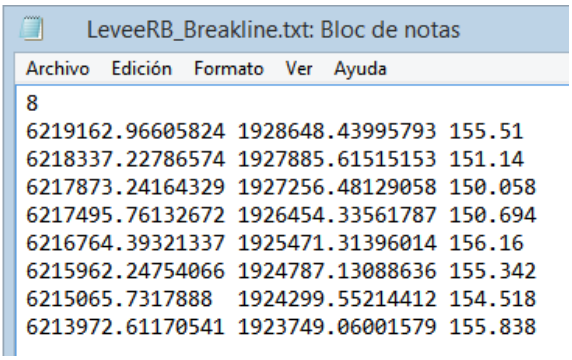


Figure 3.3 – Text file with weir alignment data.

Adding the *Weirs* layer involves the following steps:

1. Create *Weirs* layer: for this click on the *New Template Layer* button in the HydroBID Flood toolbar

2. Activate the checkbox Weirs, as shown in the Figure below:

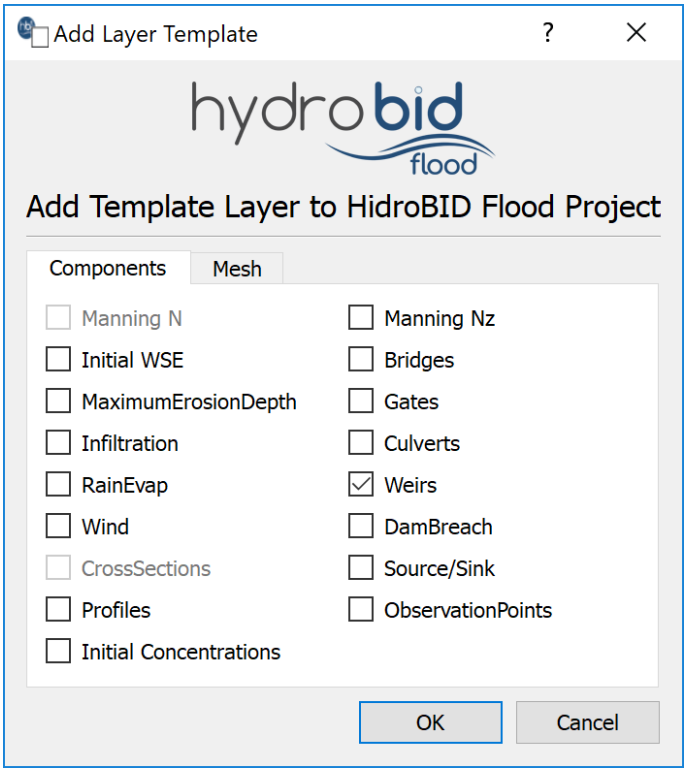


Figure 3.4 – Plugin to add a new template layer.

3. Edit the *Weirs* layer: In the layer panel, we select the *Weirs* layer and in the digitalization toolbar we click on the of the *Toggle Editing* button

A pencil icon will appear in the *Weirs* layer indicating that the layer is in edit mode:

4. Draw the line representing the weir: Using the *Add Feature* tool from the digitalization bar

Draw a line anywhere in the map area (just mark two vertices or nodes. This line will then be replaced by the coordinates of the file to be imported)

5. Right-click to finish the layout and a dialog will open to input the weir parameters.

6. Input the weir parameters: The window to input the weir attributes contains 2 tabs. In the first one there are the fields of the general parameters, within which we have:

- Weir Name (ID): Weir1
- Weir discharge coefficient: 3.2
- Weir crest elevation for all the weir: since a file will be imported this field will be left empty.
- Size Element: 150
- Import Geometry File: Click the [...] button and point to the file *LeveeRB_Breakline.txt* which is found in the tutorials folder.

The weir parameters window should look like the image shown below:

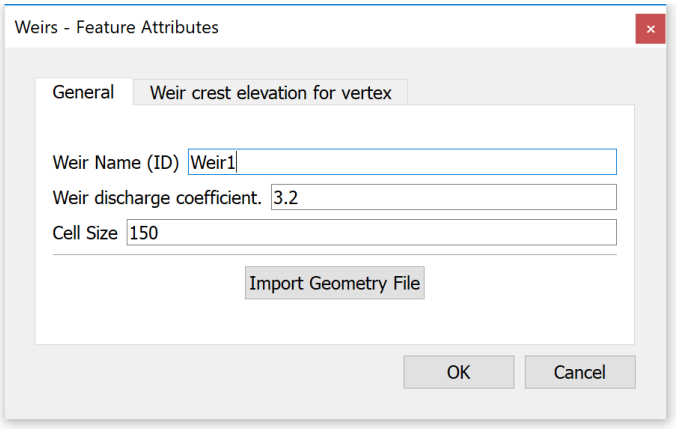


Figure 3.5 – Window to input Weir parameters.

In the second tab 'Weir crest elevation for vertex' we can see that the data contained in the file *LeveeRB_Breakline.txt* that has been loaded as shown in the following Figure:

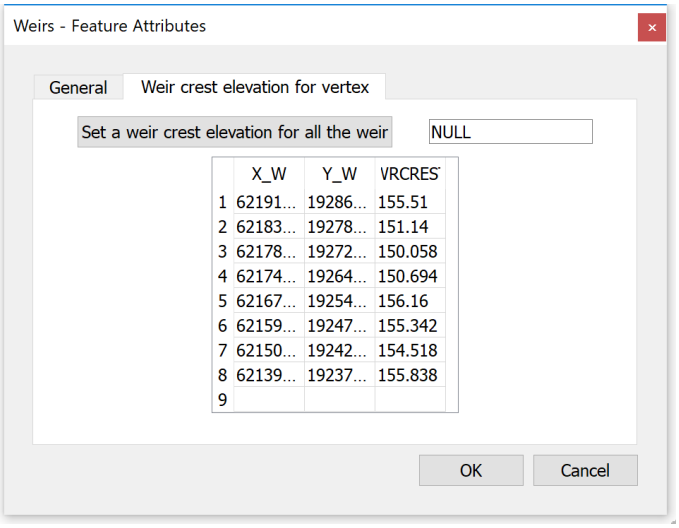


Figure 3.6 – Window for weir geometry data.

7. Then click on the [OK] button

8. Save the changes in the layer using the *Save* button of the digitalization toolbar

9. Disable the editing mode of the layer with the *Toggle Editing* button

and we will have on screen an image similar to the one shown below where you can see the layout of the weir:

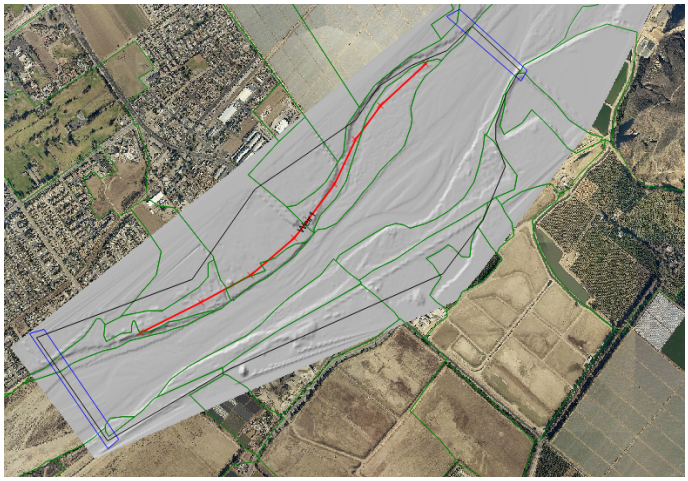


Figure 3.7 – Weir alignment loaded from the file.

3.3 Generate the mesh

The mesh is generated with the *Generate TriMesh* plugin

result should look similar to the image on Figure 3.8.

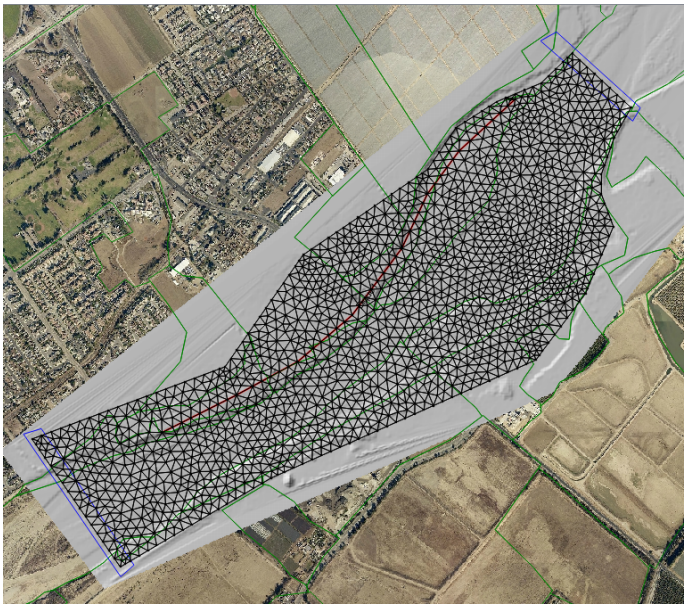


Figure 3.8 – The mesh generated.

3.4 Exporting files to RiverFlow2D

Now that you have generated the mesh and you have the other layers ready with the necessary data, you should export the files in the format required by RiverFlow2D.

1. Click the *Export to HydroBID Flood* button.

2. When running the plugin a window is displayed, here we must select the raster layer that contains the Digital Elevation Model (DEM) and the name of the project to be exported. Enter the name without any extension.

3. Before executing the plugin, activate the layer with the DEM (if it is disabled) and turn off the *MeshDensityLine* layer since it does not have any element.

   Once the plugin is executed, a window will be shown (Figure 3.9), as it should be for our example.

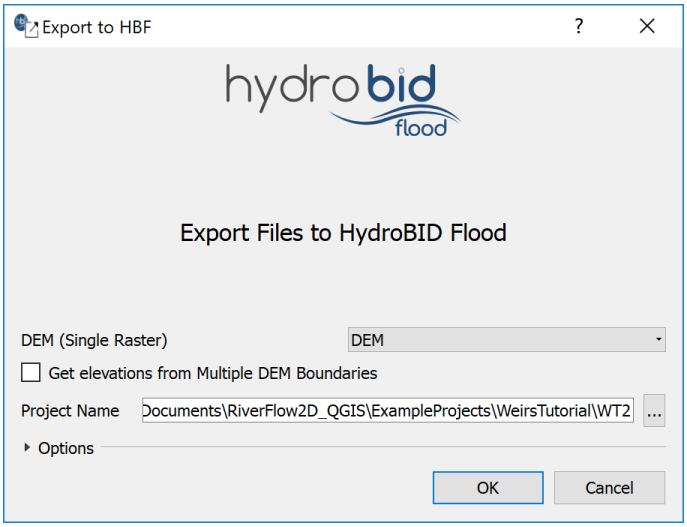


Figure 3.9 – Plugin window to export the files to RiverFlow2D.

4. After inputting the data, click on the OK button and the export process will begin. Once it is finished, Hydronia Data Input Program will be loaded as shown in Figure 3.10

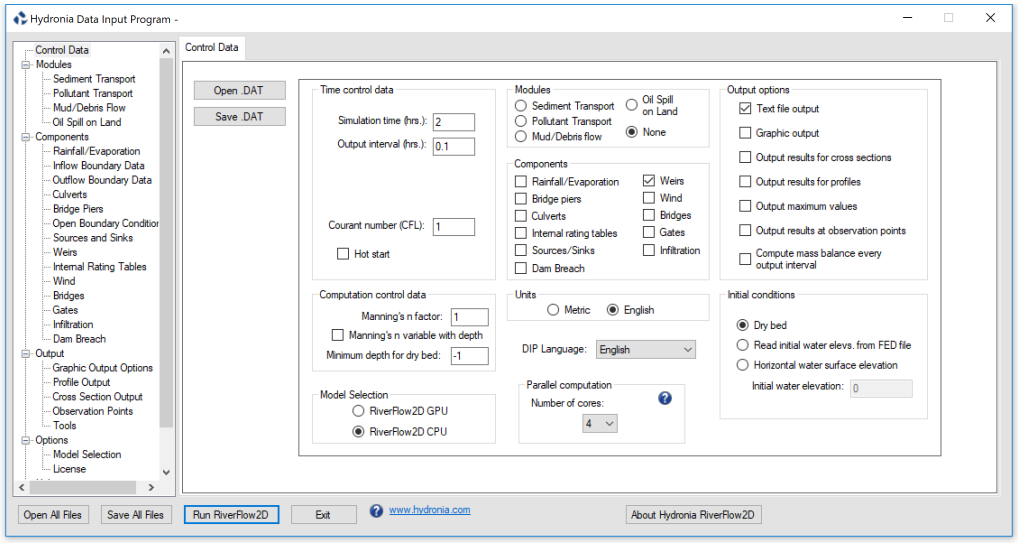


Figure 3.10 – Control data panel.

3.5 Running the model

Make sure that the Weirs Component appears selected in the Control Data panel.

1. In the list of components, select 'Weirs' and the panel of the Weirs component will appear. In this window the contents of the `.WEIRS` file content as prepared by HydroBID Flood will be displayed (Figure 3.11).

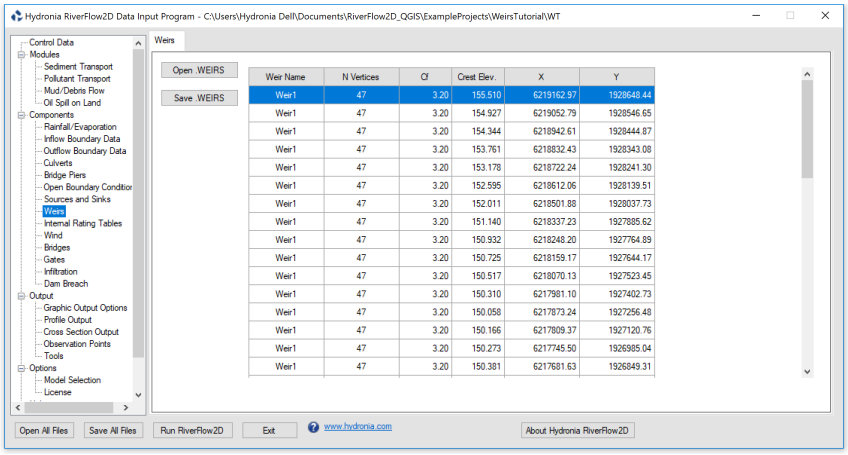


Figure 3.11 – Weirs component data panel.

2. Leave all other parameters at their default values.

3. Click on the *Run RiverFlow2D* button in the lower section of Hydronia Data Input Program. A window will appear indicating that the model began to run. The window also informs the simulation time, the volume conservation error, the total input and output discharge and other parameters as the execution progresses (Figure 3.12).

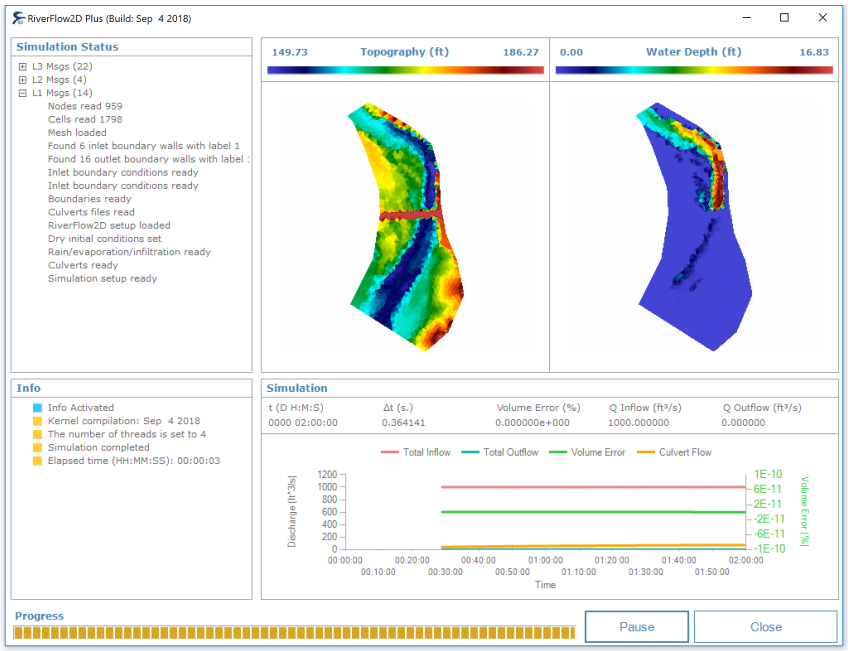


Figure 3.12 – HydroBID Flood output graphics.

3.6 Review the output files

HydroBID Flood creates an output file with the name of the project and extension `.WEIRI` for metric units and `.WEIRE` for English units. The files report results for each weir and for each output interval. Output includes the following information:

- EDGE: The segments into which the weir is divided and is given by the length of the cells in contact with the weir,
- N1 and N2: The numbers that identify the cells that share the EDGE segment in the weir,
- WSE1 and WSE2: The elevations of the water surface in the cells indicated by N1 and N2,
- D1 and D2: The depths of the flow in the cells indicated by N1 and N2,
- Distance: The length of the EDGE segment,
- Q: The discharge that passes through the EDGE segment.

The `.WEIRE` file contains the following:

```
================================================================================
                          RiverFlow2D
                          Build Aug 29 2018
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                       (C) COPYRIGHT 2009-2018 Hydronia, LLC.
                                    ALL RIGHTS RESERVED
                                    RUN DATE: 04/Sep/2018
================================================================================

 WEIR RESULTS IN ENGLISH UNITS

 TIME: 0000 days,00 hours,06 min.,00 secs.

 WEIR NO.:   1 WEIR ID: Weir1

 EDGE      N1        N2       WSE1      WSE2       D1        D2     Distance     Q
                              (ft)      (ft)      (ft)      (ft)      (ft)     (ft3/s)
   1      1757       156    130.07    131.01      4.13      5.26    150.00      0.00
   2      1707      2048    127.55    131.72      1.14      9.56    150.00      0.00
   3      1483      1189    124.85    130.96      3.57      5.71    150.00      0.00
   4      1071       163    124.69    129.79      0.28      5.24    150.00      0.00
   5       224       821    122.11    127.84      0.00      4.80    150.00      0.00
   6      1593      1130    123.22    126.83      0.00      3.70    150.00      0.00
   7      1974       904    123.22    126.00      0.00      4.70    112.08      0.00
   8        46      1861    120.05    125.32      0.00      4.84    112.08      0.00
   9      1261      1215    119.93    123.82      0.00      4.27    150.00      0.00
  10      1658       399    116.88    121.61      0.00      0.73    150.00      0.00
  11      1981       746    116.52    119.54      0.00      0.00    150.00      0.00
  12      1104      1188    118.75    117.06      0.00      0.00    150.00      0.00
  ...
  36      1681      1442    124.51    123.69      0.00      0.00    150.00      0.00
  37       402      1525    138.45    123.51      0.00      0.00    150.00      0.00
  38      2093       656    122.99    111.51      0.00      0.00    150.00      0.00
  39      1298      1840    123.98    125.30      0.00      0.00    150.00      0.00
  40      1138       808    123.42    124.66      0.00      0.00    135.26      0.00
  41       277       933    124.00    122.09      0.00      0.00    135.26      0.00
  42      1007      1831    124.82    123.91      0.00      0.00    150.00      0.00
  43      1790      1315    118.71    123.37      0.00      0.00    150.00      0.00
  44       270      1075    122.80    120.37      0.00      0.00    150.00      0.00
  45       999       102    121.86    119.07      0.00      0.00    150.00      0.00
  46       494      1108    125.75    124.11      0.00      0.00    150.00      0.00
  47       164      1355    131.84    131.44      0.00      0.00    150.00      0.00
  48      1640      2063    130.34    129.82      0.00      0.00    150.00      0.00
  49       419       855    131.81    121.48      0.00      0.00    173.91      0.00
Total discharge over weir Q =     0.000 ft3/s
```

Figure 3.13 – Extract of the output file of the Weir1.

This concludes the *Simulating Levees using Weirs* Tutorial.

4 — Simulating bridges

This tutorial illustrates how to incorporate a bridge into an existing HydroBID Flood project using the Bridge Component through the QGISinterface. The procedure involves the following steps:

1. Create the bridge geometry data.
2. Open an existing HydroBID Flood project.
3. Enter the bridge polyline.
4. Enter the bridge data.
5. Generate the mesh.
6. Export the files of RiverFlow2D.
7. Running the model.

The files required to follow this tutorial can be found in the following directory:

`...MyDocuments\RiverFlow2D_QGIS\ExampleProjects\BridgesTutorial`

4.1 Create a bridge geometry file

Integrating bridges in HydroBID Floodrequires preparing the bridge cross section geometry data prior to running the model. With the QGISinterface you have the option of creating a simplified bridge geometry from the terrain profile obtained from the digital elevation model, and then you can use the Bridges panel in the Hydronia Data Input Program to perform the adjustments necessary to the geometry generated by QGIS. This option is useful when the bridge is located on a natural section of the river, and the geometry of the bridge is simple (Figure 4.1a). However, other cases may involve more complex bridge geometries (Figure 4.1b), that require a more detailed preparation of the bridge geometry file.

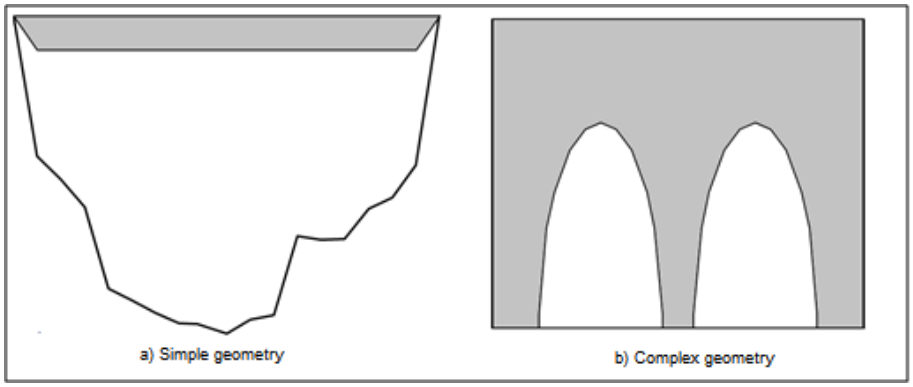


Figure 4.1 – Bridge geometries.

Figure 4.2 shows the front view of the bridge that you will to incorporate into the model for this tutorial. The data is also in the file `BRIDGEGEOMETRY.DAT` contained in the directory of this tutorial.

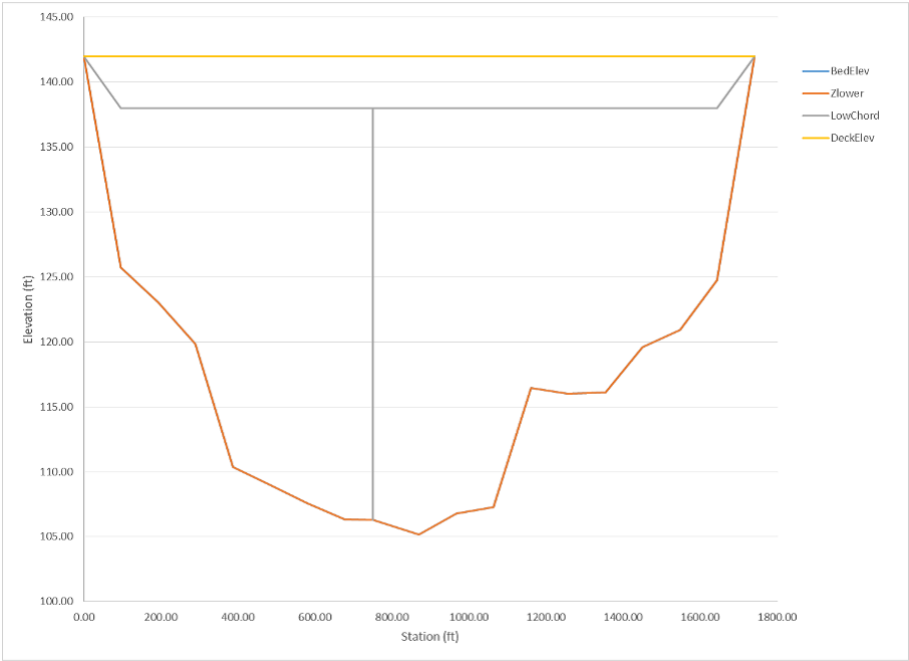


Figure 4.2 – Front view of the bridge.

It represents the cross section of a bridge with only one central pier, although this is just for the purpose of illustrating the Bridges component in this tutorial since the actual bridge in this location has about 12 sets of piers. This geometry is represented in the HydroBID Flood model using the bridge geometry file shown below, where the header row is presented only to describe the parameters, and they should not be included in the actual data file:

NP	22	X (Station)	BedElev	ZLower	LowChord	DeckElev
		0.000	142.000	142.000	142.000	142.000
		96.684	125.724	125.724	138.000	142.000
		193.367	123.034	123.034	138.000	142.000
		290.052	119.860	119.860	138.000	142.000
		386.736	110.366	110.366	138.000	142.000
		483.420	109.004	109.004	138.000	142.000
		580.103	107.578	107.578	138.000	142.000
		676.787	106.350	106.350	138.000	142.000
		750.000	106.298	106.298	138.000	142.000
		750.000	106.298	106.298	106.298	142.000
		751.000	106.298	106.298	106.298	142.000
		751.000	106.298	106.298	138.000	142.000
		870.155	105.182	105.182	138.000	142.000
		966.839	106.772	106.772	138.000	142.000
		1063.524	107.302	107.302	138.000	142.000
		1160.207	116.470	116.470	138.000	142.000
		1256.891	116.018	116.018	138.000	142.000
		1353.575	116.094	116.094	138.000	142.000
		1450.259	119.608	119.608	138.000	142.000
		1546.943	120.924	120.924	138.000	142.000
		1643.627	124.736	124.736	138.000	142.000
		1740.311	142.000	142.000	142.000	142.000

Figure 4.3 – Data of the bridge geometry.

You may use the *Bridges* panel in Hydronia Data Input Program to create to some extent, or edit a bridge geometry file (see Figure 4.4). The program lets you enter data in tabular form and view a graph of the bridge geometry. You may also manipulate the graphical lines, which will make that the tabular data be modified.

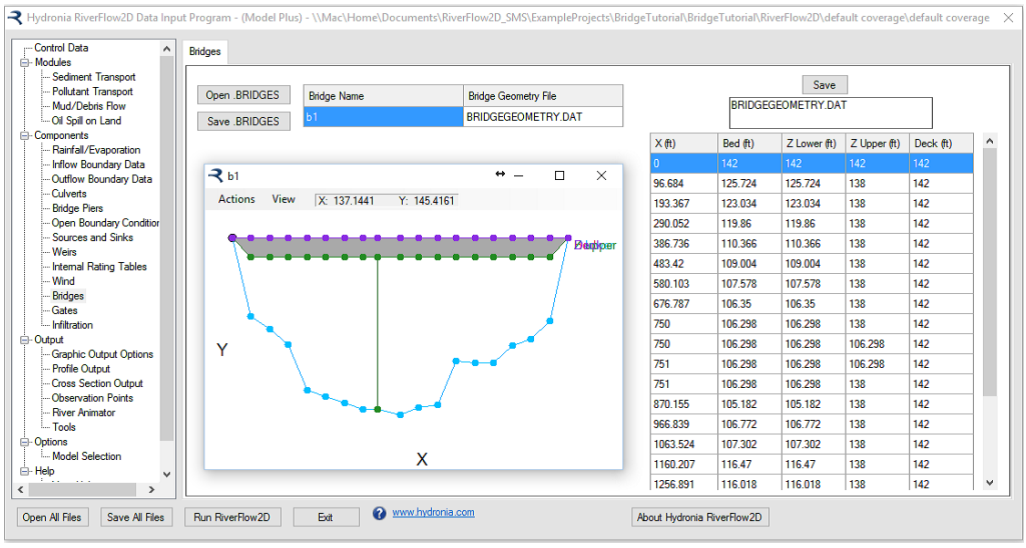


Figure 4.4 – Bridges panel.

An alternative way to create the bridge geometry file is to use a spreadsheet. In the folder for this tutorial there is a MS-Excel sheet (`BridgesGeometryPlot.xlsx`) that allows editing and plotting bridge geometry files.

4.2 Open an existing project

On the *Project* menu click *Open...* to load the existing project: `BridgeTutorial.qgs`. This project contains the layers of the domain contour, the Digital Elevation Model DEM of the river bed in raster format, the polygons with the Manning's n for the different land coverages, an aerial image, and the boundary conditions. Inflow is located in the upper right segment, and outflow in the lower left. The boundary conditions are a hydrograph with a peak discharge of 220,000 ft^3/s (cfs), and outflow condition is set to uniform flow. When you open the project you will have an image of the project loaded in QGISas shown in Figure 4.5.

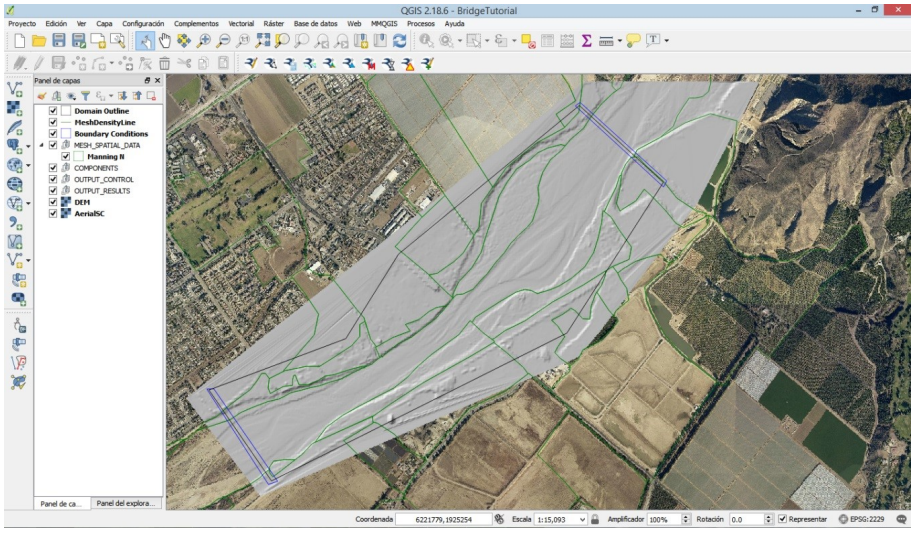


Figure 4.5 – Project screen loaded in QGIS.

4.3 Enter the bridge polyline in the *Bridges* layer

This step ensures that the mesh will conform to the bridge alignment, so that there will be nodes generated along the bridge. In this case we will enter the bridge is a straight line approximately 1740 feet long as follows:

1. Create a new *Bridges* layer: for this, go to the HydroBID Flood toolbar of and click with the mouse on the *New Template Layer* button

   in the plugin window we activate the Bridges checkBox, as shown in the Figure below

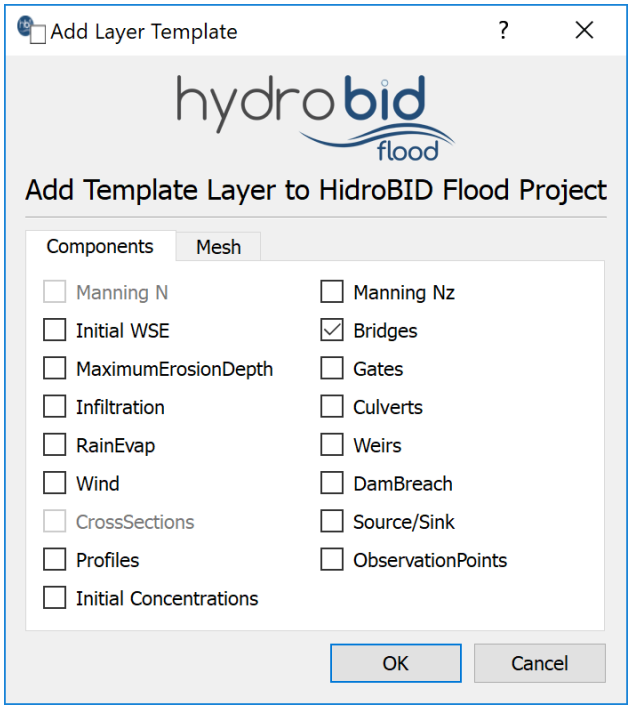


Figure 4.6 – HydroBID Flood dialog to add new layer.

2. Edit the *Bridges* layer: In the layers panel we select the *Bridges* layer then in the digitalization toolbar we click with the mouse on the *Toggle Editing* button

A pencil icon will appear in the *Bridges* layer that tells us that the layer is in edit mode:

3. Draw the line representing the bridge: (if necessary, turn off the DEM layer so that it does not interfere with the identification of the bridge site in the aerial photograph).

4. Using the *Add Feature* button of the digitalization toolbar

draw the line indicating the location of the bridge. In the case shown, to demarcate the line that indicates the location of the bridge, it is only necessary to indicate two vertices (initial and final), then click with the right button to finish the drawing and we will have an image similar to the one shown in the following figure:

Figure 4.7 – Bridge drawing.

5. Enter the bridge data: After the bridge layout is finished, the window to input the attributes of the bridge created is immediately displayed, these are:

   - Bridge Name (ID): Bridge1
   - Size Element: 150 feet
   - Bridge file: Bridge1.txt (In this case, we will let QGIS build the bridge file, so we only indicate the name of the file)
   - Elevation of the lower bridge deck (LOWCHORD): 138
   - Elevation of the bridge deck (DECKELEV): 142

   The figure below shows the attributes window of the *Bridges* layer:

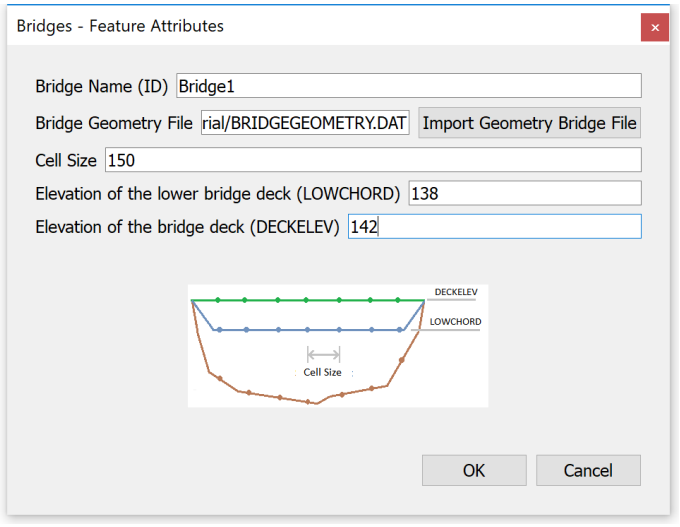


Figure 4.8 – Attribute dialog window of the *Bridges* layer.

6. After entering the values, click on the [OK] button, save the changes in the layer using the *save* button of the digitalization toolbar

and disable the editing mode of the layer with the *Toggle Editing*

4.4 Generate the mesh

The mesh is generated with using the HydroBID Flood emphGenerate TriMesh icon

check that the resulting mesh is perfectly aligned with the bridge as shown in Figure 4.9.

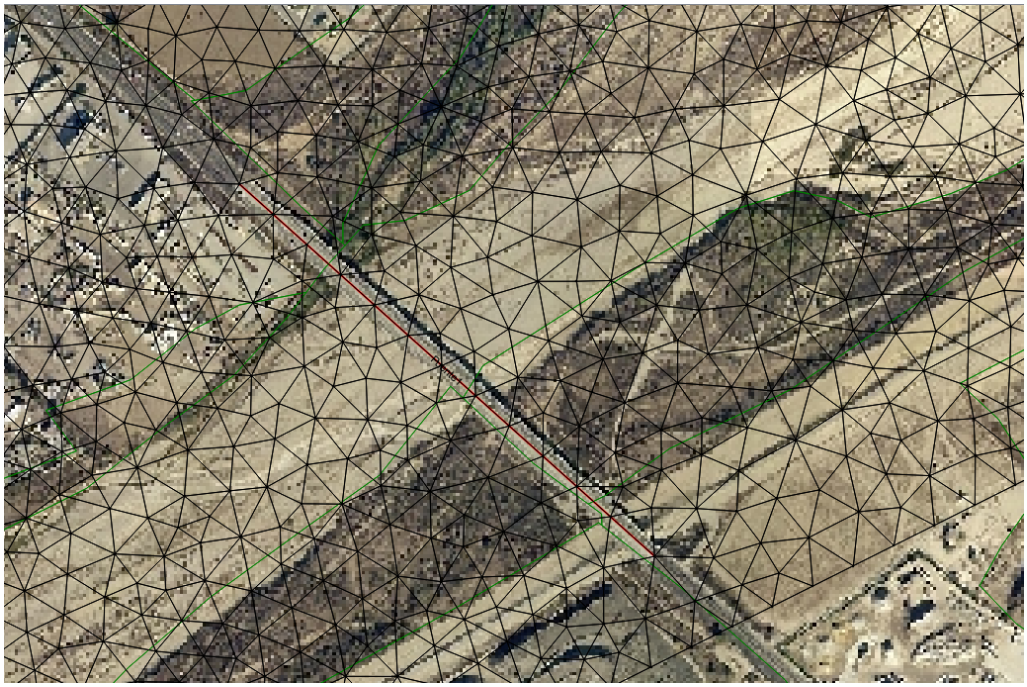


Figure 4.9 – Mesh aligned with bridge polyline.

4.5 Exporting files to HydroBID Flood

Now that you have generated the mesh, and you have ready the other layers with the necessary data, you should export the files in the format required by HydroBID Flood.

1. Click on *Export HydroBID Flood* button:

   In the export files dialog we need to make sure that the appropriate raster layer corresponding to the Digital Elevation Model (DEM) is selected. Also enter the name of the project; only the name must be indicated without any extension.

   Once the complement is executed, a window is presented as shown in Figure 4.10, as it should be for our example.

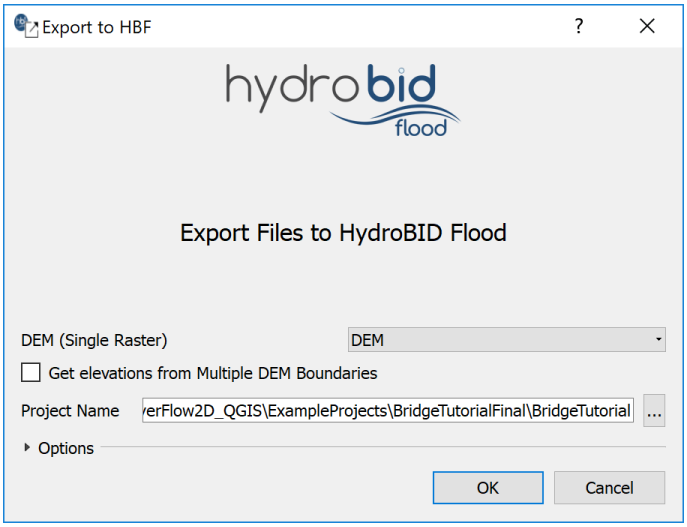


Figure 4.10 – Export HydroBID Flood tool.

2. After input the information, click on the OK button and the export process will begin.

   Once finished, the RiverFlow2D program will be loaded with the `.DAT` file of the specific example.

4.6 Running the Model

After exporting the files, the HydroBID Flood program is loaded with the project file of the `bridge.DAT` example, and the *Control Data* panel is shown (Figure 4.11).

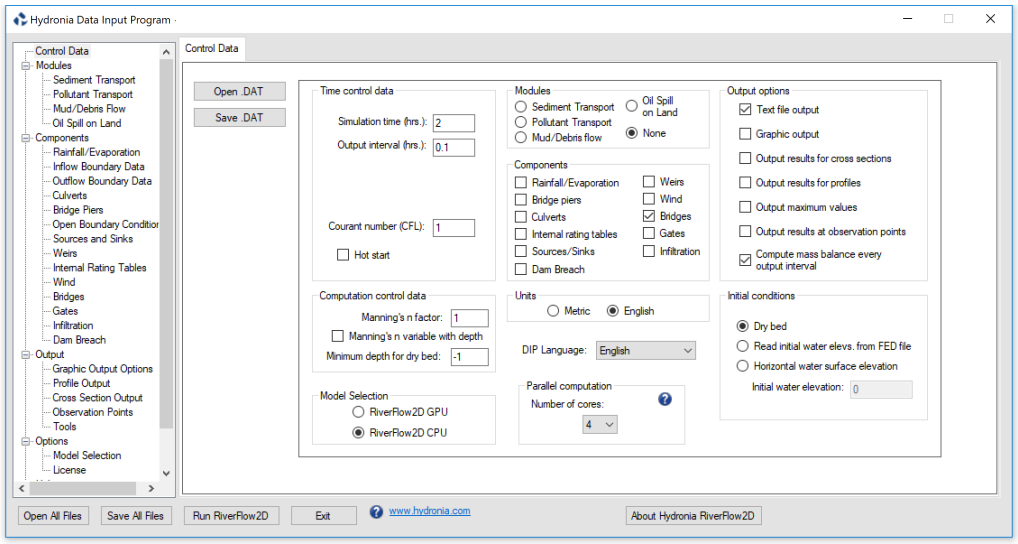


Figure 4.11 – Control Data panel.

Select the Bridges panel to review the contents of the bridge geometry file (Figure 4.12). Note

how the bridge profile was discretized every 150 feet according to the element size imposed to the bridge line.

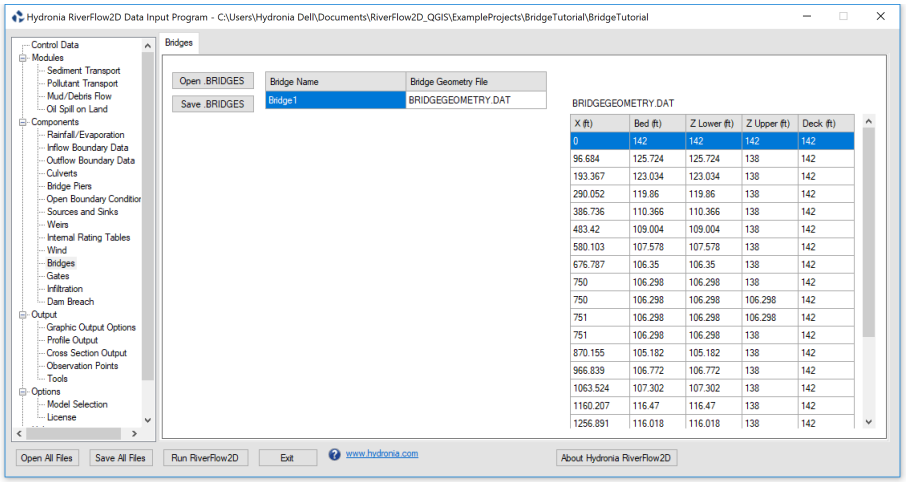


Figure 4.12 – Bridges component data panel.

Leave all other parameters at their default values.

To run the model, click on the Run HydroBID Flood button in the lower section of Hydronia Data Input Program. A window will appear indicating that the model run started. The window also reports the simulation time, the volume conservation error, the total input and output discharge, and other parameters as the execution progresses (Figure 4.13).

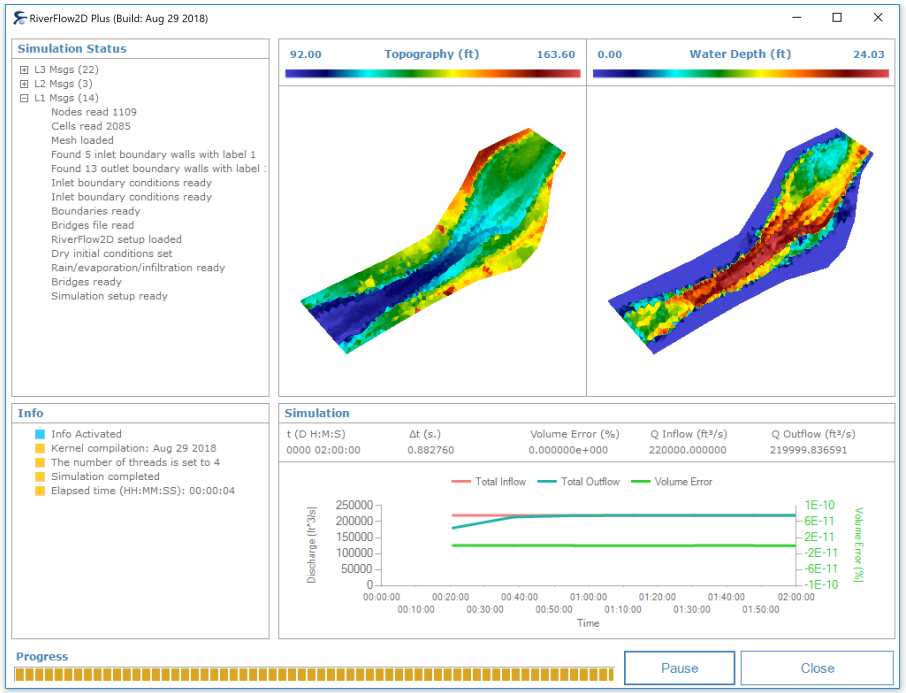


Figure 4.13 – RiverFlow2D output graphics.

You may take some time to review the `BridgeTutorial.bridgeh` file:

```
======================================================================
======================================================================
                          RiverFlow2D
                          Build Aug 29 2018
======================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                       (C) COPYRIGHT 2009-2018 Hydronia, LLC.
                                       ALL RIGHTS RESERVED
                                       RUN DATE: 04/Sep/2018
======================================================================

 Hydrograph for bridges in cfs, time in hours

  Time        Bridge1

0.000000       0.000
0.002385       0.000
0.100000       0.000
0.200000  167458.624
0.300000  193509.056
0.400000  204219.632
0.500000  211604.297
0.600000  214603.543
0.700000  216401.046
0.800000  217273.535
0.900000  217646.590
1.000000  217936.200
1.100000  218088.732
1.200000  218165.809
1.300000  218207.101
1.400000  218228.820
1.500000  218239.789
1.600000  218244.150
1.700000  218247.920
1.800000  218249.237
1.900000  218249.603
2.000000  218250.306
```

Figure 4.14 – Bridge hydrograph file `.bridgeh`.

This concludes the *Simulating bridges* tutorial.

5 — Simulating culverts

This tutorial shows how to incorporate culverts in an existing HydroBID Flood project using the QGIS interface. The problem consists of a natural channel crossed by a road embankment. A culvert structure is to be used to connect the upstream and downstream parts of the channel divided by the embankment as shown in the following figure:

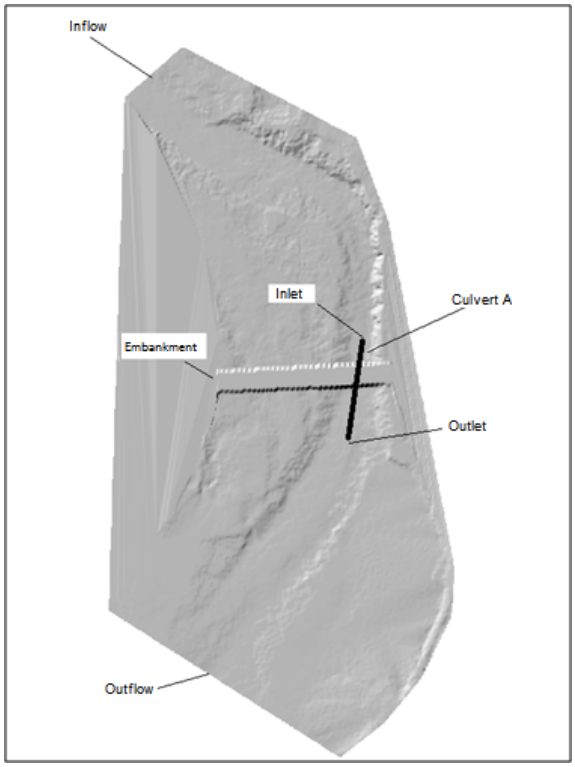


Figure 5.1 – Culvert scheme.

The water enters from upstream with a constant discharge of 1000 cfs, and outflows downstream along the indicated section. The area is initially dry. The culvert has a circular cross section, and other characteristics as summarized in the following Table (CulvertA). CulvertB data is provided in case that you wanted to extend the tutorial adding a second culvert to the project.

Parameter	Description	Culvert A	Culvert B
Nb	Number of identical barrels	1	1
Ke	Entrance Loss Coefficients	0.5	0.7
n	Mannings n Coefficient	0.014	0.015
Kp	Inlet Control Coefficient	0.3	0.4
M	Inlet Control Coefficient	2	2
Cp	Inlet Control Coefficient	1.28	1.1
Y	Inlet Control Coefficient	0.67	0.69
m	Inlet Control Coefficient	-0.5	-0.5
Dc	Diameter (feet)	3	2
-	Inlet Inverted elevation	-9999	-9999
-	Outlet Inverted elevation	-9999	-9999

The procedure to integrate this culvert into a HydroBID Flood simulation involves the following steps:

1. Open an existing HydroBID Flood project.
2. Add a Culvert component layer.
3. Draw the line of culvert alignment.
4. Input the data or attributes for the culvert.
5. Export the files to the HydroBID Flood program.
6. Running the model.
7. Review culvert output files.

The files required to follow this tutorial can be found in the following directory:

`...MyDocuments\RiverFlow2D_QGIS\ExampleProjects\CulvertTutorial`

5.1 Open an existing project

1. Open QGIS.
2. On the *Project* menu click *Open...* and browse to the existing project: `CulvertTutorial.qgs`.

This project contains the following layers: domain outline, Digital Elevation Model (DEM) in raster format, polygon with the Manning's n coefficient, and the boundary condition polygons. The inflow is located in the upper left, and the outflow in the lower left. The boundary conditions corresponds to a constant discharge of 1000 ft^3/s, and outflow conditions is set to free. Figure 5.2 shows the opened project.

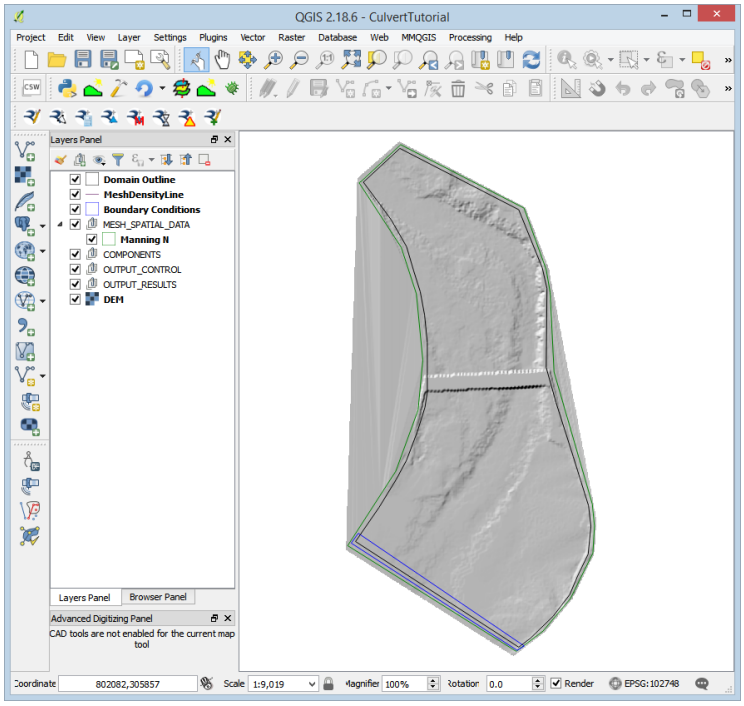


Figure 5.2 – Example of the tutorial loaded in QGIS.

5.2 Create Culverts layer and draw the Culvert

1. To create the *Culverts* layer, in the HydroBID Flood toolbar click on the *New Template Layer* icon

2. In the window select the Culverts checkBox, as shown:

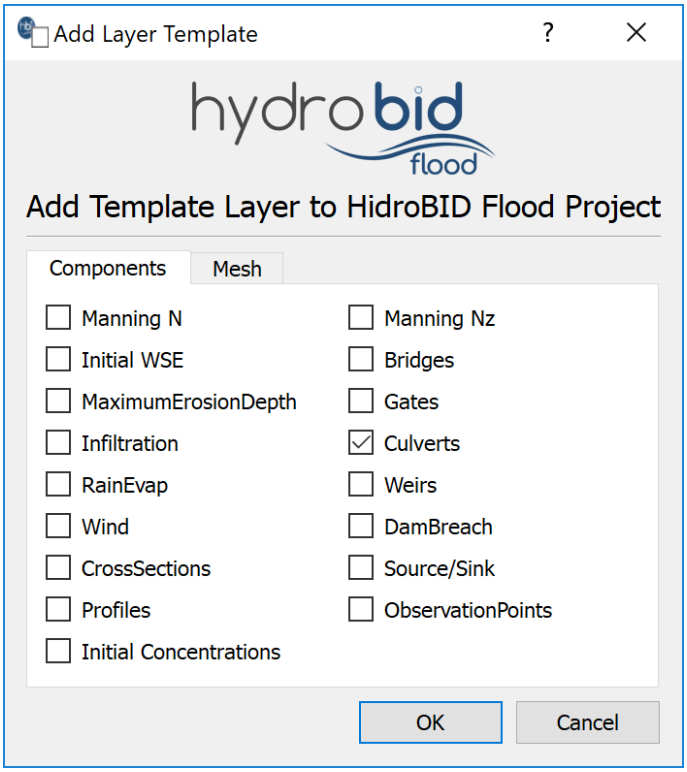


Figure 5.3 – Plugin to add a New Template Layer.

3. Edit the *Culverts* layer: In the layers panel, select the *Culverts* layer and in the digitalization toolbar we click on the *Toggle Editing* button

A pencil icon will appear in the *Culverts* layer indicating that the layer is in edit mode:

4. Draw the line representing the culvert alignment: Using the tool *Add Feature* from the digitalization toolbar

draw the line that represents the culvert. It is only necessary to indicate two vertices.

5. Click with the right button to finish, and you should get an image similar to the one shown in the following Figure:

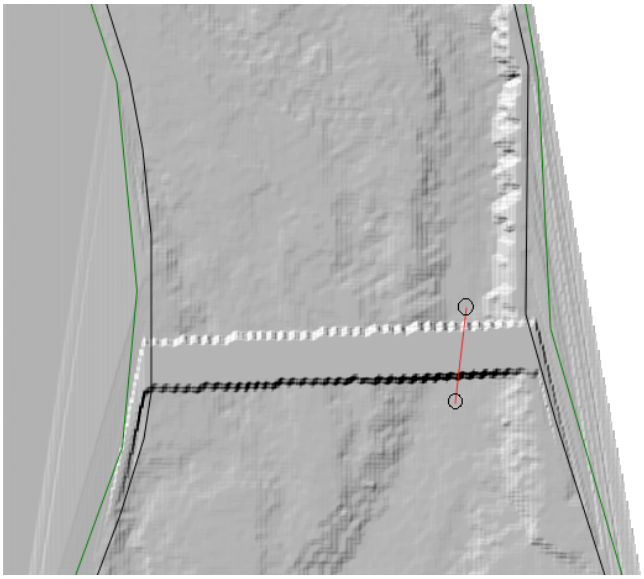


Figure 5.4 – Culvert alignment.

6. Enter the culvert data: After the culvert drawing is finished, the window to input the culvert attributes immediately appears. The dialog window has 2 tabs, in the General tab you enter the basic data for circular and box culverts:

   - Culvert Name (ID): Culv1,
   - Type of culvert: Type 2 (Circular culvert)
   - Culvert File: Culv1.txt
   - The rest of the parameters are coefficients that needed to compute the culvert discharge. To help with the introduction of these parameters, the window presents a list with default values for different types of culverts. If one of the values of the list parameters is not appropriate, you can choose the option where the value is defined by the user (user defined). The window of the culvert parameters should be similar to the one shown in the Figure below:

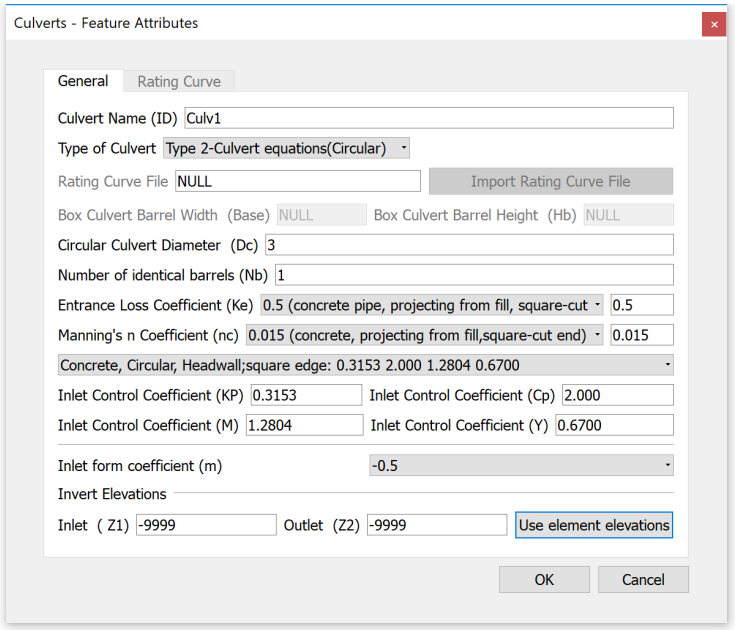


Figure 5.5 – Window to input Culvert parameters.

7. After inputting the values, click on the *OK* button.
8. Save the changes in the layer using the *Save* button of the digitalization toolbar

9. Disable the editing mode of the layer with the *Toggle Editing* button

5.3 Generate the mesh

The mesh is generated with the *Generate Trimesh* tool

the results obtained as shown in Figure 5.6.

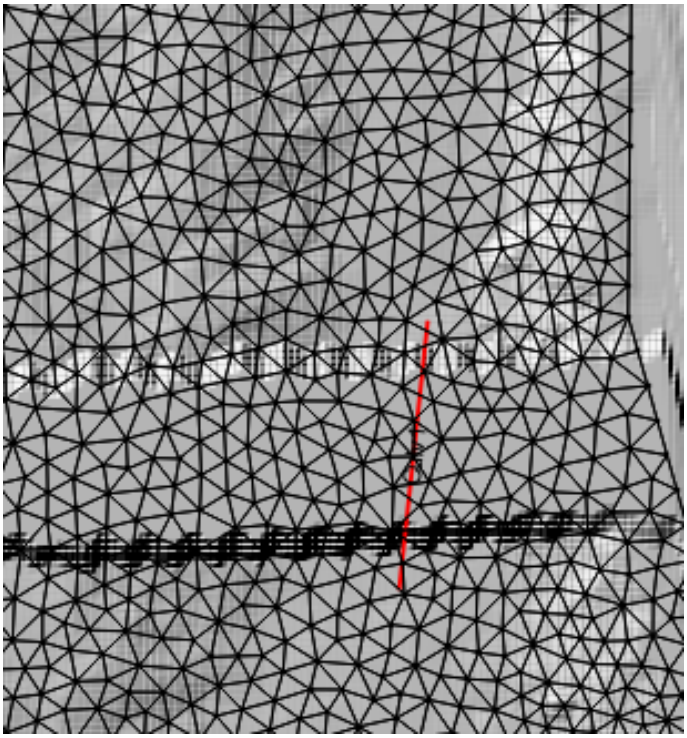


Figure 5.6 – Final mesh.

5.4 Exporting files to HydroBID Flood

Now that you have generated the mesh and you have the other layers ready with the necessary data, you should export the files in the format required by HydroBID Flood.

1. Click the *Export HydroBID Flood* button

2. Select the raster layer that contains the Digital Elevation Model (DEM) and the name of the project to be exported. Indicate a name without any extension, in this case, *CulvTutorial*.
3. Before running the plugin, activate the layer with the DEM (if it is disabled) and deactivate the MeshDensityLine layer, since it does not have any element.

Once the plugin is executed, a window will be shown (Figure 5.7), as it should be for our example.

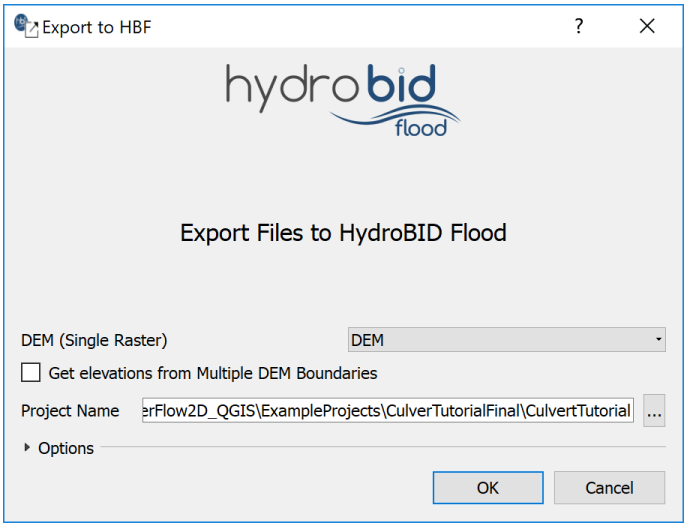


Figure 5.7 – Plugin window to export the files.

After input the information, click on the OK button and the export process will begin, once finished, the HydroBID Flood program will be loaded with the `CulvTutorial.DAT` file.

5.5 Running the model

After exporting the files, the HydroBID Flood program is loaded with the project file from the `culverts.DAT` example and shows the *Control Data* panel to it as illustrated in Figure 5.8.

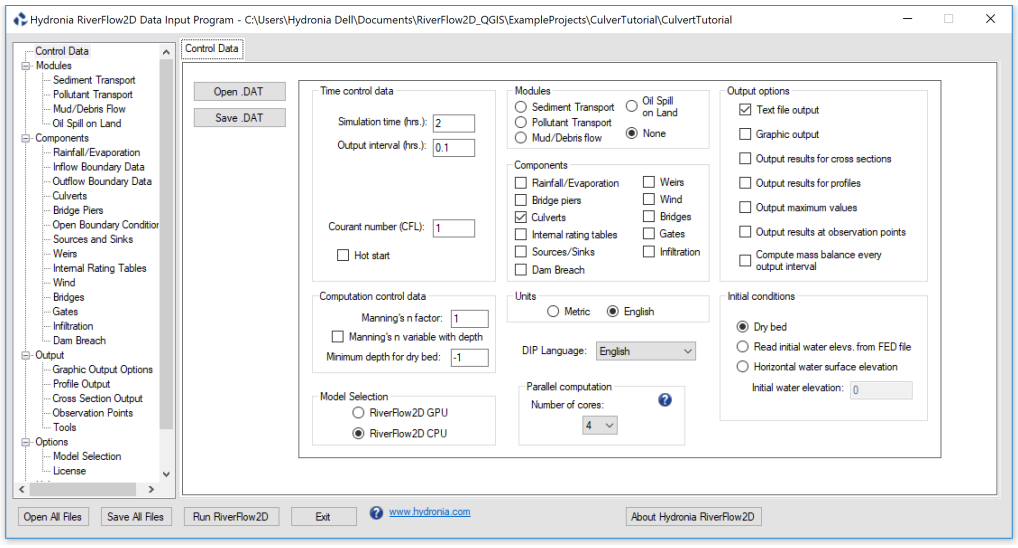


Figure 5.8 – Control data panel.

Note that the Culverts Component appears selected. On the left side of the *Control Data* panel, in the list of components select *Culverts* to activate the *Culverts* panel. The contents of the

culvert file prepared by QGIS will be displayed (Figure 4.10).

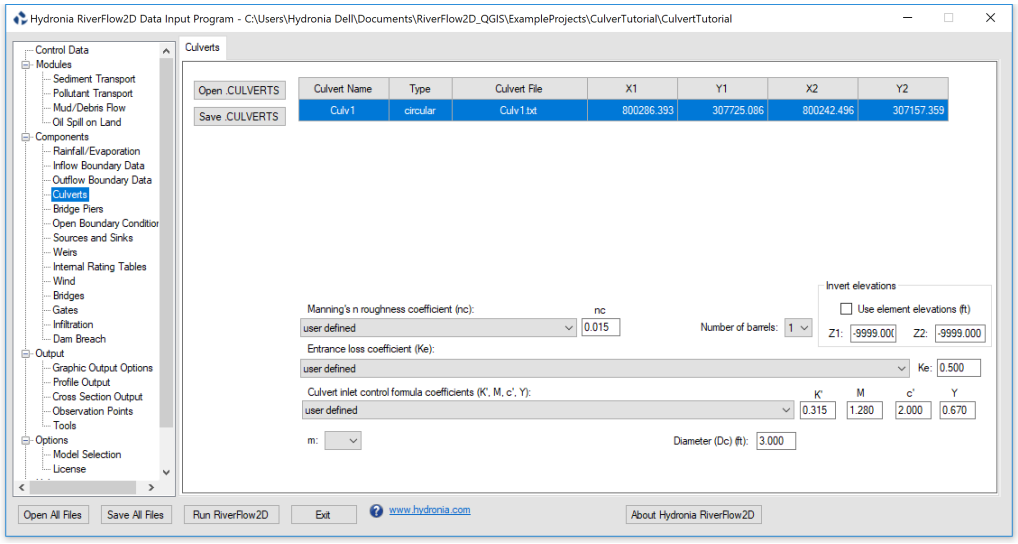


Figure 5.9 – Culverts component data panel.

Leave all other parameters at their default values.

To run the model, click on the *Run RiverFlow2D* button in the lower section of Hydronia Data Input Program. A window will appear indicating that the model began to run. The window also reports the simulation time, volume conservation error, total input and output discharge, and other parameters as the execution progresses (Figure 5.10).

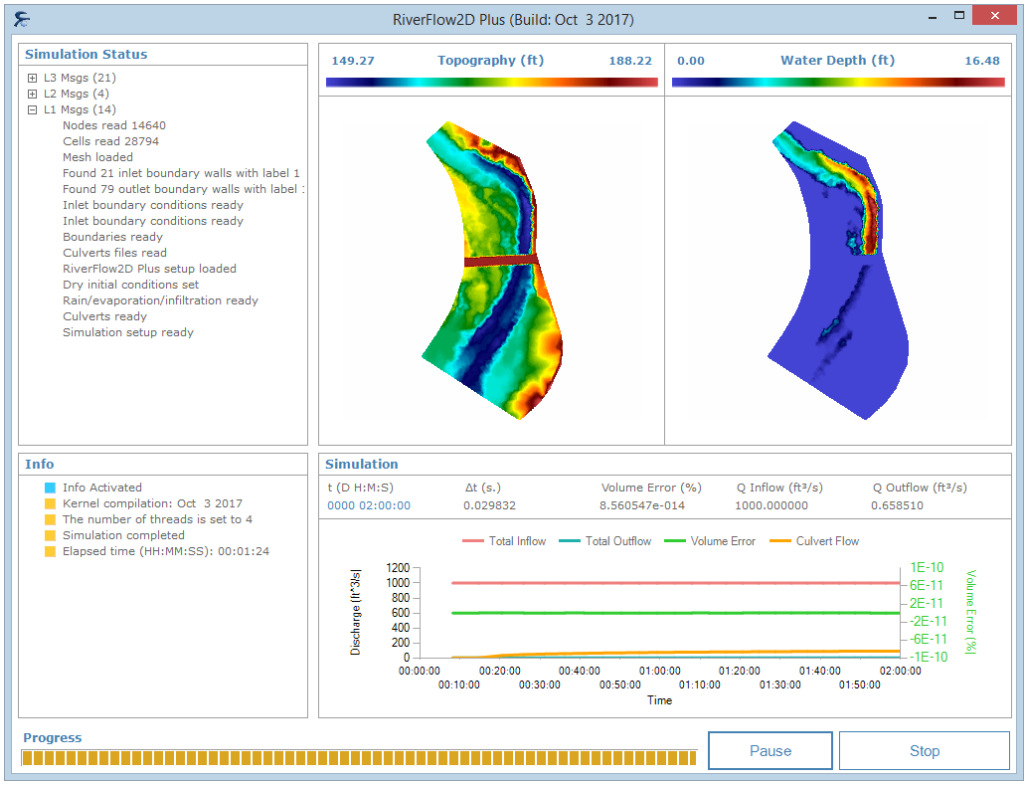


Figure 5.10 – Runtime graphics.

5.6 Review culvert output file

For each culvert, HydroBID Flood creates an output file called: `CULVERT_culvertID.out`, where `culvertID` is the name (ID) entered when we created the culverts. Output includes the series of discharge versus time through the culvert and the elevations of the water surface at the inflow and outflow locations. For this tutorial, you will find a file called `CULVERT_Culv1.out`, whose content is shown in the following figure:

```
======================================================================
======================================================================
                          RiverFlow2D
                          Build Sep  4 2018
======================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                       (C) COPYRIGHT 2009-2018 Hydronia, LLC.
                                            ALL RIGHTS RESERVED
                                            RUN DATE: 04/Sep/2018
======================================================================

======================================================================
Results for Culvert no.:                1 Culvert ID: Culv1
======================================================================
 Time       Qc       WSEL1     WSEL2
 hrs.             ft3/s     ft.       ft.
0.10000   0.000   152.949    155.719
0.20000   0.000   152.949    155.719
0.30000   16.521   156.054    155.461
0.40000   32.341   157.901    155.630
0.50000   39.857   159.060    155.612
0.60000   44.564   159.965    155.653
0.70000   48.381   160.769    155.688
0.80000   51.668   161.498    155.702
0.90000   54.357   162.126    155.712
1.00000   56.515   162.652    155.719
1.10000   58.386   163.124    155.724
1.20000   60.139   163.581    155.730
1.30000   61.805   164.027    155.734
1.40000   63.390   164.462    155.739
1.50000   64.908   164.889    155.743
1.60000   66.324   165.296    155.746
1.70000   67.597   165.670    155.751
1.80000   68.666   165.990    155.755
1.90000   69.628   166.282    155.757
2.00000   70.527   166.559    155.761
```

Figure 5.11 – Culvert1 output file.

This concludes the *Simulating culverts* tutorial.

6 — Simulating dam breaches

This tutorial illustrates how to incorporate dam breach simulation into an existing HydroBID Flood project using the QGIS interface. The exercise consists of modeling the a dam break flood. The dam is approximately 1575 feet long, the breach center is 550 feet from the right margin of the dam, the opening of the breach has the final dimensions shown in the following Figure, a top width of 160 feet, and at the bottom width of 100 feet, the final breach height is 30 feet.

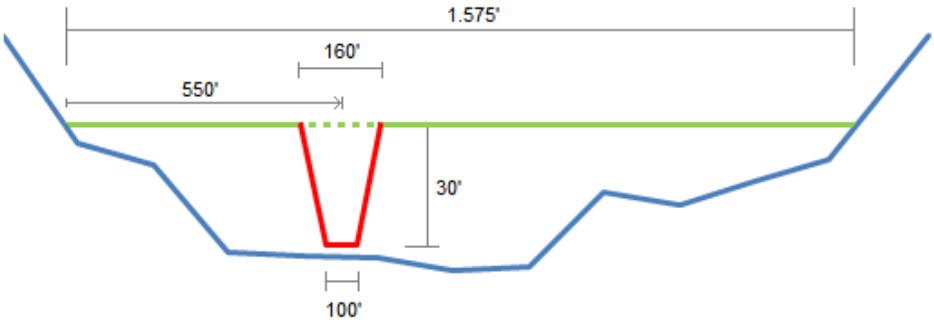


Figure 6.1 – Final dimensions of the dam breach.

For this exercise, a file with the time evolution of the breaching dimensions is required, this file can be prepared before setting the model or it can be created when entering the input the parameters. The procedure to model the dam break involves the following steps:

1. Open an existing HydroBID Flood project.
2. Create a *DamBreach* layer and to draw a line that represent the transverse axis of the dam.
3. Generate the mesh.
4. Run the model.

The files required to follow this tutorial can be found in the following directory:
`...MyDocuments\RiverFlow2D_QGIS\ExampleProjects\DamBreachTutorial`

6.1 Open an existing project

1. Open QGIS
2. On the *Project* menu click *Open...* and browse to the existing project: `DamBreach.qgs`.

This project contains the *Domain Outline* layer, the digital elevation model DEM of in raster format, outflow conditions are set to free outflow in the lower left, and an initial condition of water surface elevation behind the dam. Figure 6.2 shows the project in QGIS.

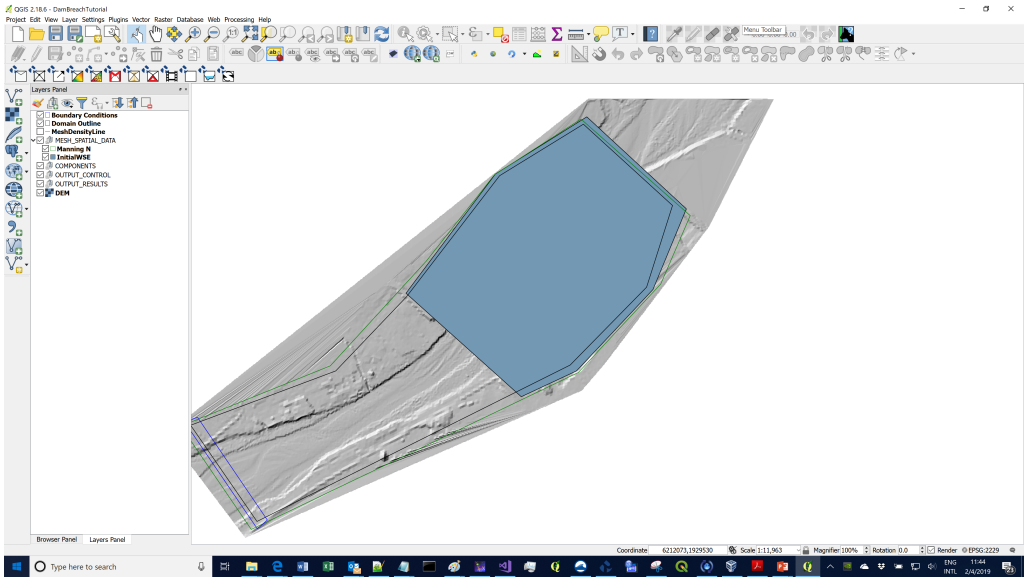


Figure 6.2 – Project loaded in QGIS.

6.2 Create the DamBreach layer and draw the line that defines the dam

Creating the dam involves the following steps:

1. Create the template for the *DamBreach* layer: in the HydroBID Flood toolbar click on the *New Template Layer* button

2. In the dialog select *DamBreach*, as shown in the figure below:

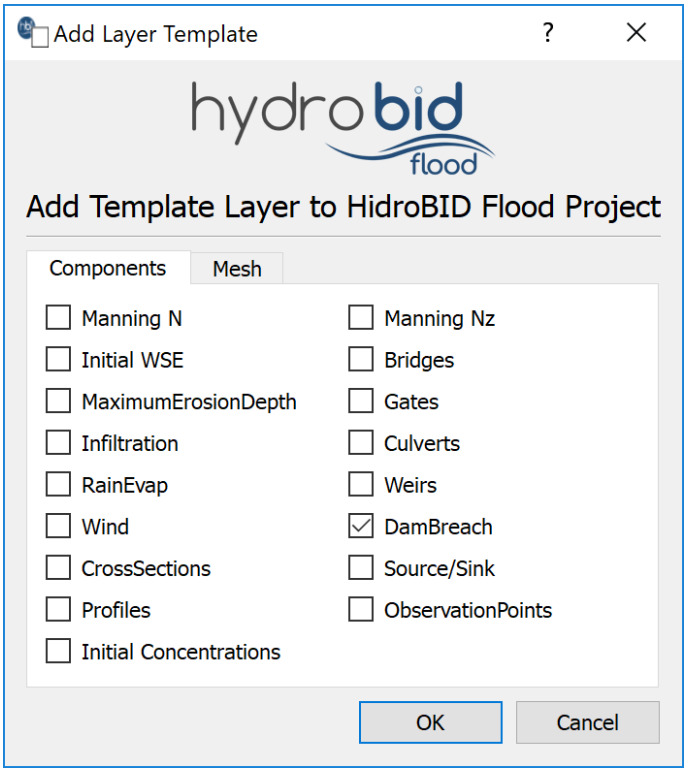
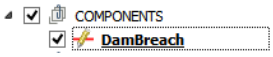


Figure 6.3 – Dialog to add a new layers.

3. Edit the *DamBreach* layer: In the layers panel, select the *DamBreach* layer.

4. In the digitalization toolbar we click on the *Toggle Editting* button

A pencil icon will appear in the *DamBreach* layer, indicating that the layer is in edit mode:



5. Draw the line that defines the axis of the dam: Using the *Add Feature* tool of the digitalization toolbar

6. Draw the line that defines the dam axis. Keep in mind that the breach centroid is measured from the first vertex of the dam line. In this example it occurs 550 feet from the left margin of the dam (Figure 6.1). The dam axis is drawn from the top of the channel (point v0) to

the bottom (point v1) along one side of the polygon that defines the initial water surface elevation, as illustrated in the image below.

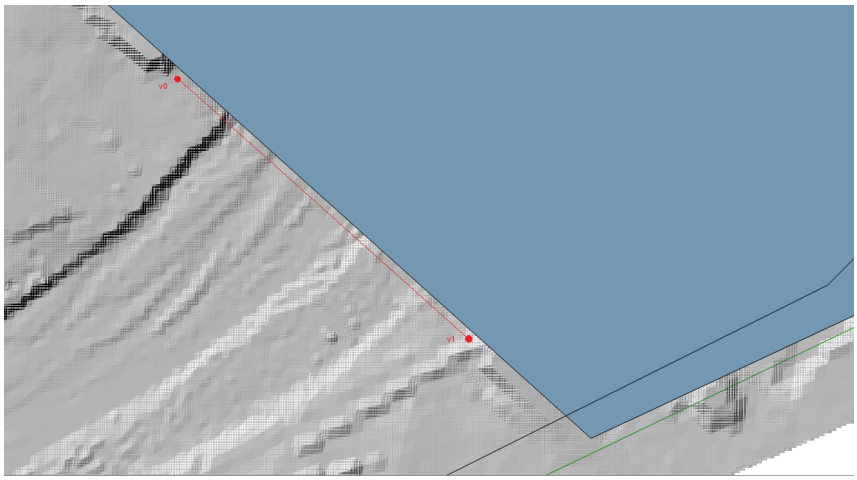


Figure 6.4 – Dam axis.

7. Input of the dam breach parameters: Once finished drawing the dam axis, the window to input the parameters of the *DamBreach* appears.

8. The tool will create the file with the breach temporal evolution data on the table. The other parameters of the breach are shown in the figure below:

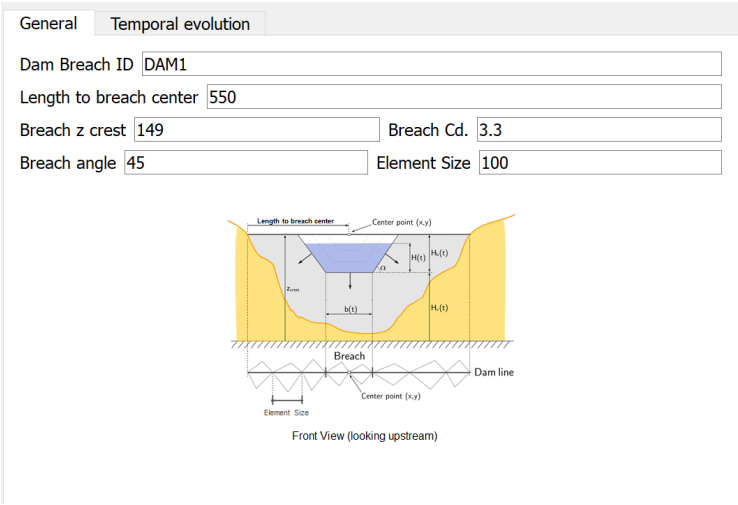


Figure 6.5 – Dialog to input the dam breach parameters.

9. The temporal evolution of the `Breach1.TXT` file is shown *Temporal evolution* tab as shown below:

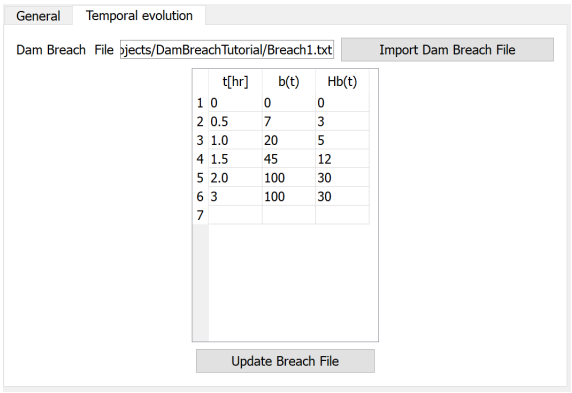


Figure 6.6 – Evolution of the breach of the dam.

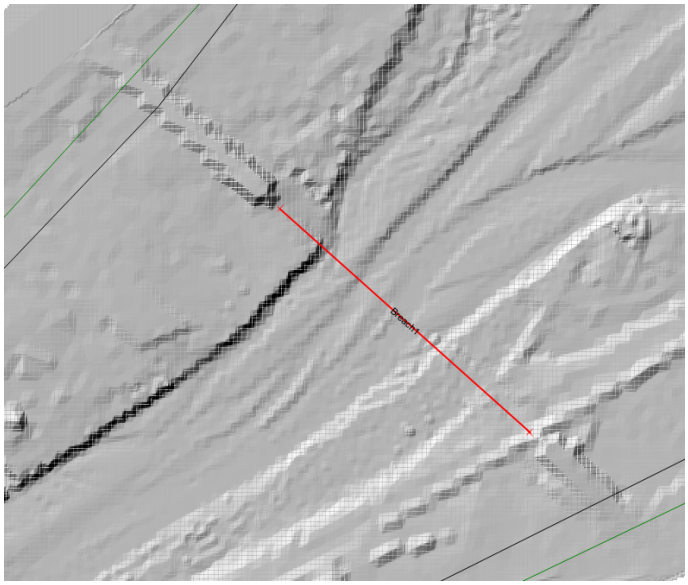


Figure 6.7 – Dam axis.

6.3 Generate the mesh

The mesh is generated using the *Generate TriMesh* tool

Figure 6.8 shows the resulting mesh of almost 11,000 cells

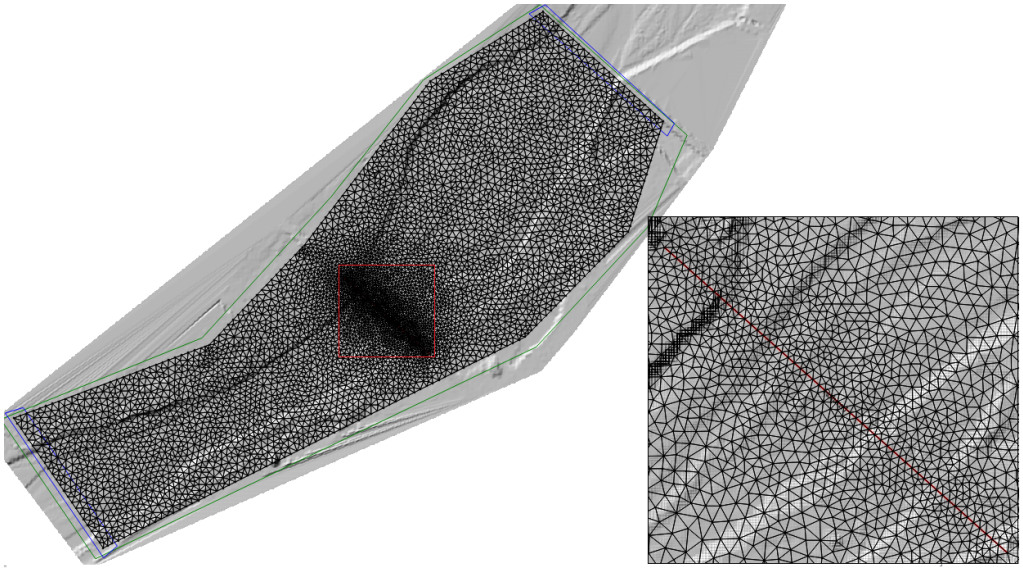


Figure 6.8 – The resulting dam breach mesh. Detail show mesh along the dam axis.

6.4 Exporting files to HydroBID Flood

1. Click on the *Export HydroBID Flood* button.

2. Select the raster layer that contains the Digital Elevation Model (DEM) and the name of the project.
3. Input the name without any extension. For this example it will be: *DamBreach*.

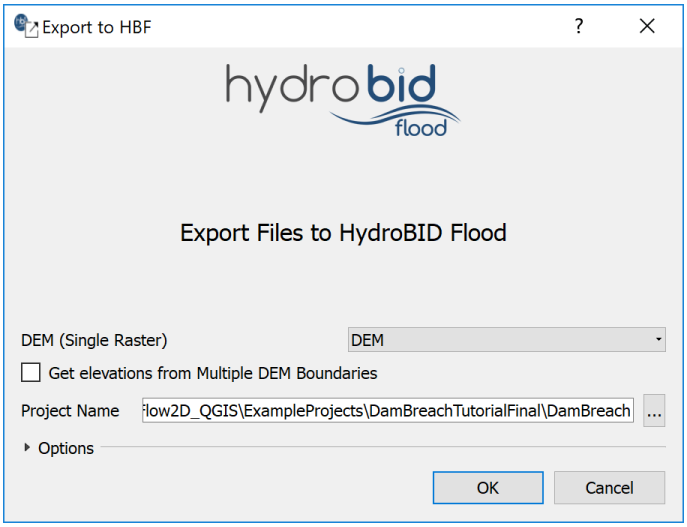


Figure 6.9 – Export dialog.

4. Once finished inputting the information, click on the OK button and the export process will begin. Once it is finished,HydroBID Flood will be loaded with the `DamBreach.DAT` file.

6.5 Running the model

After exporting the files, Hydronia Data Input Programis loaded with the project file of the `DamBreach.DAT` example and shows the *Control Data* panel as illustrated in Figure 6.10.

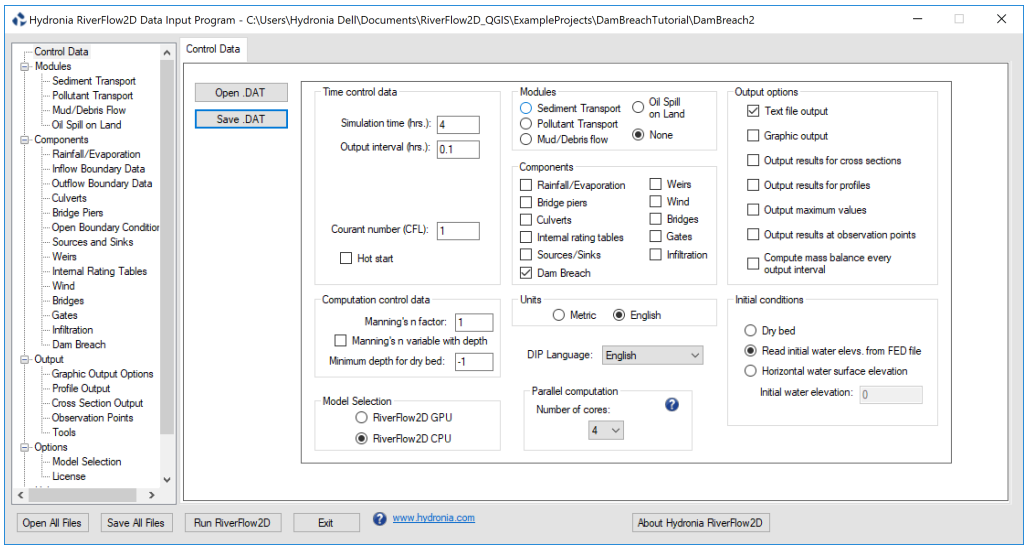


Figure 6.10 – Hydronia Data Input Program.

It can be seen that the *Dam Breach* component is selected as well as the initial condition that indicates that the initial elevation of the water surface of the `.FED` file will be read. Selecting from

the list on the left panel the *Dam Breach* component will show the panel where you can see the parameters of the dam breach as shown in the figure below:

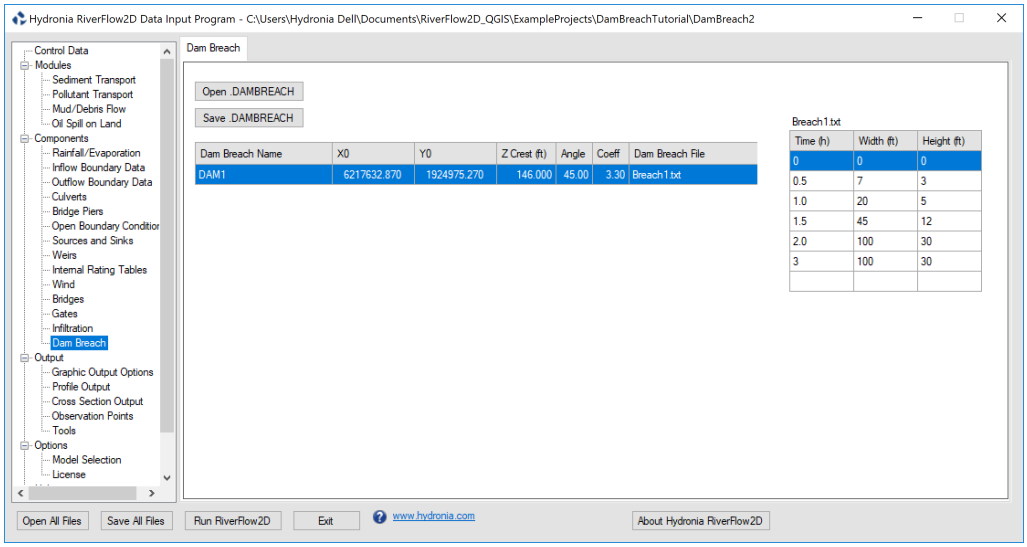


Figure 6.11 – *Dam Breach* component.

1. Before running the model, set the simulation time to 4 hours.
2. Leave all other parameters at their default values.
3. To run the model, click on the Run RiverFlow2D button in the lower section of Hydronia Data Input Program.
4. Save the changes with the same name as the `DamBreach.DAT` file.

A window will appear indicating that the model has started running. The window thatHydroBID Flood presents while running the model shows simulation time information, volume conservation error, the total input and output discharge as well as other parameters as execution progresses (Figure 6.12).

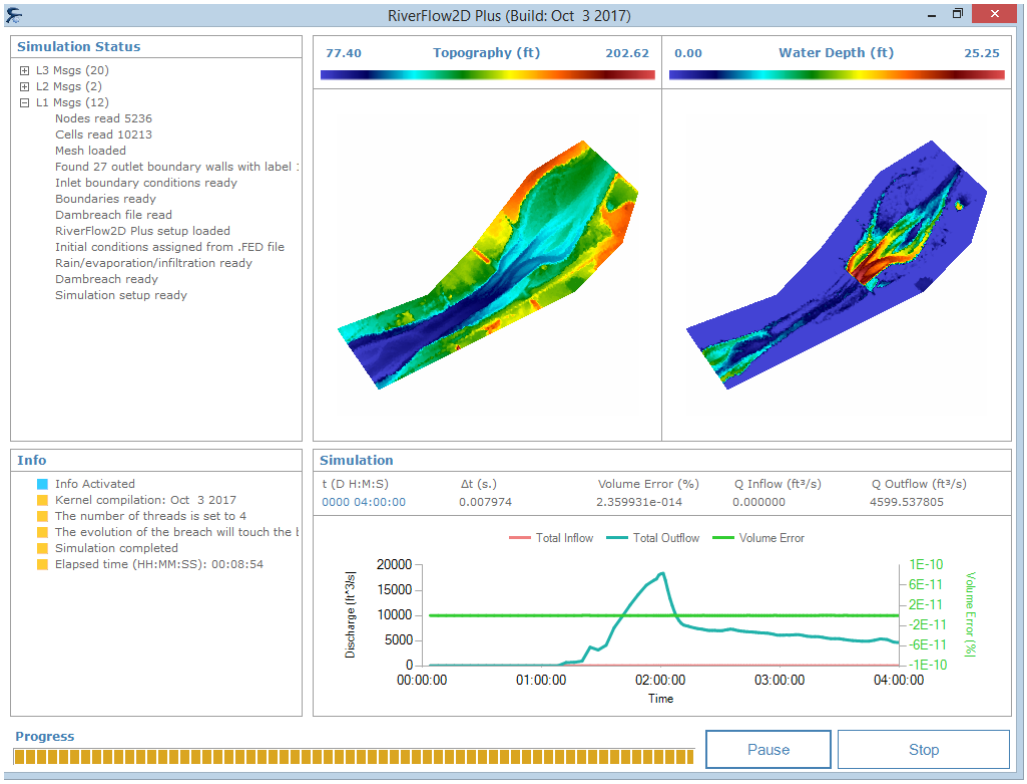


Figure 6.12 – HydroBID Flood output graphics.

6.6 Review the output files

HydroBID Flood output the dam breach hydrograph in a file with extension `.dambreachh`. Figure 6.13 shows a section of that file for this exercise.

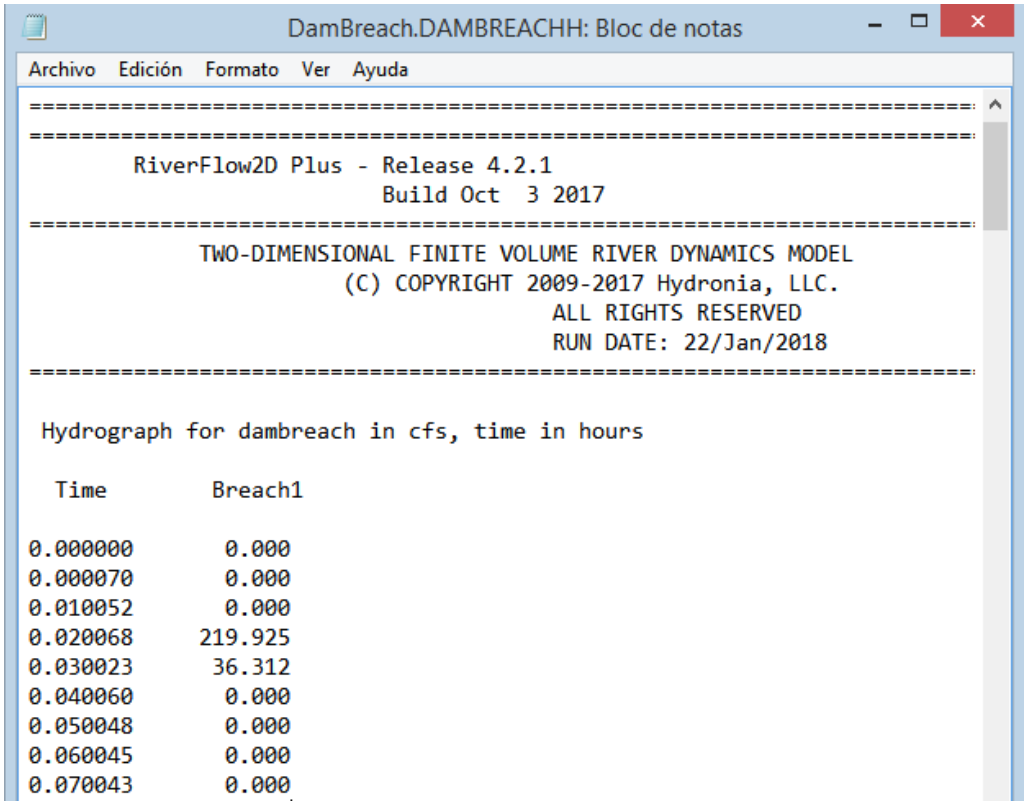


Figure 6.13 – Extract of the `DamBreach.dambreachh` file

This concludes the *Simulating dam breaches* tutorial.

7 — Using Manning's n ESRI shape files

This tutorial illustrates how to use Manning's n files in ESRI shape file format to assign Manning's n values to an existing project using the QGIS interface. The procedure includes the following steps:

1. Open an existing HydroBID Flood project.
2. Load the shape files with the Manning's n polygons.
3. Import the Manning's n geometry and values to the *Manning N* layer

The files required to follow this tutorial can be found in the following directory:

`...\Documents\RiverFlow2D_QGIS\ExampleProjects\ManningsNShapefileTutorial`

7.1 Open an existing project

1. Open QGIS
2. On the *Project* menu click *Open...* and browse to the existing project: `ManningsN_shapefile.qgs`.

   This project contains the layers of the domain contour, the Digital Elevation Model (DEM) of the river bed in raster format, an aerial photograph, and the boundary conditions layer where the inflow is located in the upper right and outflow in the lower left. The inflow boundary condition is a hydrograph with a peak discharge of 220,000 ft^3/s, and the outflow conditions is set to *Free outflow*. When you open the project you will have an image of the project loaded in QGIS as shown in 7.1.

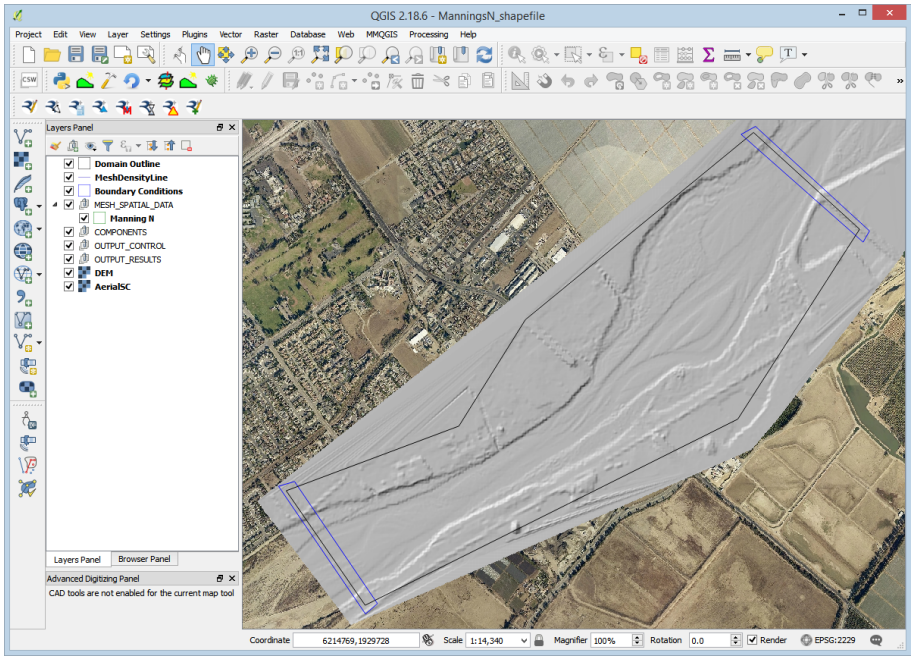


Figure 7.1 – Project loaded in QGIS.

7.2 Load the shape file with the Manning's n polygons

1. In order to load the shape file with the polygons containing the Manning's n values, click on *Add Vector Layer* button

of Manager layer toolbar or from the main menu *Layer* → *Add layer* → *Add Vector Layer...*

2. In tutorial folder, point to the `SaltRiver_ManningsN.shp` file (Figure 7.2).

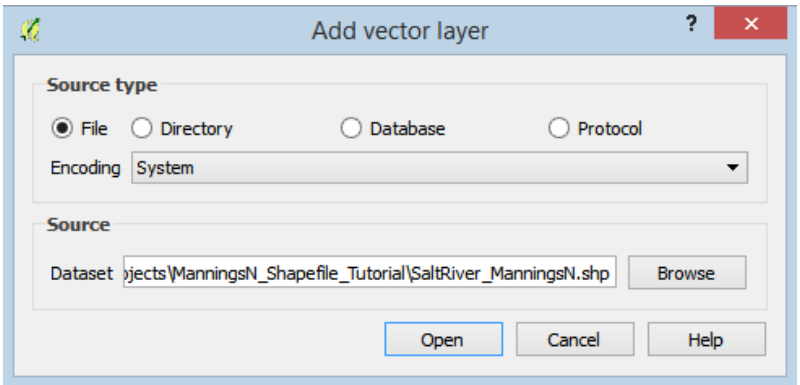


Figure 7.2 – Window to find and open the shape file.

When loading the file, an image similar to the one shown in the following figure will be displayed on the screen:

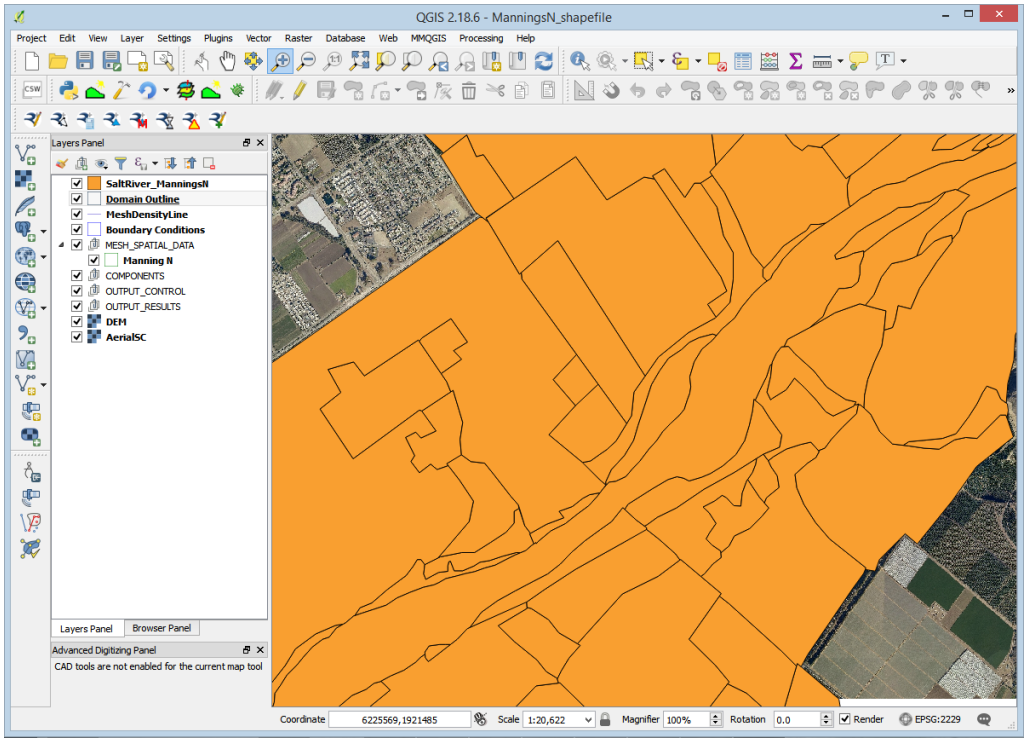
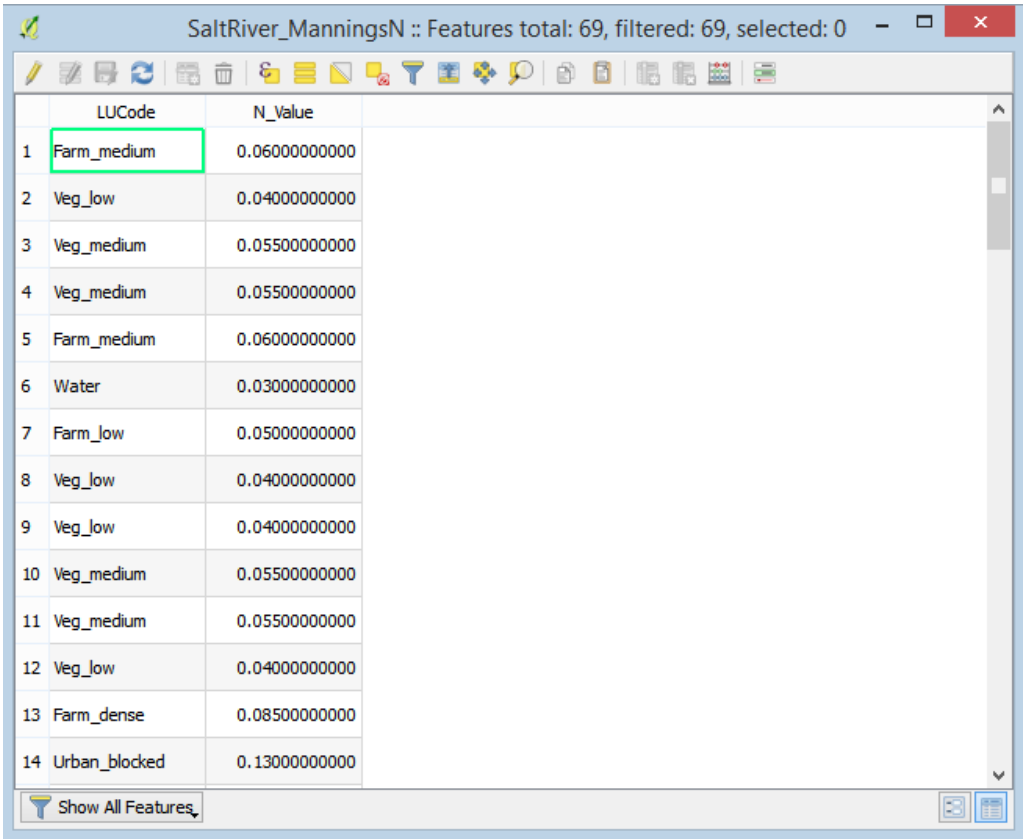


Figure 7.3 – SaltRiver_ManningsN Layer.

7.3 Import the Mannings n geometry and values to the Manning N layer

To transfer spatial and attributive information from the shape file to the *Manning N* layer, a copy and paste operation is performed, but it must be ensured that both layers have a field or column with the same name. The procedure is as follows:

1. Check the fields name of shape file: Right-click on the layer label and in the pop-up menu select the option *Open attribute table* (Figure 7.4).



	LUCode	N_Value
1	Farm_medium	0.06000000000
2	Veg_low	0.04000000000
3	Veg_medium	0.05500000000
4	Veg_medium	0.05500000000
5	Farm_medium	0.06000000000
6	Water	0.03000000000
7	Farm_low	0.05000000000
8	Veg_low	0.04000000000
9	Veg_low	0.04000000000
10	Veg_medium	0.05500000000
11	Veg_medium	0.05500000000
12	Veg_low	0.04000000000
13	Farm_dense	0.08500000000
14	Urban_blocked	0.13000000000

Figure 7.4 – Attribute table of the *SaltRiver_ManningsN* layer.

You can see that the shape file loaded has two fields, *LUCode* and *N_Value*, the first one with the coding of the land cover type and the second corresponds to the value of the Manning's n, in the case of the Manning N layer, it has a single field called *ManningN*.

2. Change the name of the field in the shape file: proceed to change the name of the field *N_Value* to *ManningN*, for this close the table of attributes and right-click on the layer label. In the pop-up menu, select Properties then in window that opens select the Fields tab as shown in Figure 7.5:

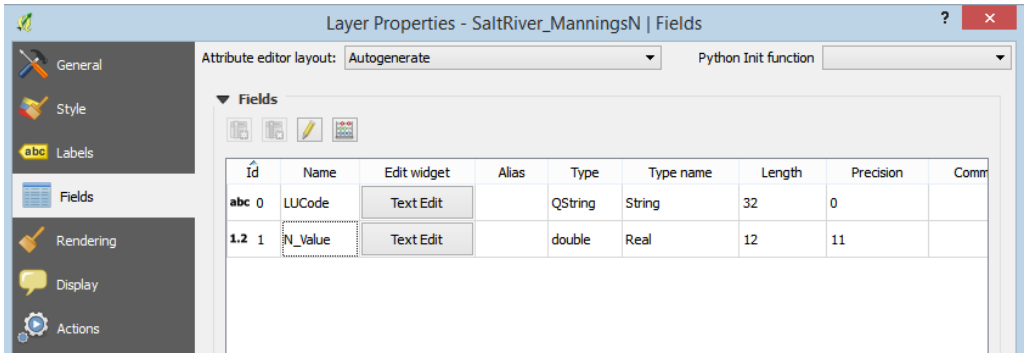


Figure 7.5 – Properties the *SaltRiver_ManningsN* layer.

3. Click on the *Toggle Editing* button

then change the *N_Value* field name by *ManningN* (Figure 7.6), and click on the *Toggle Editing* button again, and save.

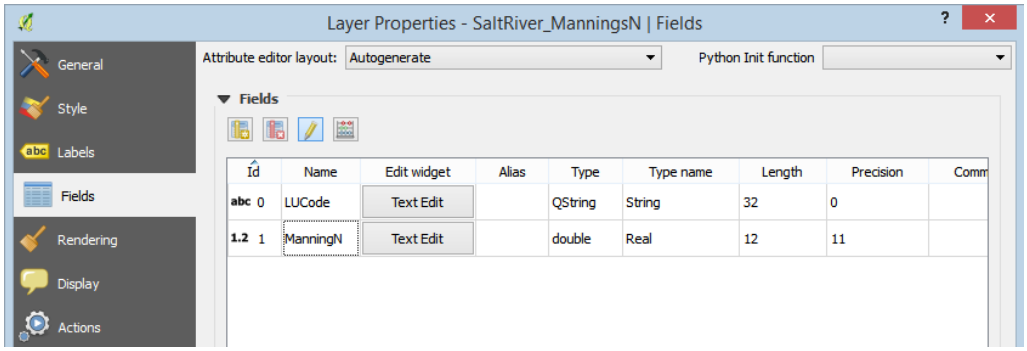


Figure 7.6 – Field properties of the edited *SaltRiver_ManningsN* layer.

4. Copy the polygons of the shape file: select the *SaltRiver_ManningsN* layer in the Layers Panel.

5. With the select tool

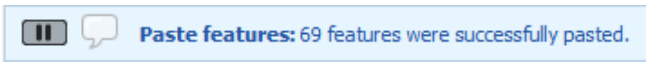
we draw a rectangle that covers the entire layer.

6. Copy the spatial elements by clicking on the *Copy* button of the digitization toolbar

7. Paste the spatial elements in the *Manning N* layer: select *Manning N* layer of the Layers Panel and set it in edit mode by clicking on the *Toggle Editing* button

8. Click on the *Paste Feature* button

   and a message will appear that indicates that features were successfully pasted.



9. Click on the *Toggle Editing* button again

   changes made to the layer are accepted and saved.

To check, open the table of attributes of the *Manning N* layer and you can see that the polygons have been copied with their Manning n values. As shown in the Figure below:

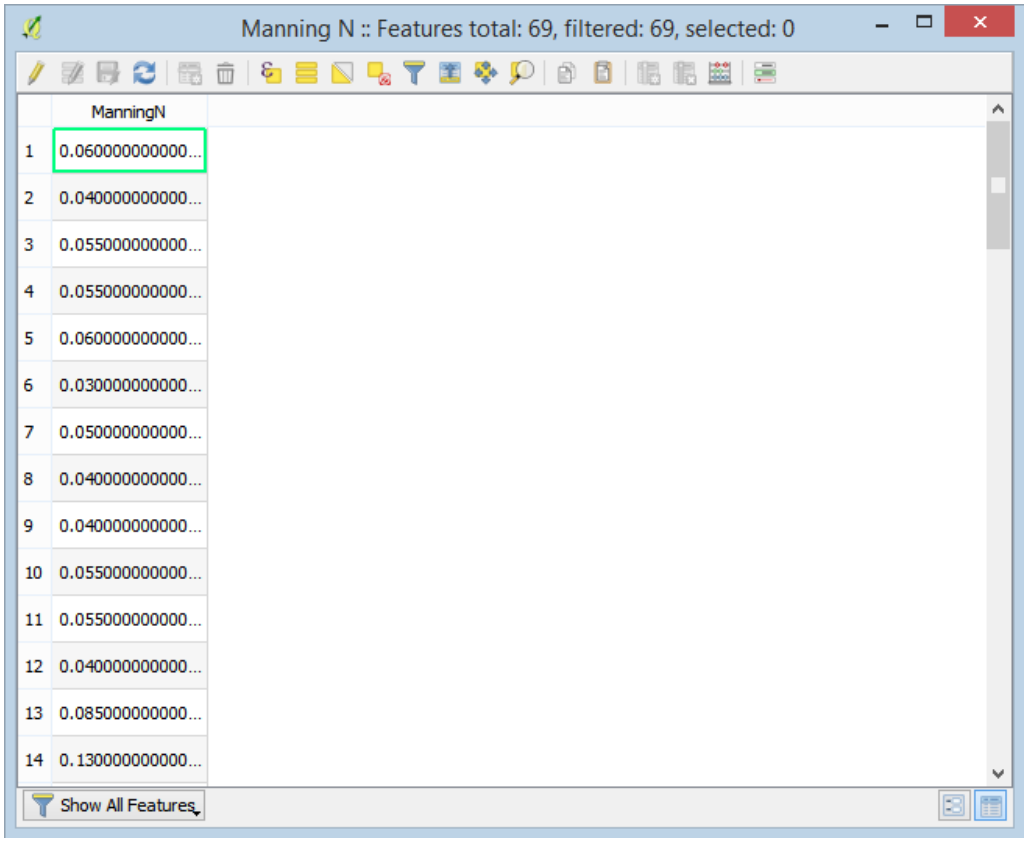


Figure 7.7 – Attribute table of the *Manning N* layer.

You can now remove the *SaltRiver_ManningsN* layer from the Layer Panel.

This concludes the *Using Manning's n ESRI shape files* tutorial.

8 — Simulating bed load sediment transport with limited erosion bed areas

In the Sediment Transport model you can define areas with a maximum erosion depth. This is useful to represent pavements, rock outcrops or any surface that does not erode or that it has a known erodible layer of sediment above it. This tutorial illustrates how to perform a sediment transport simulation in which there is a non-erodible area using the QGIS interface. The procedure includes the following steps:

1. Open an existing HydroBID Flood project.
2. Create a `MaximumErosionDepth` layer and the polygons that define the limited erosion areas.
3. Generate the mesh.
4. Running the model.

The files required to follow this tutorial can be found in the following directory:

`...\Documents\RiverFlow2D_QGIS\ExampleProjects\MaxErosionDepth`

8.1 Open an existing project

1. Open QGIS
2. In the main menu go to *Project* → *Open...* browse to the existing project: `MaxErosionDepth.qgs`.

This project contains the layers of the domain contour, of the digital elevation model DEM of the river bed in raster format, the layer with the boundary conditions where inflow is located in the upper left and outflow in the lower left. The boundary conditions are a hydrograph with a pick discharge of 6,500 ft^3/s and outflow condition is set to free outflow. When you open the project you will have a project image loaded in QGIS as shown in Figure 8.1.

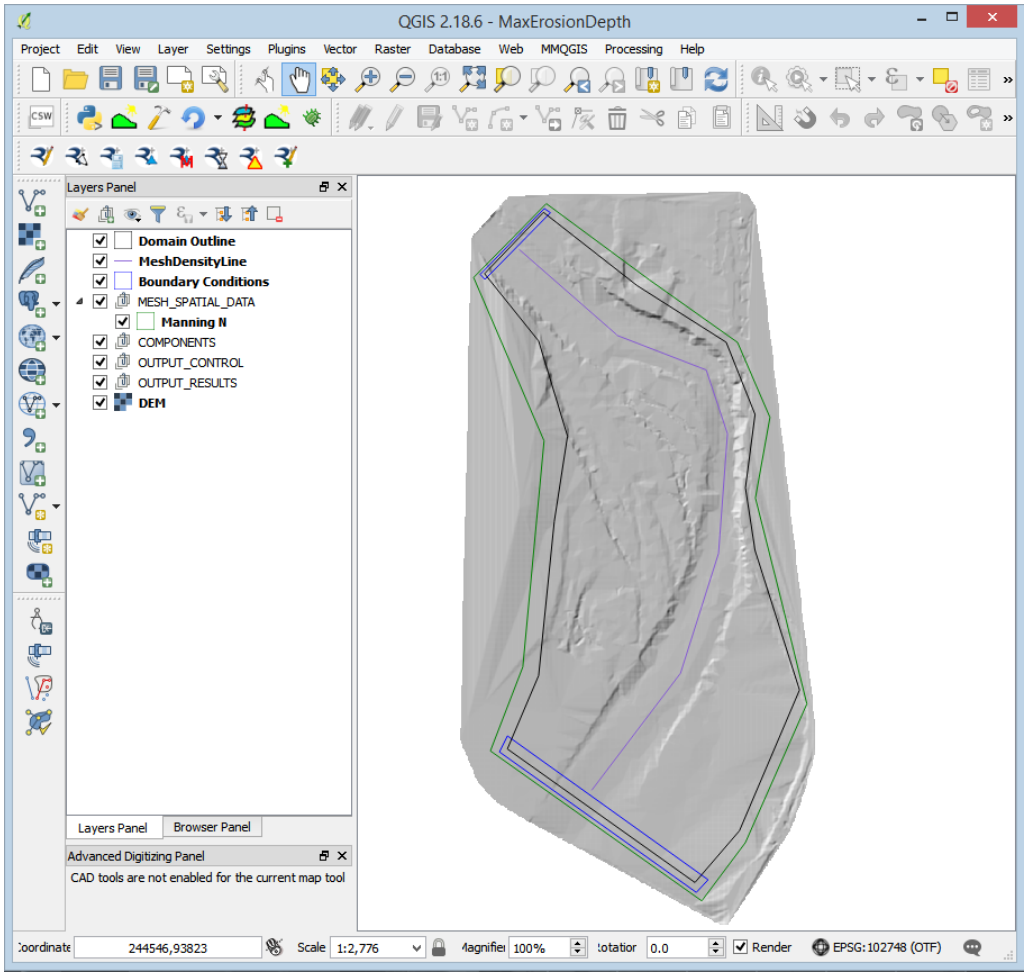


Figure 8.1 – Project screen loaded in QGIS.

8.2 Add template of the MaximumErosionDepth layer and draw the polygon that define the area of limited erosion

To add the template where the areas with limited erosion are drawn involves the following steps:

1. Create the template for the *MaximumErosionDepth* layer: In the HydroBID Flood toolbar click on the *New Template Layer* button

2. Activate the checkbox *MaximumErosionDepth*, as shown in the Figure below:

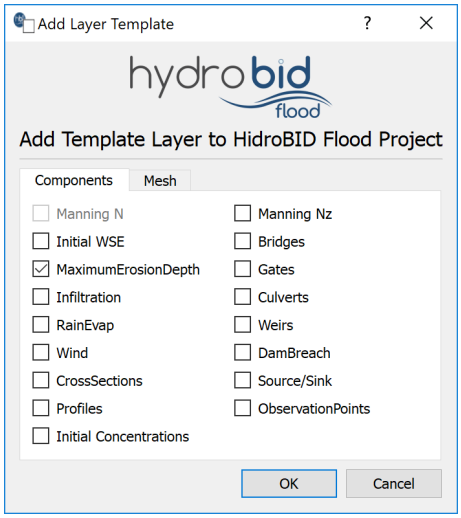
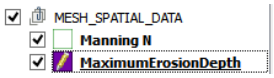


Figure 8.2 – Plugin to add a New Template Layer.

3. Edit the *MaximumErosionDepth* layer: In the layers panel, select the *MaximumErosionDepth* layer and in the digitalization toolbar we click on the *Switch tool* button

A pencil icon will appear in the *MaximumErosionDepth* layer, indicating that the layer is in edit mode:



4. Draw the polygon of the limited erosion area: Using the *Add Feature* tool

draw the polygon that defines the area of limited erosion. The polygon should cover all the cells that will have limiting erosion. In this tutorial we will assume that an area on the river has the maximum erosion depth limited to 0.1 feet, at the end you should have an image similar to the one shown in the following figure:

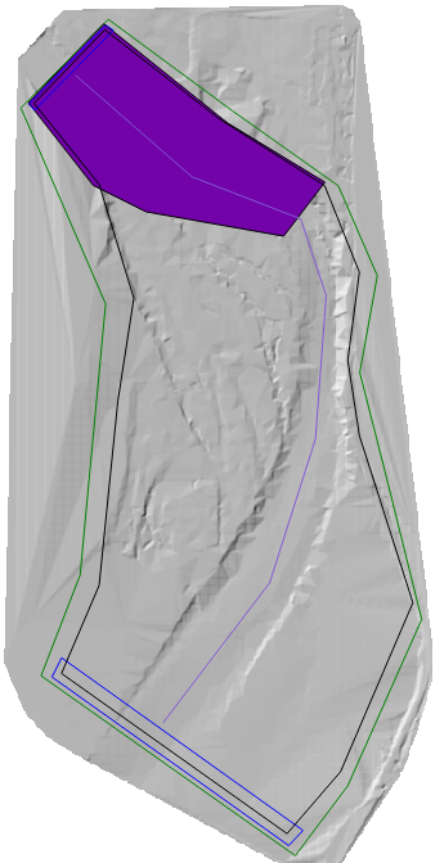


Figure 8.3 – Polygon of the area with limited erosion.

Once you finish drawing the polygon, the window to input the area parameters immediately appears. Input a maximum depth of erosion of 0.1 feet, as shown below:

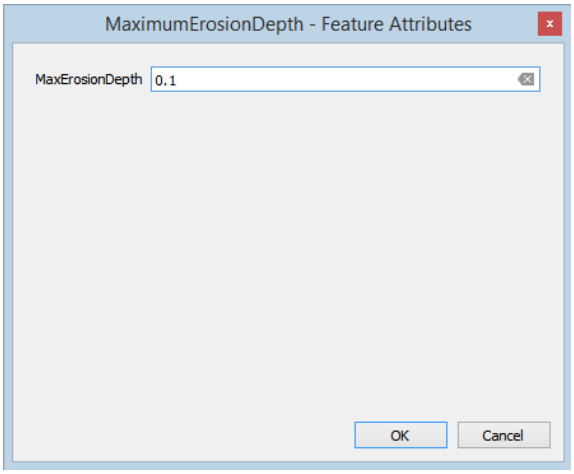


Figure 8.4 – Window for input the polygon parameters of *MaximumErosionDepth* layer.

8.3 Generate the mesh

Generate the mesh using the *Generate TriMesh* button

The results obtained as shown in Figure 8.5 (mesh of close to 17,500 cells).

Figure 8.5 – The mesh generated.

8.4 Exporting files to HydroBID Flood

Now that you have generated the mesh and you have the other layers with the necessary data, export the files in the format required by HydroBID Flood.

1. Click on the *Export HydroBID Flood* button

2. When run the plugin a window is displayed, select the raster layer that contains the Digital Elevation Model (DEM) and the name of the project to be exported. Input the name without any extension. For this example it will be: `MaxErosion`.

3. Before running the plugin activate the layer with the DEM (if it is deactivated).

   Once the plugin is executed, a window will be shown (Figure 8.6), as it should be for our example.

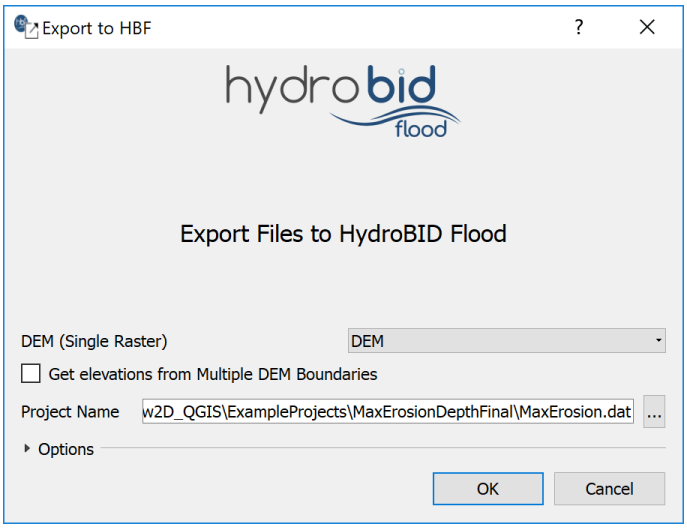


Figure 8.6 – Plugin window to export the files to HydroBID Flood.

4. Once finished inputting the information, click on the OK button and the export process will begin.

Once it is finished, tHydroBID Flood will be loaded with the `MaxErosion.DAT` file of the specific example.

8.5 Running the model

After exporting the files, HydroBID Flood is loaded with the project file of the `MaxErosion.DAT` example and shows the *Control Data* panel to it as illustrated in Figure 8.7

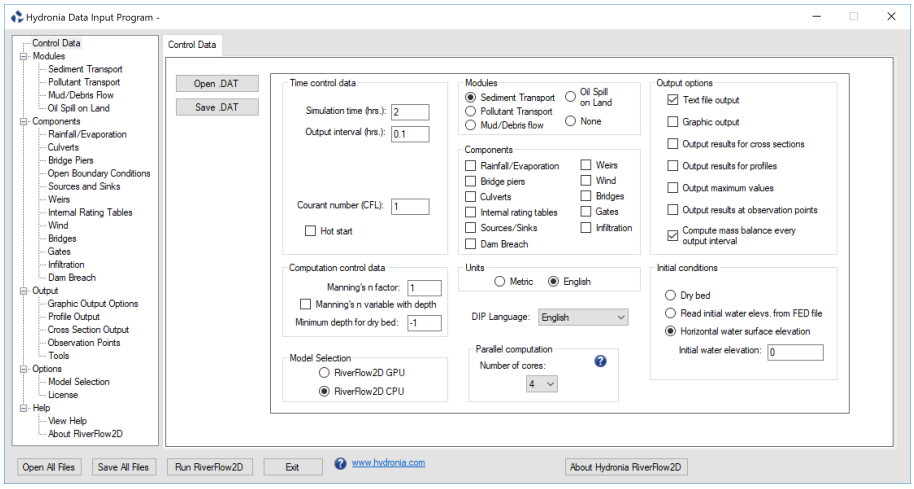


Figure 8.7 – Control data panel.

Note that the sediment transport module appears selected and displays a message warning that the file with the sediment information must be created. The procedure includes the following steps:

1. To create the `.SEDB` file with the parameters to calculate sediment transport: in the Modules list select SedimentTransport.
2. Enter the parameters for transport in suspension and bed load transport, for this example the transport of sediments in suspension is deactivated and bed load transport is left active.
3. Add the sediment fractions to be considered: for this example add a single fraction with the default values presented by the Hydronia Data Input Program, we will have an image similar to the one shown in the following Figure:

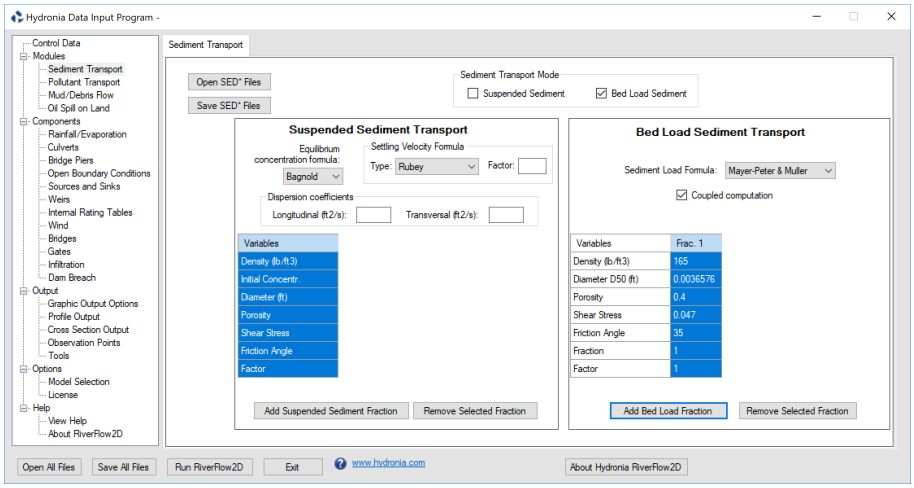


Figure 8.8 – HydroBID Flood Sediment Transport Module.

Leave all other parameters at their default values.

4. To run the model, click on the *Run RiverFlow2D* button in the lower section of Hydronia Data Input Program.

5. Save the changes with the same name of the `MaxErosion.DAT` file, then a window will appear indicating that the model started running.

The window presented while running the model shows: information of the simulation time, the volume conservation error, the total discharge of the liquid flow in and out and in this case also shows the sediment load at the inlet and outlet as well as other parameters as the execution progresses (Figure 8.9).

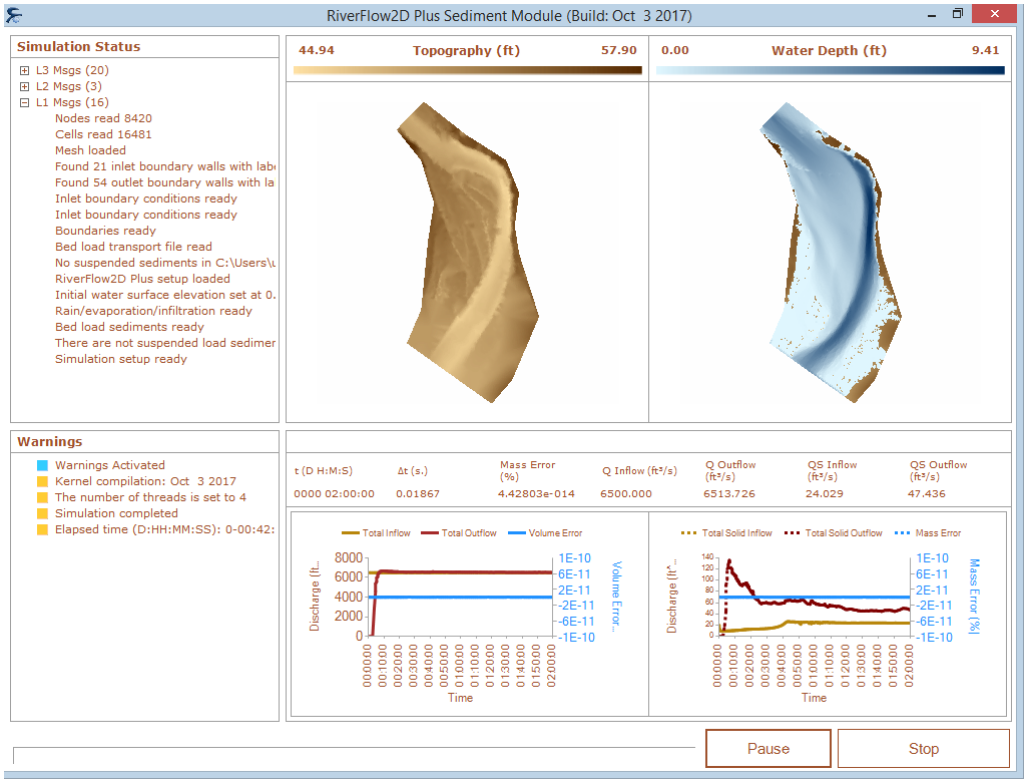


Figure 8.9 – HydroBID Flood output graphics.

8.6 Check the output files

HydroBID Flood creates the following files for output time interval defined in the *Control Data* panel:

`CELL_TIME_METRIC_DDDD_HH_MM_SS.TEXTOUT` (Metric Units) or

`CELL_TIME_ENG_DDDD_HH_MM_SS.TEXTOUT` (English Units)

where DDDD indicates the date, HH, hour, MM minutes and SS seconds.

Column 6 reports the changes in the elevation of the river bed with respect to the initial elevation. We can also visualize the changes in the elevation of the riverbed generating layers in vectorial format map from the aforementioned files using the *Maps of Results vs Time*

In the following figure, the river elevation difference map for the end of the run. At the time 0000:02:00:00 it can be observed that the zone where the erosion was limited, does not present erosion, but deposition:

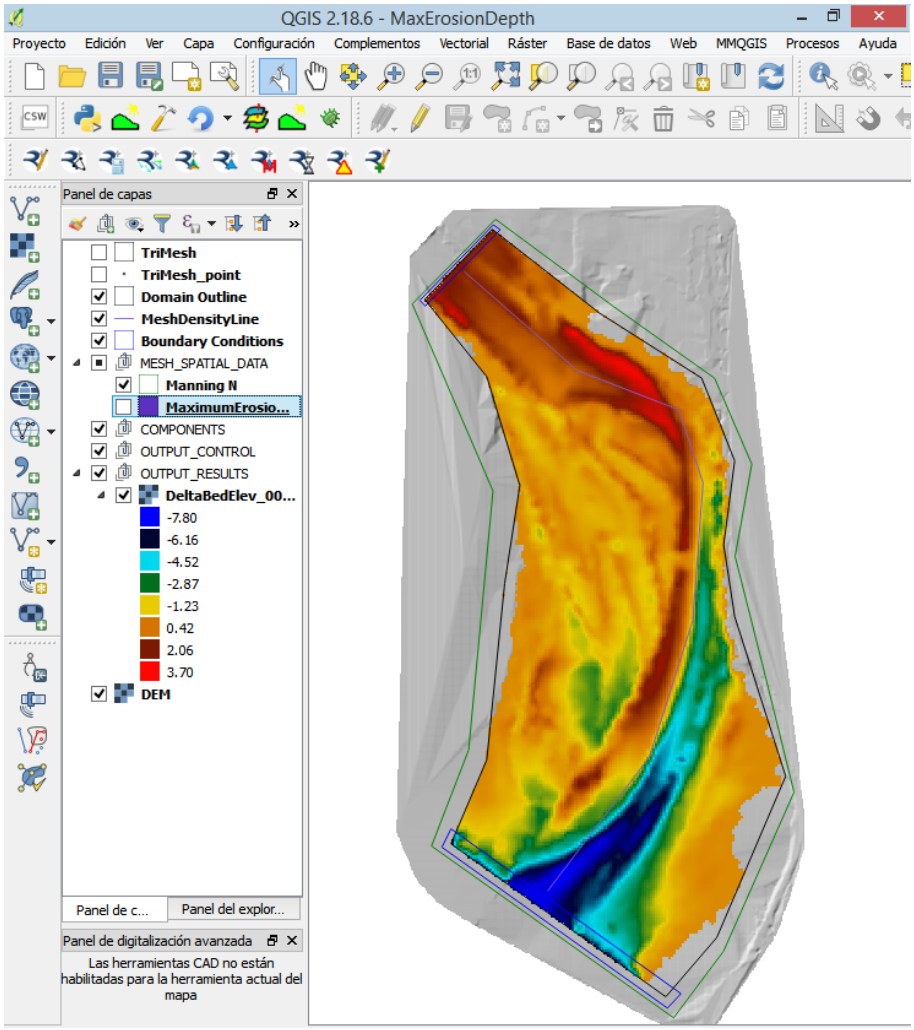


Figure 8.10 – Maps of elevation difference of the river bed between the initial time and at the end of the run.

This concludes the *Simulating bed load sediment transport* tutorial.

9 — Hydrologic simulations

The conceptual model of a hydrologic simulation with RiverFlow2D requires a series of non-overlapping polygons where the rainfall/evaporation and infiltration data will be assigned to the mesh. Only areas covered by polygons will receive rainfall or consider infiltration depending on the case. Each Rainfall/Evaporation polygon should be associated with a file containing a rainfall and evaporation time-series file. Similarly, each Infiltration polygon should correspond with a file containing the infiltration calculation method and its parameters for the polygon. The user will need to generate the rainfall and infiltration data files associated with each polygon, and copy them to the project folder prior to running the model.

This tutorial illustrates how to perform a hydrologic simulation accounting for rainfall, evaporation and infiltration. The procedure includes the following steps:

1. Create the rainfall and evaporation time series data files.
2. Create the infiltration data files.
3. Open an existing RiverFlow2D project.
4. Add the template of the *RainEvap* component layer and the rain/evaporation polygons.
5. Add the template of the Infiltration component layer and the Infiltration polygons.
6. Generate the mesh.
7. Running the model.
8. Visualize model results.

Note: The files required to follow this tutorial can be found in the following directory:

`...MyDocuments\RiverFlow2D_QGIS\ExampleProjects\RainfallInfiltrationTutorial`

9.1 Create the rainfall and evaporation time series data file

To run a hydrologic simulation with RiverFlow2D, polygons will be created on which the rainfall/evaporation data will be applied, but you must first create the ASCII text files with the rainfall and evaporation time series that will be associated with each polygon. These files can be created with any text editor, such as Notepad or Wordpad. The rainfall/evaporation file has the following format:

Line 1: NPRE `Number of points in the time series of rainfall and evaporation`

Then follows NPRE lines containing:

`Time (hr)  Precipitation intensity (mm/h or in/h)  Evaporation (mm/h or in/h)`

The following table is an excerpt of the `Rainfall1.dat` file that is included in the folder for this tutorial. In this example, evaporation is assumed to be zero at all times. Figure 9.1 shows the graphical representation of the rainfall time series in the `Rainfall1.dat` file.

```
18
0.0 0.0 0.0
0.08 1.8 0.0
0.17 3.5 0.0
0.25 7.8 0.0
0.33 12.0 0.0
0.42 15.0 0.0
...
1.17 9.0 0.0
1.25 4.7 0.0
1.33 3.0 0.0
1.42 2.4 0.0
```

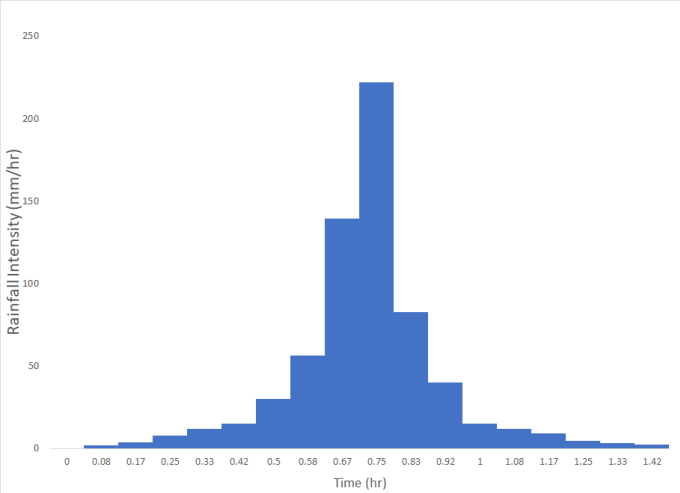


Figure 9.1 – Rain intensity time series.

9.2 Create the infiltration parameters data file

To take into account infiltration, polygons will be created in which you specify the infiltration calculation method, and its corresponding data. The infiltration polygons are completely independent from the precipitation polygons. Each polygon can use a different method with its associated parameters. You will need to create the infiltration ASCII text files for each polygon. These files can be created with any text editor, such as Notepad or Wordpad. The infiltration file is described in detail in the RiverFlow2D Reference Manual. For this tutorial, given that the watershed has an area with natural coverage in the upper area, and another with urban use in the lower area of watershed, we will use two infiltration files using the the SCS-CN method to calculate the infiltration. File `Infiltration1.dat` will be used for the upper watershed, and for the lower urban zone, `Infiltration2.dat` file. Both files are provided in the folder for this tutorial.

9.3 Open an existing project

1. Open QGIS
2. On the *Project* menu click *Open...* and browse to the existing project: `RainfallInfiltrationTutorial.qgs`. This project contains the information to simulate the rainfall runoff resulting from a 10 yr storm. The layers contained are the following:
    - Domain Outline
    - Digital Elevation Model (DEM) in raster format
    - Aerial photography
    - Polygons with the Manning's n coefficients

- Cross section at watershed outlet
- Outflow boundary conditions set to free outflow

When you open the project you will have an view similar as that shown in Figure *6-2*.

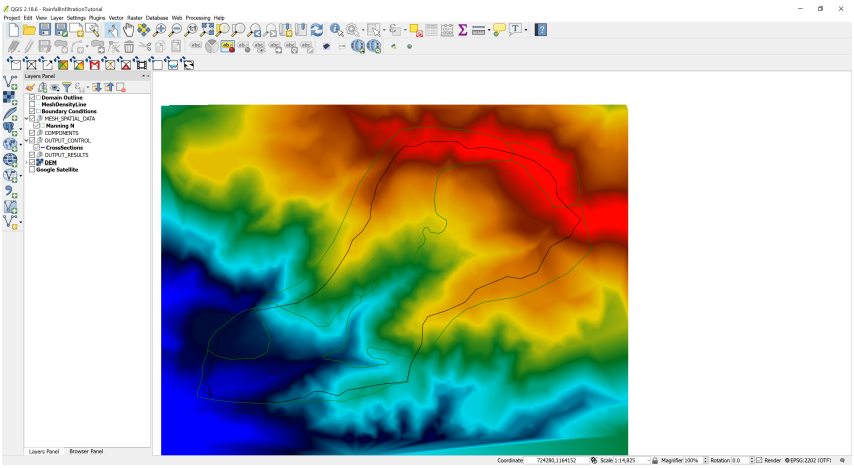


Figure 9.2 – Rainfall and infiltration tutorial loaded in QGIS.

9.4 Add the *RainEvap* component layer, and the rainfall/evaporation polygons

To add the *RainEvap* where the polygons are drawn with the rainfall and evaporation data, do the following:

1. To create *RainEvap* layer use the *New Template Layer* button

2. In the window check *RainEvap*, as shown in the Figure below. Then click OK:

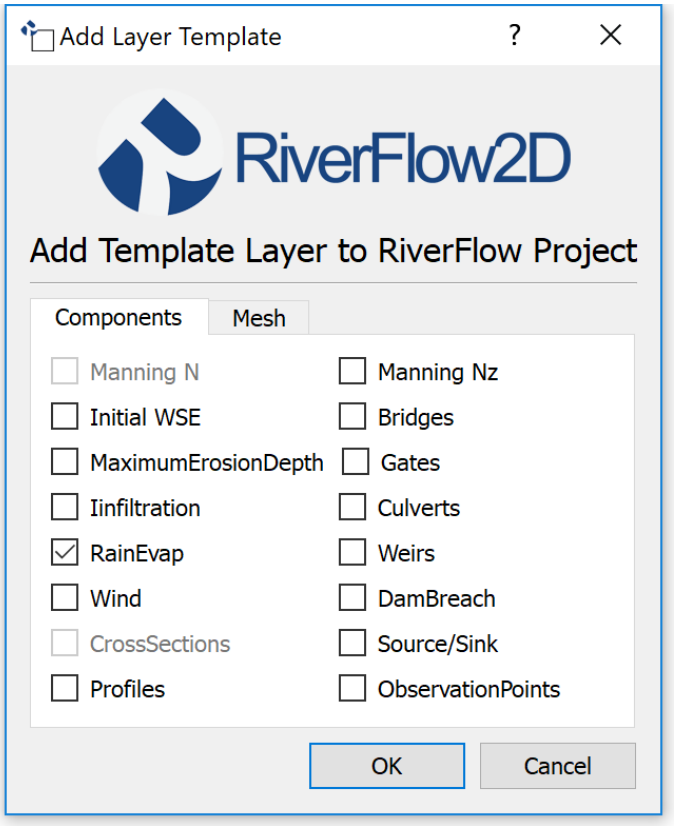
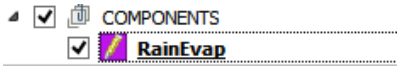


Figure 9.3 – Dialog to add a new layer.

3. Edit the *RainEvap* layer: In the layers panel, select the *RainEvap* layer and click on the *Toggle Editing* button in the digitization bar

A pencil icon will appear in the *RainEvap* layer, indicating that the layer is in edit mode:



4. Draw the polygon that demarcates the spatial distribution of rainfall and evaporation: Using the *Add Feature* tool from the digitalization bar

draw the rainfall/evaporation polygon, as only one file will be used. The polygon should covers the entire *Domain Outline* as shown in the figure below:

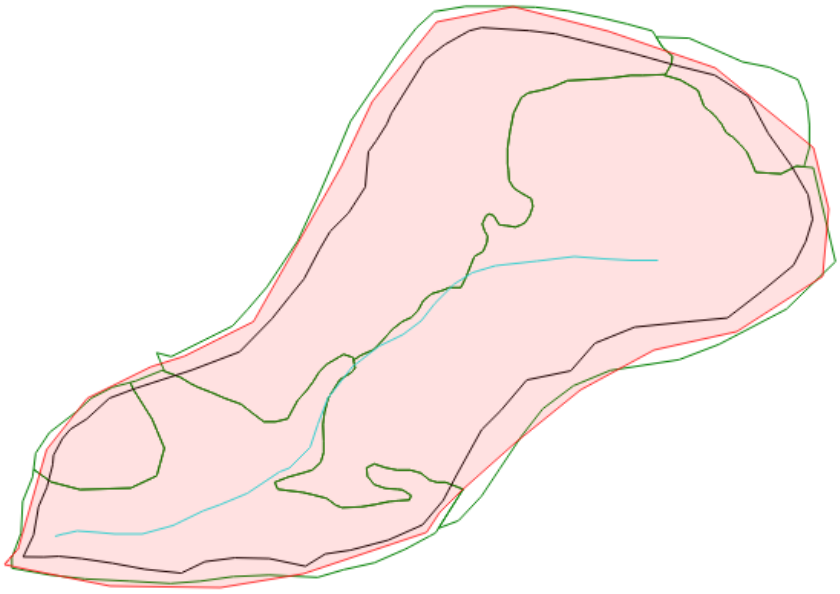


Figure 9.4 – Polygon with the spatial distribution of Rainfall/evaporation data.

5. Input the parameters or attributes of the RainEvap polygon: just after the polygon is finished, the dialog to input the parameters will open, clicking on the button [...] search the `Rainfall.dat` file in the folder of this tutorial.

9.5 Add the *Infiltration* component layer, and the Infiltration polygons

To add the infiltration information, do as follows:

1. To create the *Infiltration* layer y use the *New Template Layer* button

2. In the dialog select Infiltration, as shown in the Figure below:

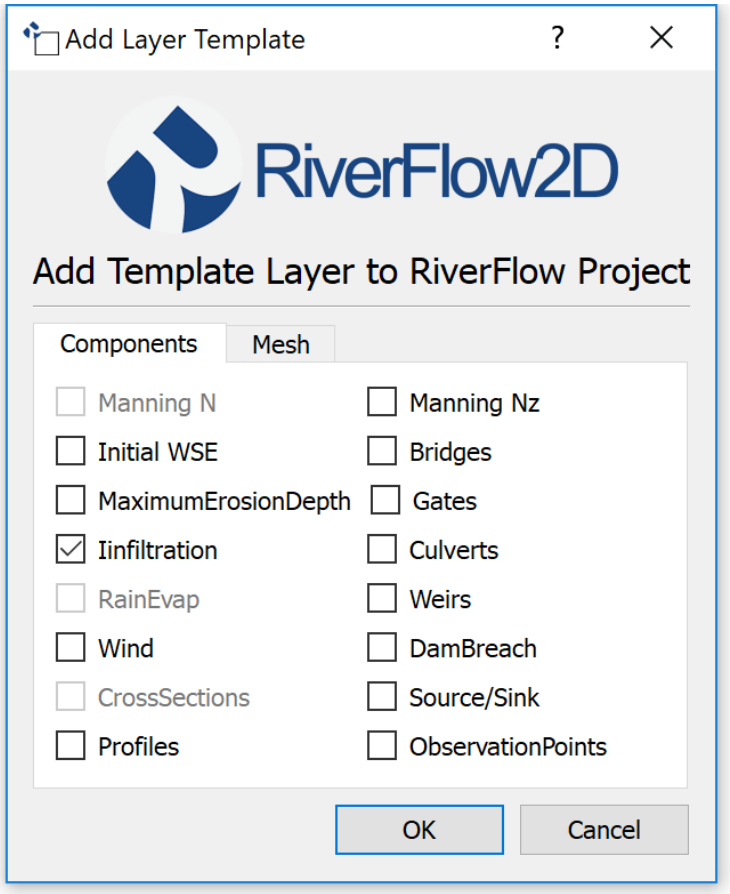
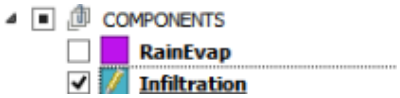


Figure 9.5 – Plugin to add a new template layer.

3. Edit the *Infiltration* layer: In the layers panel, select the *Infiltration* layer and click on the *Toggle Editing* tool

A pencil will appear in the *Infiltration* layer, indicating that the layer is in edit mode:



4. Draw the polygon that demarcates the spatial distribution of the infiltration: Using the tool Add Feature tool of the digitalization bar, draw the infiltration polygons

Figure 9.6 shows the polygons that define the two infiltration zones of the watershed that are based on the land use and vegetation cover.

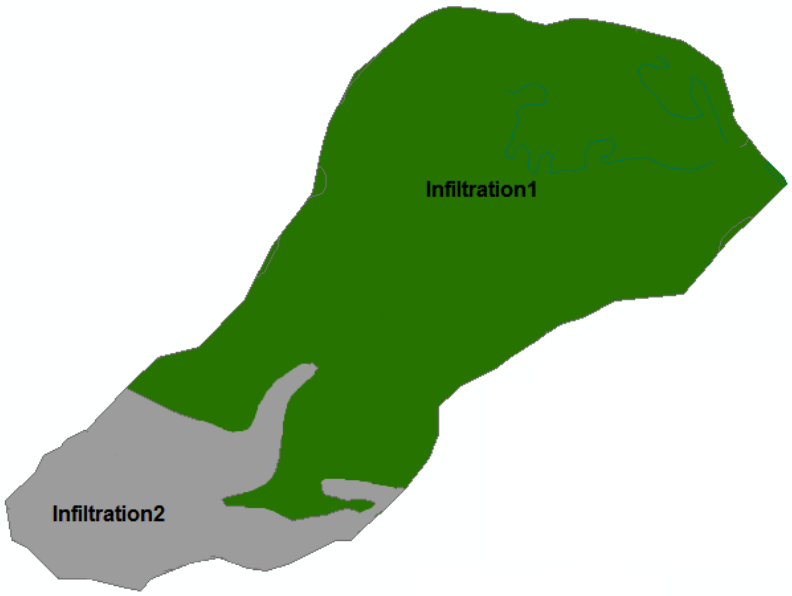


Figure 9.6 – Infiltration areas of the watershed.

5. Draw a polygon for the *infiltration2* area trying to maintain the shape as indicated in the previous figure and that protrudes from the polygon of the Domaine Outline as shown in the figure below:

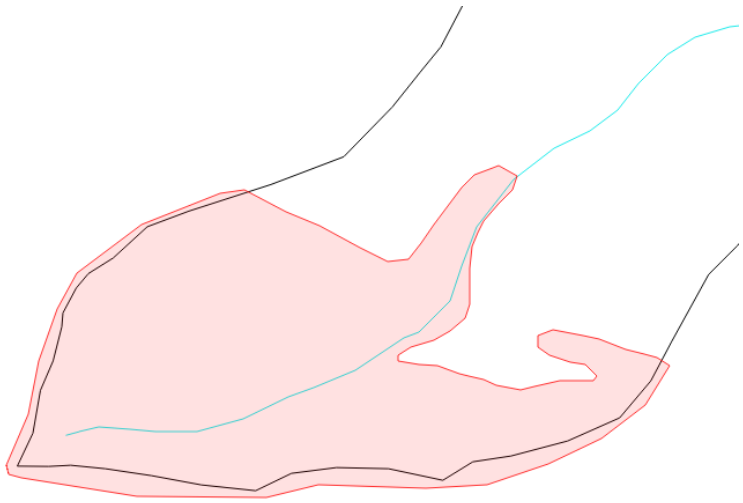


Figure 9.7 – Polygon of the infiltration2 area of the watershed.

6. Once completing the polygon, the dialog to input the parameters opens. Browse to the input file *Infiltration2.txt*, as shown below:

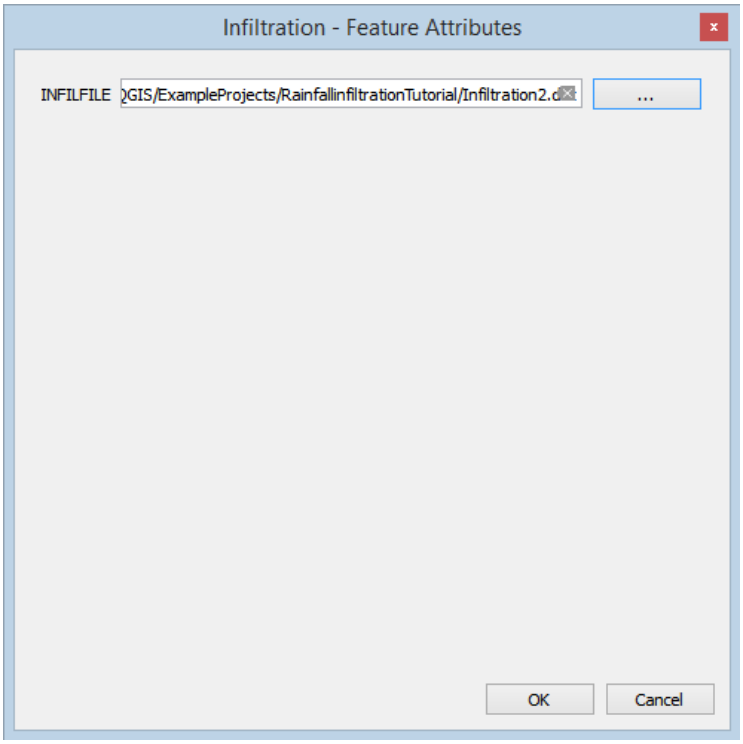


Figure 9.8 – Window to input the parameters for the infiltration2 polygon.

7. To draw the second polygon corresponding to *infiltration2* use the snapping option as shown in the section *Advanced Digitalization/Snapping Tutorial* and there should be a polygon like the one shown in the following figure:

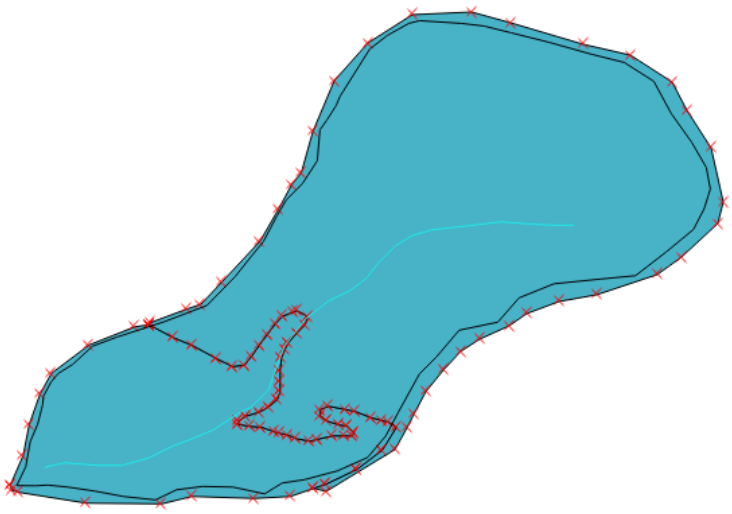


Figure 9.9 – Polygons of the infiltration areas of the watershed.

8. Once finished drawing the polygon, enter the file name `Infiltration1.txt` as shown:

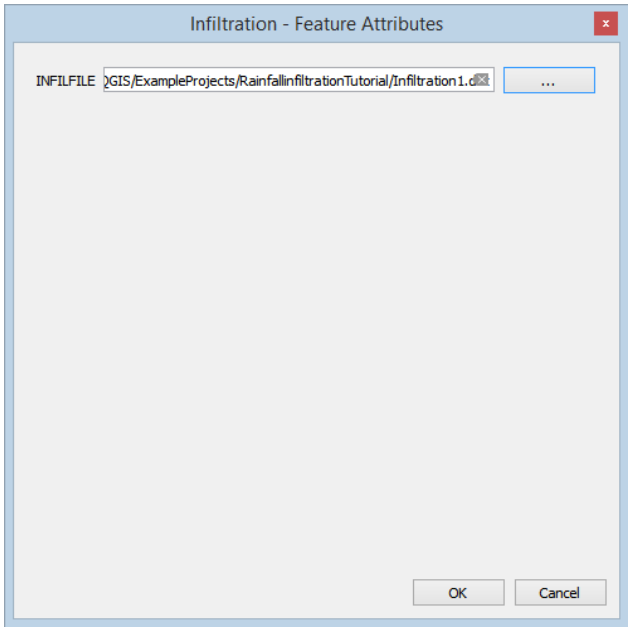


Figure 9.10 – Dialog to enter the parameters for the infiltration1 polygon.

9.6 Generate the mesh

The mesh is generated using the *Generate TriMesh* icon

Figure 9.11 shows the resulting mesh of about 140,000 cells.

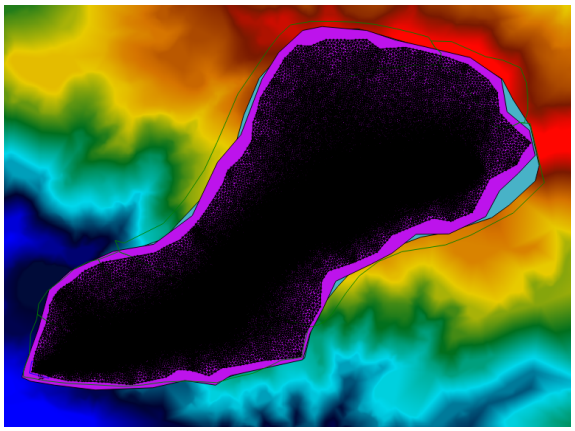


Figure 9.11 – Final mesh.

9.7 Exporting files to RiverFlow2D

Now that once you have generated the mesh, and the other layers are ready with the necessary data, you should export the files in the format required by RiverFlow2D. We will use the *Export RiverFlow* icon.

1. Click on the *Export RiverFlow2D* icon.

   When running the export command, you need to select the raster layer that contains the Digital Elevation Model (DEM), and the name of the project to be exported, only the name must be indicated without any extension.

   When the export files finalizes, a window will be appear as shown in Figure 9.12.

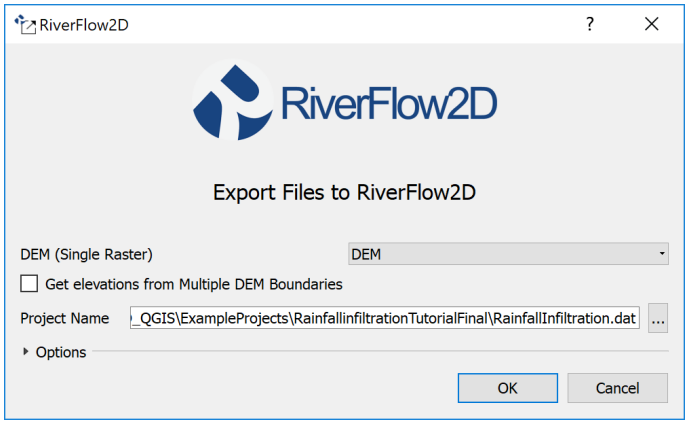


Figure 9.12 – Export dialog.

2. After entering the data, click on the OK button and the export process will begin. Once finished, the RiverFlow2D program will be loaded with the `RainfallInfiltration.DAT` file.

9.8 Running the model

After exporting the files, the RiverFlow2D program is loaded with the project file from the `RainfallInfiltration.DAT` example and shows the *Control Data* panel to it as illustrated in Figure 9.13

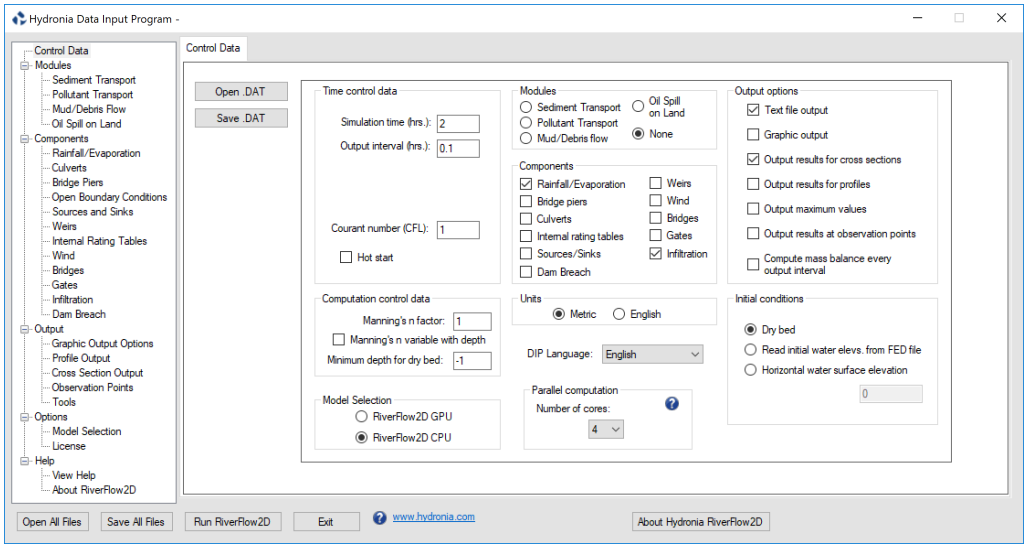


Figure 9.13 – Control data panel.

As shown above, the Rainfall Components Evaporation/Infiltration are selected in the data panel.

1. Select *Rainfall/Evaporation* and the rainfall data contained in the `rainfall1.txt` file will appear (Figure 9.14). In the *Infiltration* panel the data contained in the *Infiltration1* and *Infiltration2* files is shown (Figure 9.15).

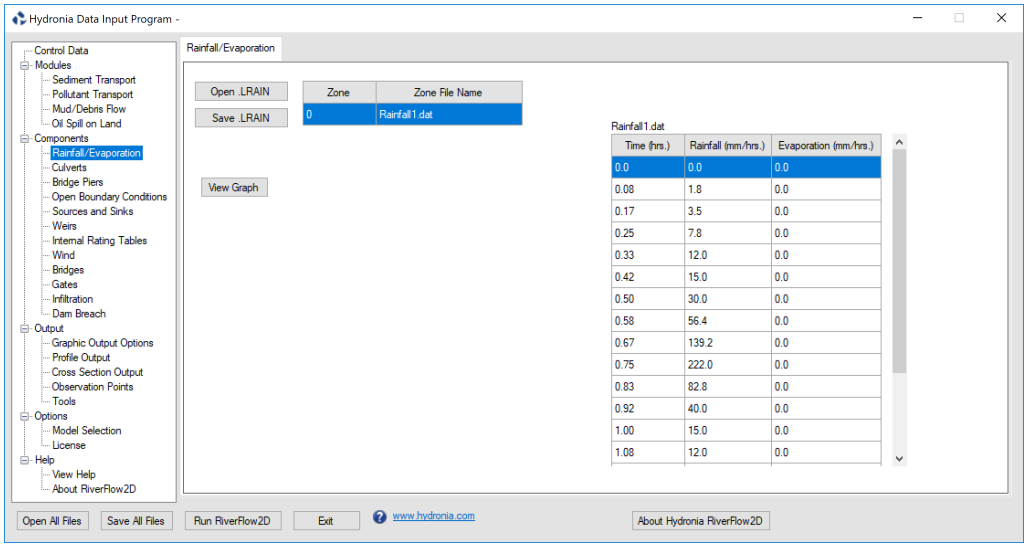


Figure 9.14 – Rainfall/Evaporation component data panel.

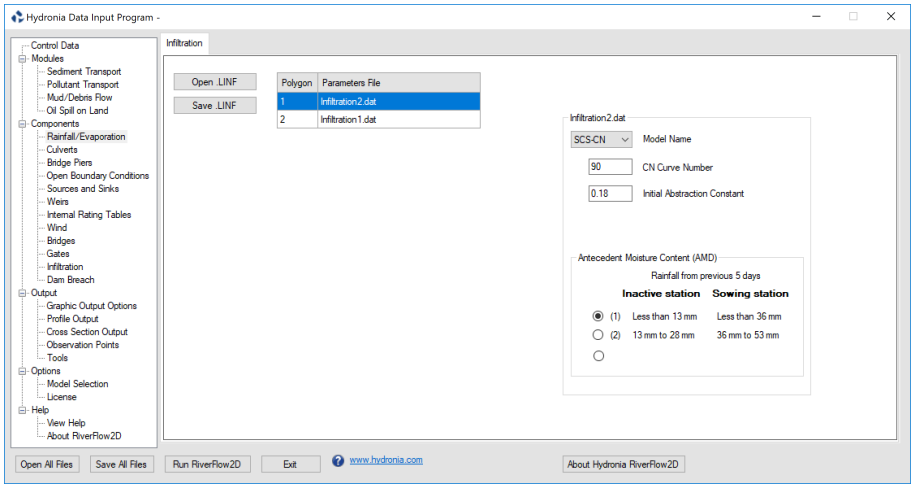


Figure 9.15 – Infiltration component data panel.

2. Leave all other parameters at their default values.

3. To run the model, click on the *Run RiverFlow2D* button. A window will appear indicating that the model began to run. The window also report the simulation time, volume conservation error, total input and output discharge, and other parameters as the execution progresses (Figure 9.16).

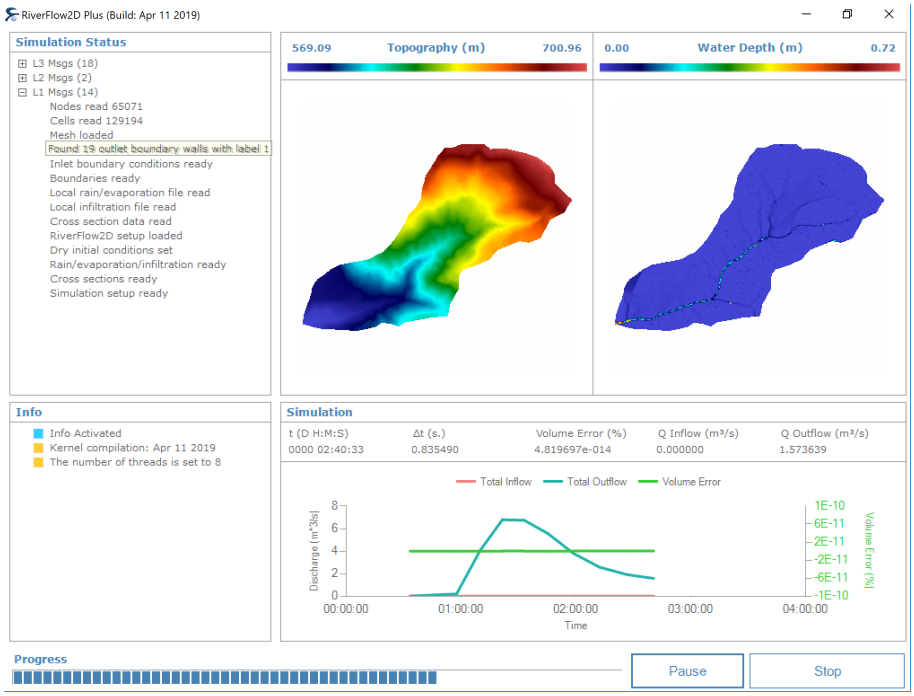


Figure 9.16 – RiverFlow2D output graphics.

9.9 Review the output files

The outflow hydrograph can be visualized using the cross section tool as shown in Figure 9.17:

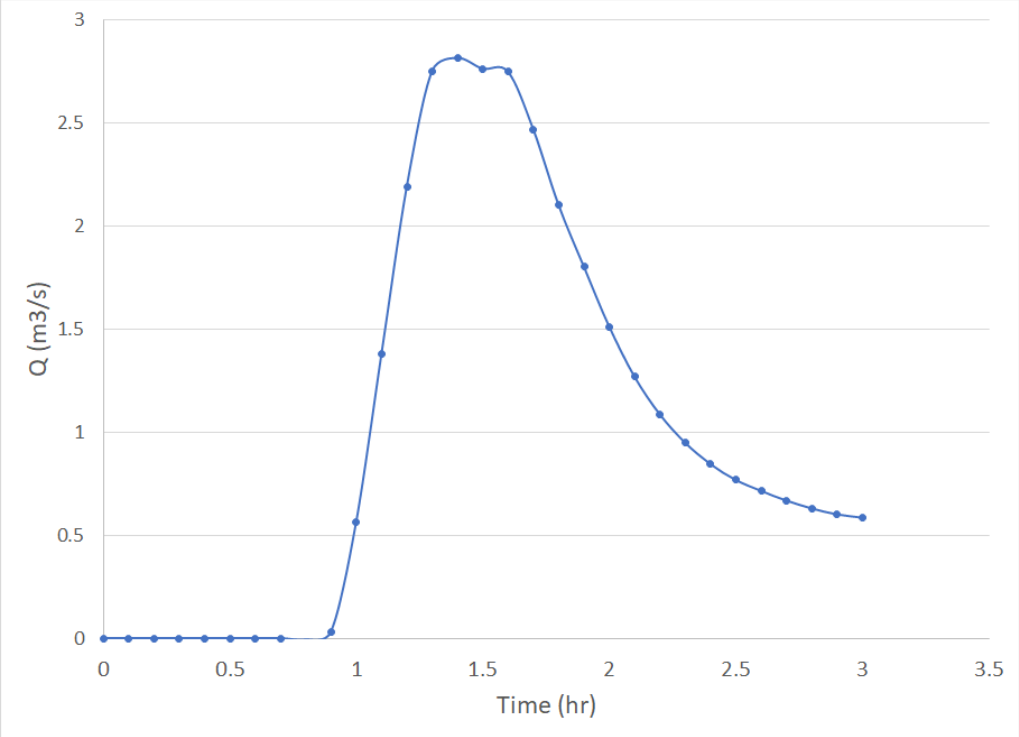


Figure 9.17 – Output hydrograph of output.

This concludes the *Hydrologic simulations* tutorial.

10 — Post-processing calculations

HydroBID Flood has three output controls that make it easier for the user to analyze the results of the runs at specific sites in the domain calculator. These output controls are: Observation points, Cross sections and Profiles.

This tutorial illustrates how to incorporate the output controls in a model using the QGIS interface. The procedure includes the following steps:

1. Open an existing HydroBID Flood project.
2. Create *ObservationPoints*, *CrossSections* and *Profiles* layers, and draw the output controls.
3. Generate the mesh.
4. Running the model.
5. Review output files.

The files required to follow this tutorial can be found in the following directory:

`...\Documents\RiverFlow2D_QGIS\ExampleProjects\outControlTutorial`

10.1 Open an existing project

1. Open QGIS
2. On the *Project* menu click *Open...* and browse to the existing project: `OutControlTutorial.qgs`.

   This project contains the layers of the domain contour, the Digital Elevation Model DEM of the river bed in raster format, the layer with the boundary conditions where inflow is located in the upper left and outflow in the lower left. The boundary conditions are a hydrograph with a peak discharge of 6,500 ft^3/s and outflow condition set to *Free outflow*. When you open the project you will have an image of the project loaded in QGIS as shown in Figure 10.1.

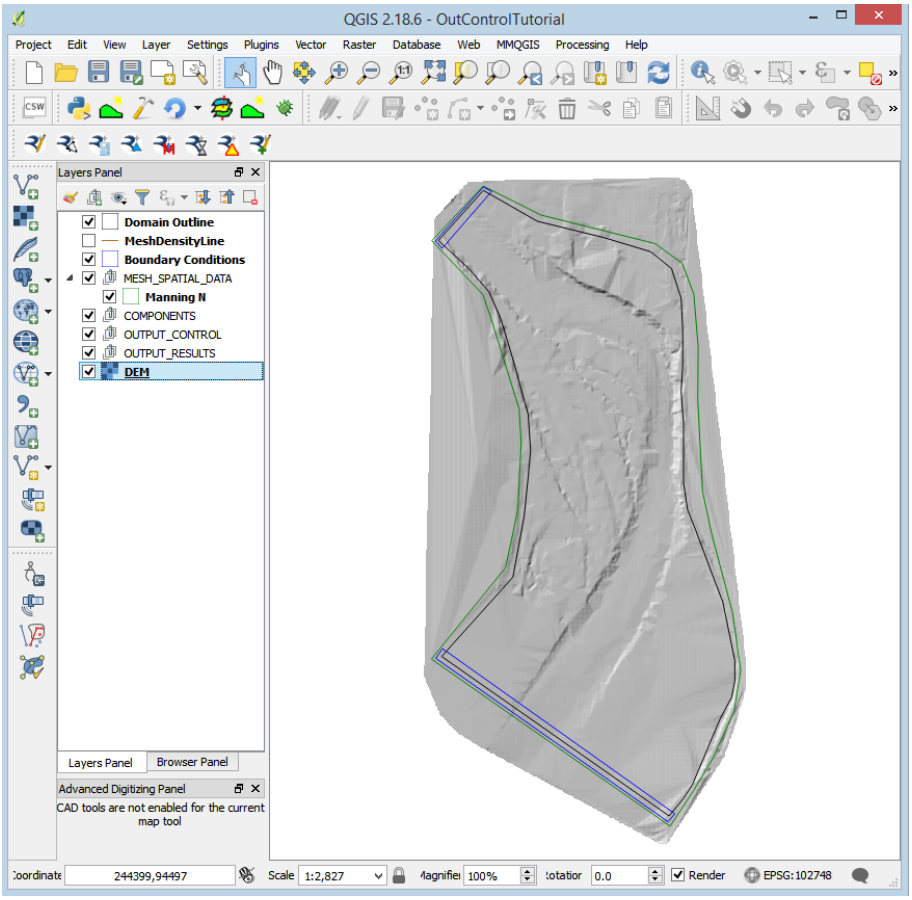


Figure 10.1 – Project screen loaded in QGIS.

10.2 Create a template of the layers *ObservationPoints*, *CrossSections* and *Profiles* and draw the output controls

To add the templates where the different output controls are drawn involves the following steps:

1. Create the templates of the layers *ObservationPoints*, *CrossSections* and *Profiles*: for this in the HydroBID Flood toolbar click on the *New Template Layer* button

2. In the plugin window activate the checkBox *ObservationPoints*, *CrossSections* and *Profiles*, as shown in the Figure below:

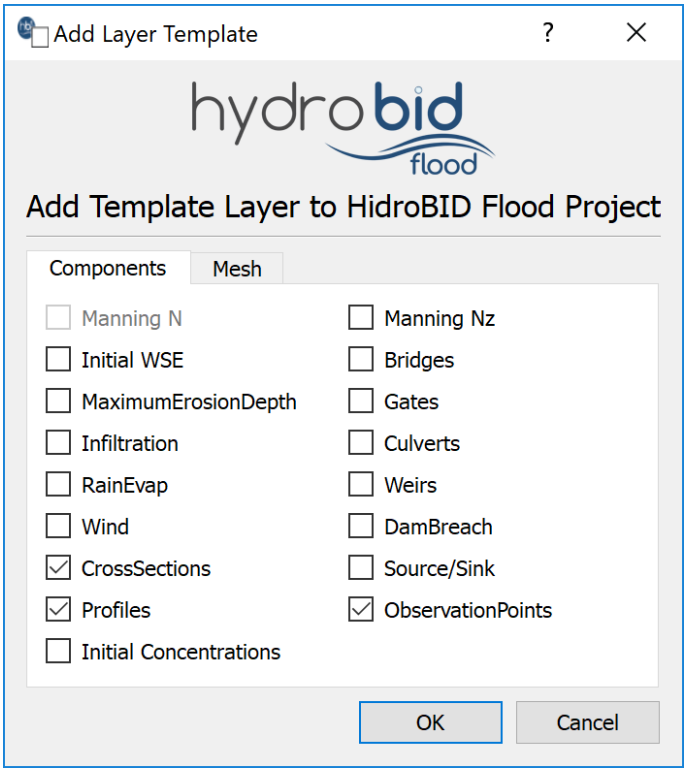
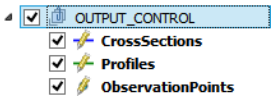


Figure 10.2 – Plugin to add the new templates layer.

3. Edit the layers and draw the output controls: Select in the layers panel, the *Observation-Points*, *CrossSections* and *Profiles* layers one by one.

4. Click on the *Toggle Editing* button:

   A pencil will appear on the label of the layers, indicating that the layers are in edit mode:

5. Draw the lines or points that represent the output control: To draw the cross sections, profiles or observation points, the *Add Feature* tool will be used.

   for the *CrossSections* and *Profiles* layers, the icon for the *Add Feature* button is

   in the case of a point layer like *ObservationPoints*, the icon is

6. Drawing the cross sections: Select the *CrossSection* layer, and activate the *Add Feature* button.

7. Proceed to draw three sections: One at the beginning of the channel, another in the middle and the third almost at the end of the channel, identify (XSECID) as: Section1, Section2 and Section3, with intervals (ND_CS) of 10, 15 and 20 respectively. The attribute tables of the sections will be as shown in Figure 10.3 and at the end of the drawing a similar image should appear as shown in the following Figure 10.4.

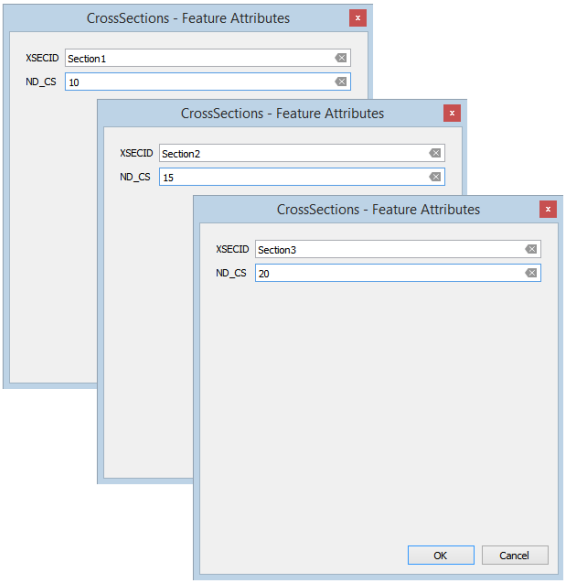


Figure 10.3 – Attribute windows for the three cross sections.

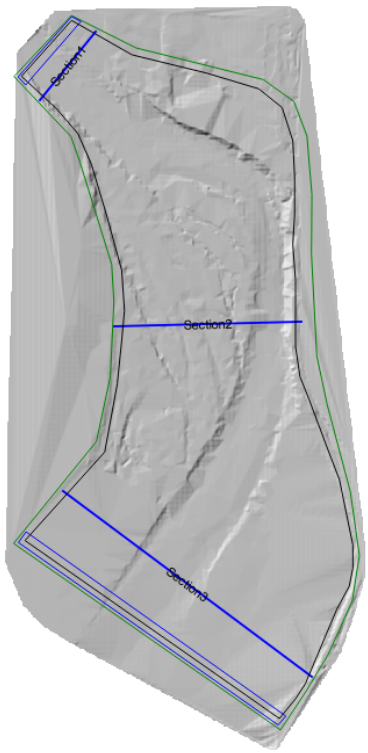


Figure 10.4 – Transversal sections incorporated into the model.

8. Save the polygon by clicking the Save button

and click on *Toggle Editing* button

to deactivate Edit mode on the CrossSections layer.

9. Drawing the Profile: Select the Profile layer and activate the Add Feature button, we proceed to draw the profile along the channel central axis, identifier (PROFILEID) is Profile1 and the number of intervals (ND_PR) equal to 50. The attribute table will be as shown in Figure 10.5. Once finished drawing, it should appear like the one shown in the following Figure 10.6.

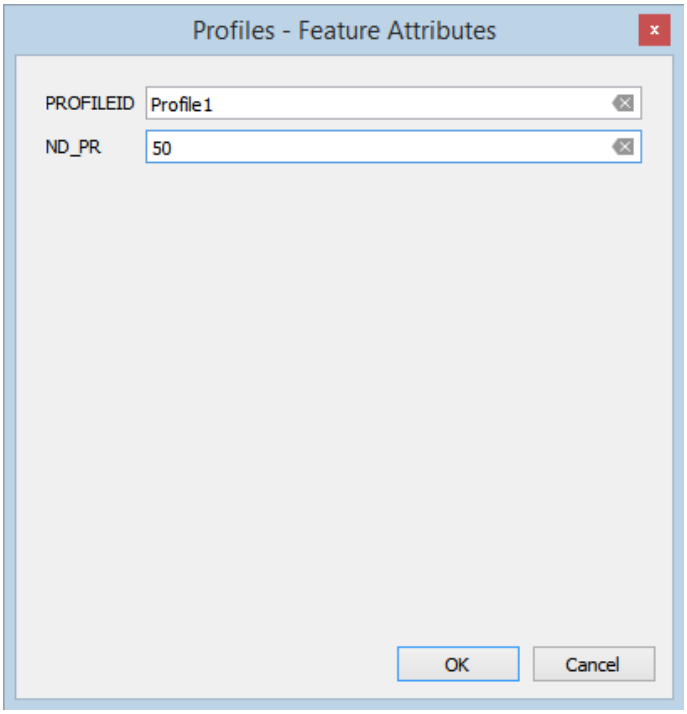


Figure 10.5 – Attribute window for the profile.

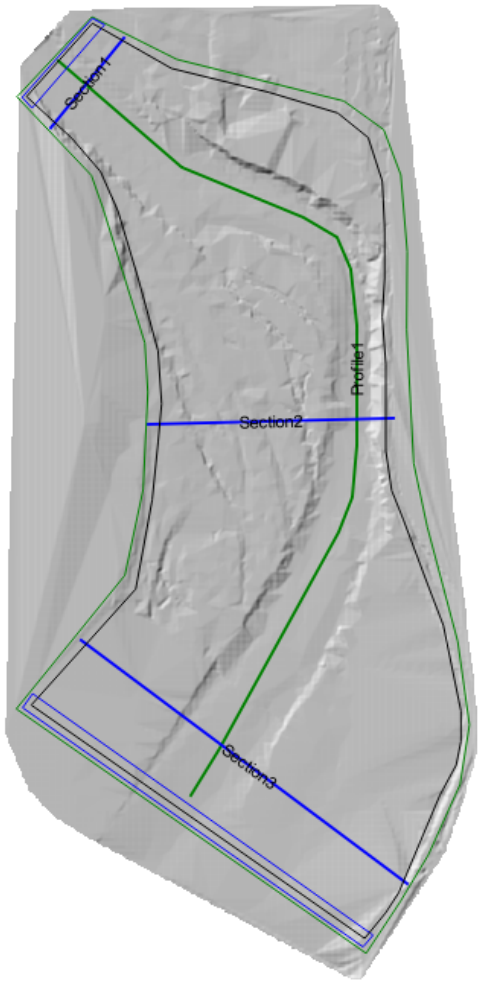


Figure 10.6 – Profile (green line) incorporated into the model.

10. To finalize the profile drawing, save the polygon by clicking the Save button

and click on *Toggle Editing* button

to deactivate Edit mode on the Profile layer.

11. Drawing the observation points: Select the *ObservationPoints* layer, and activate the *Add Feature* button, proceed to draw two observation points, the first between sections 1 and 2 and the second between sections 2 and 3. As an identifier, (Obsid) is assigned Point1 and Point2 respectively. The attribute tables will be as shown in Figure 10.7 and at the end of the drawing you should have an image similar to the one shown in the following Figure 10.8.

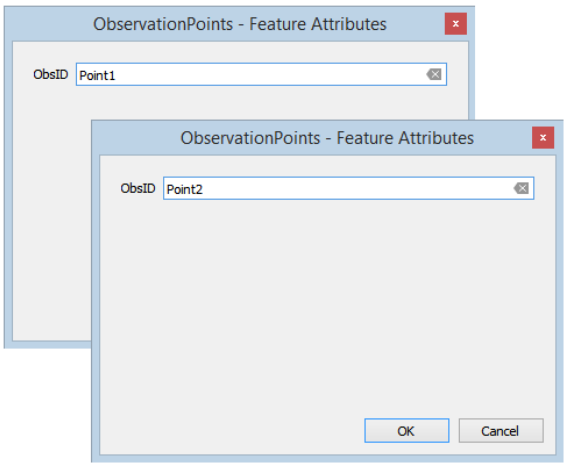


Figure 10.7 – Attribute windows for the two observation points.

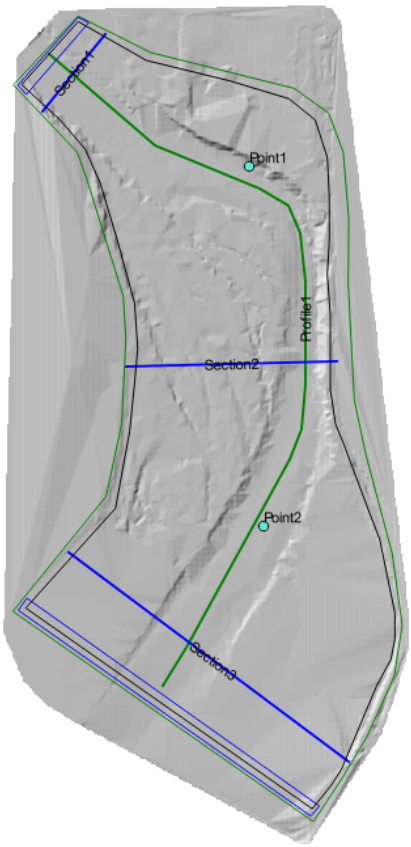


Figure 10.8 – Observation points incorporated into the model.

12. To finish the drawing of the observation points, you click again on the *Toggle Editing* button to disable the editing mode of the *ObservationPoints* layer.

10.3 Generate the mesh

The mesh is generated with the *Generate TriMesh* button

The result is a mesh of approximately 9,400 cells, as shown in Figure 10.9.

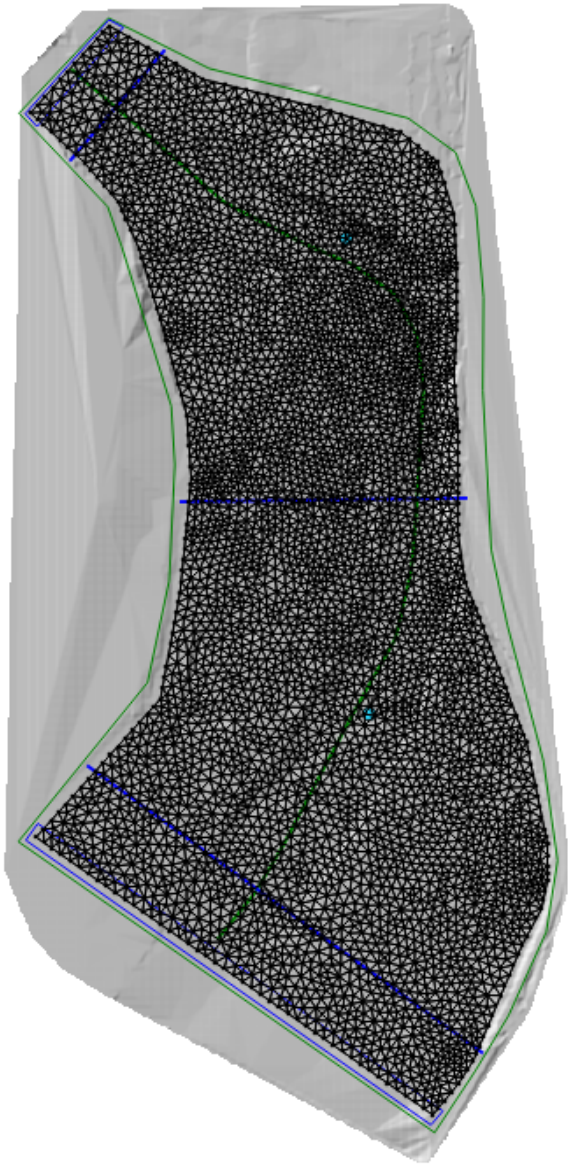


Figure 10.9 – The mesh generated.

10.3.1 Exporting files to HydroBID Flood

Now that the mesh is generated and the other layers are ready with the necessary data, export the files in the format required by HydroBID Flood.

1. Click on the *Export HydroBID Flood* button

2. When run the plugin a window is displayed, select the raster layer that contains the Digital

Elevation Model (DEM) and the name of the project to be exported.

3. Input the name without any extension. For this example it will be: OutControl.

4. Before executing the plugin, activate the layer with the DEM (if it is deactivated).

   Once the plugin is executed, a window will be shown (Figure 10.10), as it should be for our example.

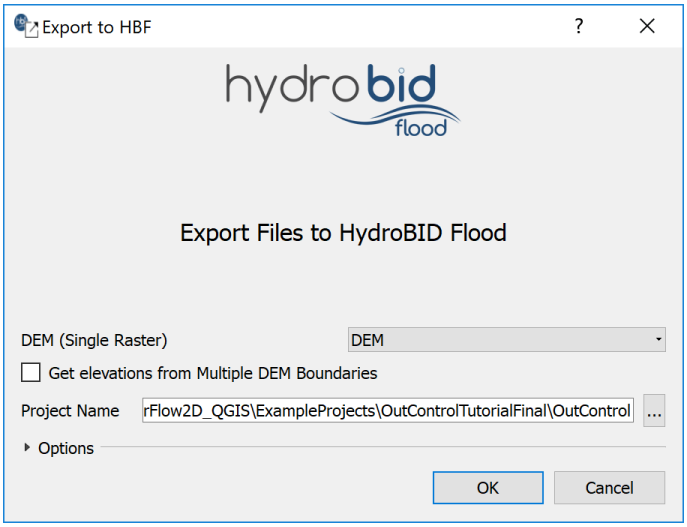


Figure 10.10 – Plugin to export the files to HydroBID Flood.

5. Once finished inputting the information, click on the OK button and the export process will begin.

Once it is finished, HydroBID Flood will be loaded with the `OutControl.DAT` file.

10.4 Running the Model

After exporting the files, HydroBID Flood opens with the project file of the `OutControl.DAT` sample and shows the *Control Data* panel to it as illustrated in Figure 10.11.

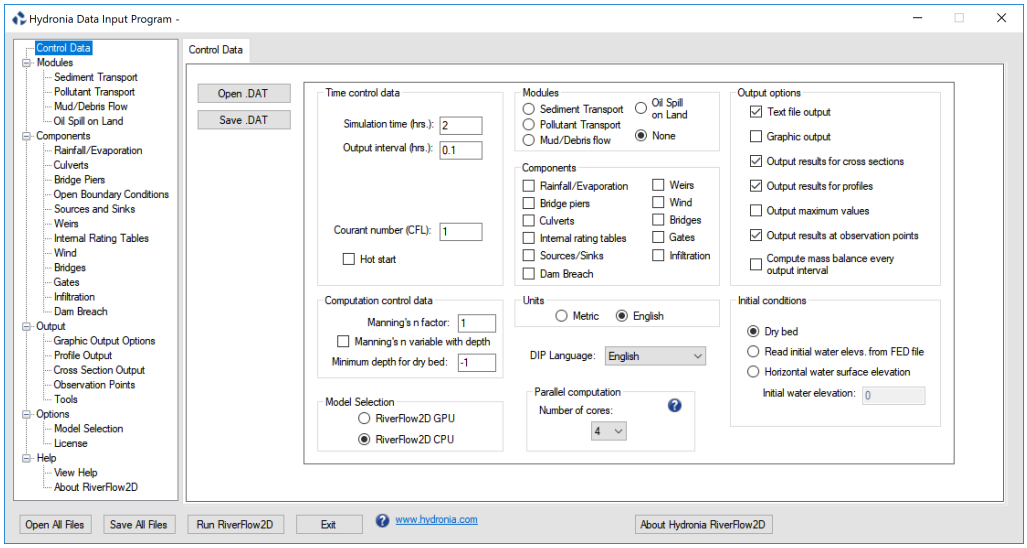


Figure 10.11 – Control data panel.

You can observe in the control panel in Output Options the outputs of results for cross sections, profiles and observation points are selected.
Leave all other parameters at their default values.

To run the model, click on the *Run RiverFlow2D* button. The window that HydroBID Flood presents while running the model shows: simulation time information, volume conservation error, total discharge of inflow in and outflow, as well as other parameters as execution progresses (Figure 10.12).

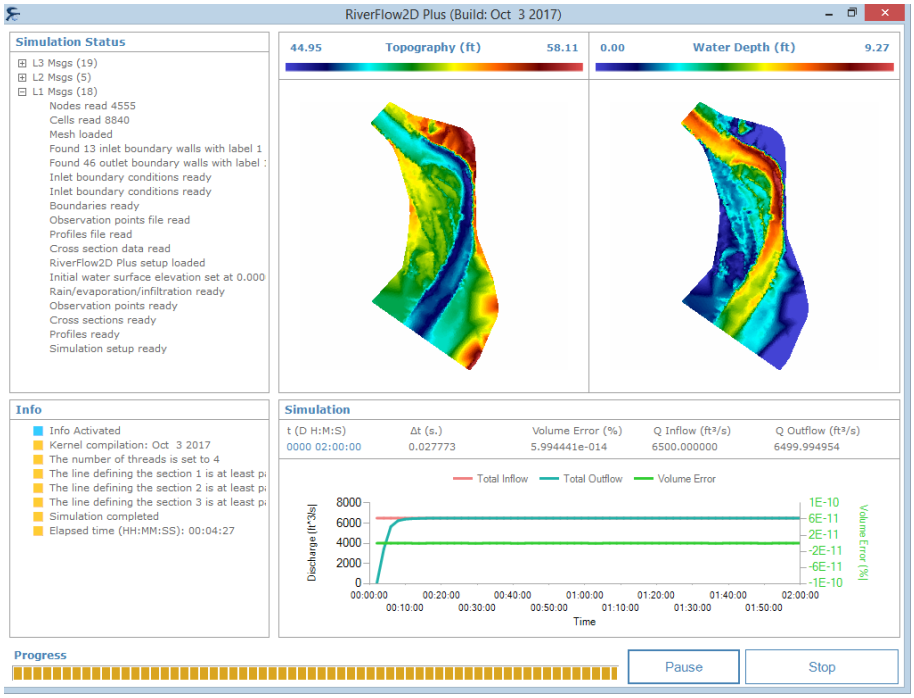


Figure 10.12 – River Flow2D output graphics.

10.5 Review output files

HydroBID Flood generates the files with the extensions `.xseci` (metric units) and `.xsece` (English units) which report the results along the cross sections. it generates files with extensions `.prfi` (metric units) and `.prfe` (english units) which report the results along the profiles and generates the files with the extensions `.outi` (metric units) and `.oute` (English units) that report the results in the observation points.

Figure 10.13 shows an extract of the `OutControl.xsece` file with results at the cross sections:

```
================================================================================
        RiverFlow2D Plus - Release 4.2.1
                           Build Oct  3 2017
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                       (C) COPYRIGHT 2009-2017 Hydronia, LLC.
                                     ALL RIGHTS RESERVED
                                     RUN DATE: 23/Jan/2018
================================================================================

 CROSS SECTION RESULTS IN ENGLISH UNITS

 TIME: 0000 days, 00 hours, 00 min.,00 secs.


 CROSS SECTION NO.:   1 CROSS SECTION ID: Section1

 (     243555.07,       94409.01), (     243437.29,       94264.25)
 ELEM    STATION    BEDEL     DEPTH     WSEL     VELOCITY  FROUDE     QS
           (ft)      (ft)      (ft)      (ft)      (ft/s)           (ft2/s)
  3824     20.74     52.26      0.00     52.26      0.00      0.00  0.000000
  6162     41.47     52.14      0.00     52.14      0.00      0.00  0.000000
  5311     62.21     51.59      0.00     51.59      0.00      0.00  0.000000
  6477     82.94     49.20      0.00     49.20      0.00      0.00  0.000000
  1132    103.68     48.73      0.00     48.73      0.00      0.00  0.000000
   788    124.42     49.10      0.00     49.10      0.00      0.00  0.000000
  1792    145.15     49.51      0.00     49.51      0.00      0.00  0.000000
  1953    165.89     51.61      0.00     51.61      0.00      0.00  0.000000
 Q =      0.000 cfs.


 CROSS SECTION NO.:   2 CROSS SECTION ID: Section2

 (     243989.36,       93795.60), (     243594.33,       93785.79)
 ELEM    STATION    BEDEL     DEPTH     WSEL     VELOCITY  FROUDE     QS
           (ft)      (ft)      (ft)      (ft)      (ft/s)           (ft2/s)
  2897     28.23     50.65      0.00     50.65      0.00      0.00  0.000000
  8304     56.45     46.40      0.00     46.40      0.00      0.00  0.000000
  2483     84.68     48.42      0.00     48.42      0.00      0.00  0.000000
  3240    112.90     50.78      0.00     50.78      0.00      0.00  0.000000
```

Figure 10.13 – OutControl.XSECE file.

Figure 10.14 shows an extract of the `OutControl.prfe` file with the report of the profile results:

```
======================================================================
======================================================================
        RiverFlow2D Plus - Release 4.2.1
                           Build Oct  3 2017
======================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                        (C) COPYRIGHT 2009-2017 Hydronia, LLC.
                                            ALL RIGHTS RESERVED
                                            RUN DATE: 23/Jan/2018
======================================================================

 PROFILE RESULTS IN ENGLISH UNITS

 TIME: 0000 days,00 hours,00 min.,00 secs.

 PROFILE NO.:   1 PROFILE ID: Profile1

 ELEM   DISTANCE    BEDEL     DEPTH      WSEL    VELOCITY  FROUDE
          (ft)      (ft)       (ft)      (ft)     (ft/s)
  3736      0.00     48.61      0.00     48.61      0.00      0.00
  6466     32.83     49.13      0.00     49.13      0.00      0.00
  3156     65.67     48.77      0.00     48.77      0.00      0.00
    82     98.50     48.78      0.00     48.78      0.00      0.00
  1122    131.34     48.57      0.00     48.57      0.00      0.00
  1971    164.17     48.60      0.00     48.60      0.00      0.00
  4933    197.01     48.94      0.00     48.94      0.00      0.00
   372    229.84     48.76      0.00     48.76      0.00      0.00
  2028    260.18     48.47      0.00     48.47      0.00      0.00
  3745    290.51     48.46      0.00     48.46      0.00      0.00
  6980    320.84     48.57      0.00     48.57      0.00      0.00
  1804    351.18     47.72      0.00     47.72      0.00      0.00
```

Figure 10.14 – OutControl.PRFE file.

Figure 10.15 shows an extract of the `RESvsT_0004938.oute` file with the report of the results of the observation point. Point1:

```
================================================================================
================================================================================
        RiverFlow2D Plus - Release 4.2.1
                           Build Oct  3 2017
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                        (C) COPYRIGHT 2009-2017 Hydronia, LLC.
                                      ALL RIGHTS RESERVED
                                      RUN DATE: 23/Jan/2018
================================================================================

 RESULTS FOR CELL:     4938 OBSERVATION POINT ID: Point1
   LOCATED AT COORDINATE: (   243824.97),(   243824.97)

      TIME         U         V     VELOCITY    DEPTH      WSEL    BEDEL_ORI    BEDEL    DELTA_BED  FROUDE
     (hours)    (ft/s)     (ft/s)   (ft/s)      (ft)      (ft)      (ft)       (ft)       (ft)
     0.01000     7.957    -4.306     9.048     3.660    50.511    46.851     46.851     0.000     0.834
     0.02000     4.590    -2.370     5.165     6.293    53.144    46.851     46.851     0.000     0.363
     0.03000     3.037    -1.391     3.341     7.128    53.979    46.851     46.851     0.000     0.221
     0.04000     2.351    -1.040     2.571     7.435    54.286    46.851     46.851     0.000     0.166
     0.05000     2.126    -0.915     2.315     7.609    54.460    46.851     46.851     0.000     0.148
     0.06000     2.024    -0.879     2.207     7.720    54.571    46.851     46.851     0.000     0.140
     0.07000     1.946    -0.834     2.117     7.800    54.651    46.851     46.851     0.000     0.134
     0.08000     1.900    -0.813     2.066     7.857    54.708    46.851     46.851     0.000     0.130
     0.09000     1.875    -0.794     2.036     7.892    54.743    46.851     46.851     0.000     0.128
     0.10000     1.847    -0.781     2.006     7.921    54.772    46.851     46.851     0.000     0.126
     0.11000     1.830    -0.771     1.986     7.939    54.790    46.851     46.851     0.000     0.124
     0.12000     1.819    -0.766     1.974     7.950    54.801    46.851     46.851     0.000     0.123
     0.13000     1.813    -0.763     1.967     7.960    54.811    46.851     46.851     0.000     0.123
     0.14000     1.809    -0.761     1.963     7.965    54.816    46.851     46.851     0.000     0.123
     0.15000     1.804    -0.760     1.957     7.967    54.818    46.851     46.851     0.000     0.122
```

Figure 10.15 – RESvsT_0004938.oute file.

11 — Advanced digitization/Snapping

In many cases the drawing of polygons that share borders (contiguous) is required. Trying to do this manually usually produces polygon overlap errors, or it may generate gaps as illustrated in the following figure:

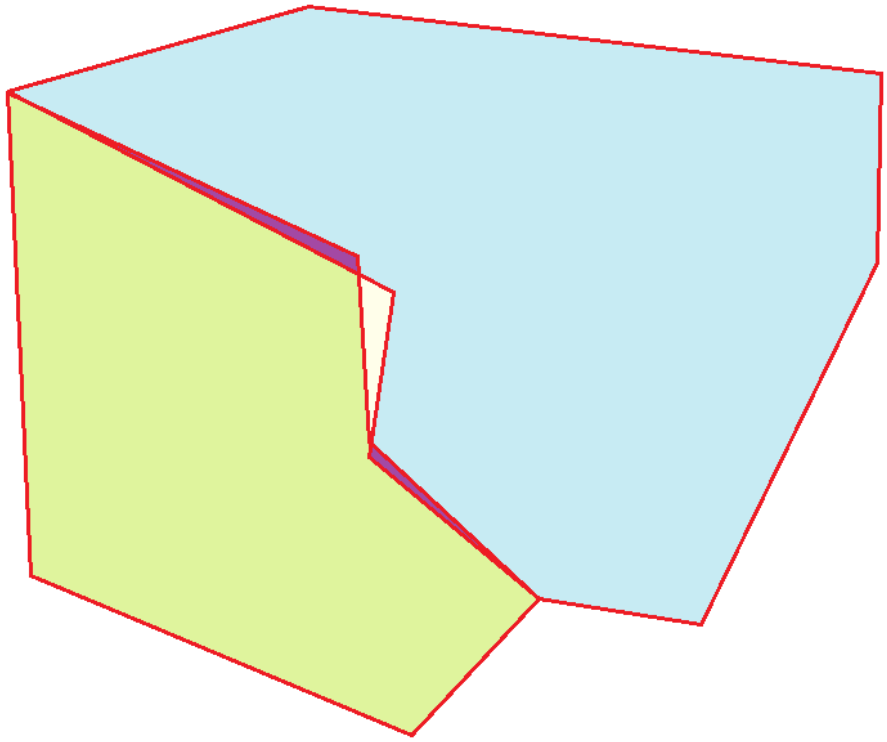


Figure 11.1 – Areas with overlap and empty spaces in manual digitization of adjacent polygons.

This tutorial illustrates how to use the QGIS Snapping tool to facilitate the scanning of adjacent polygons. The procedure includes the following steps:

1. Open an existing HydroBID Flood project.
2. Activate the *Snapping* tool.
3. Configure the *Snapping* tool

4. Draw contiguous or adjacent polygons using the *Snapping* tool.

The files required to follow this tutorial can be found in the following directory:

`...\Documents\RiverFlow2D_QGIS\ExampleProjects\SnappingTutorial`

11.1 Open an existing project

1. Open QGIS

2. In the main menu go to *Project* → *Open...* browse to the existing project: `SnappingTutorial.qgs`.

This project contains the basic templates of a HydroBID Flood project and a raster with 4 areas with different Mannings n coefficients derived from land cover and land use maps. From this raster, the polygons of the *Manning N* layer will be drawn. When the project is opened, a project image will be loaded in QGIS as shown in Figure 11.2.

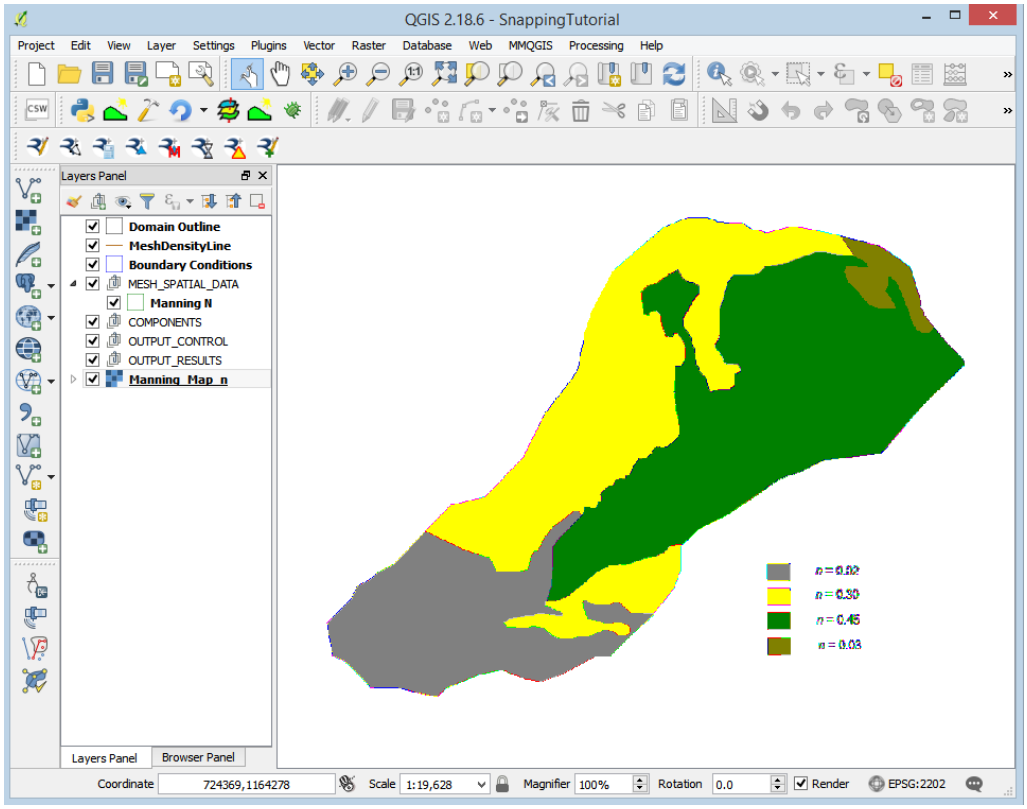


Figure 11.2 – Project screen loaded in QGIS.

11.2 Activate the Snapping tool

To have the Snapping tool, you must activate it beforehand:

1. Open the configuration options window: In main menu → Settings → Options...
2. In the Settings window, click on the Digitizing tab.
3. In Snapping, activate the option *Open snapping options in dock window* (Figure 11.3). As indicated by the label
4. After accepting the changes, exit and restart QGIS.

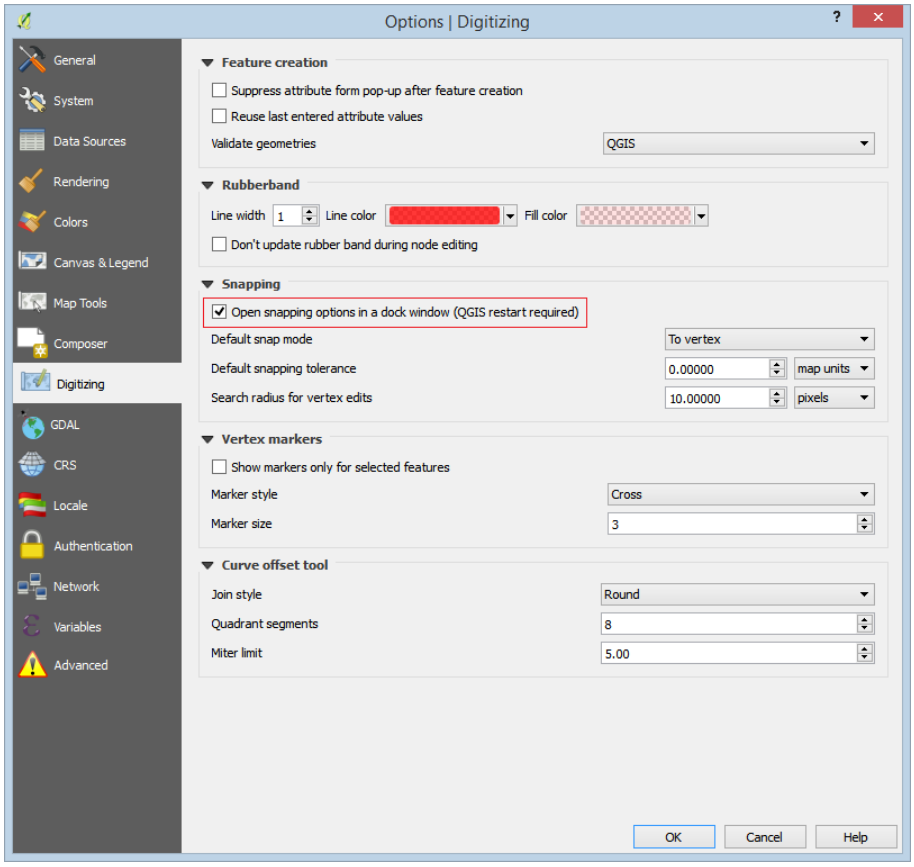


Figure 11.3 – QGIS Options / Digitizing Window.

Once you restart QGIS and open the project again you should have an image similar to the one shown in Figure 11.4 where you can see the Snapping tool panel at the bottom. If it does not appear, Open the main menu → Configuration → Snapping options..., to load panel.

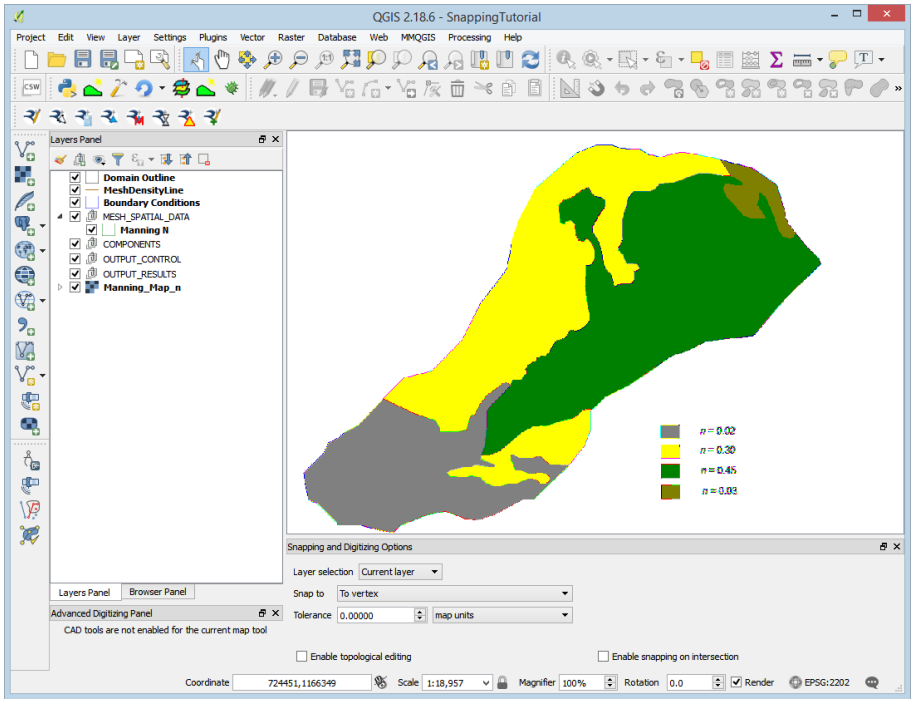


Figure 11.4 – QGIS with snapping panel.

11.3 Configure the snapping tool

Before starting to draw the Manning's n polygons, the snapping tool should be configured as follows:

1. In Layer selection, chose *Advanced*
2. In the list of layers, you unselect all but the layer where the polygons will be drawn. In this case the *Manning N* layer is selected.
3. Select the box to avoid intersections in the *Manning N* layer.

The configuration window of the Snapping tool should be as shown in the following Figure:

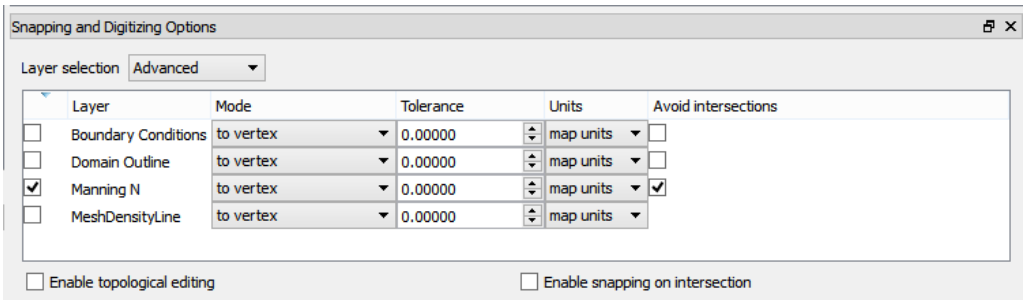


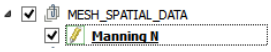
Figure 11.5 – Snapping configuration panel.

11.4 Draw contiguous or adjacent polygons using the Snapping tool

To draw the polygons with the information of the Manning's n coefficients we proceed as follows:

1. Edit the *Manning N* layer: In the layers panel, select the *Manning N* layer
2. In the digitalization toolbar click on the *Toggle Editing* button

   A pencil icon will appear in the *Manning N* layer indicating that the layer is in edit mode:

3. Draw the polygon that demarcates the Manning's n area: Using the *Add Toggle Editing* button in the digitalization toolbar

   proceed to draw the polygons. Start by drawing the polygon in the bottom of the watershed shown in gray which corresponds to n = 0.02. After completing the drawing you should have an image similar to the one shown in the following figure:

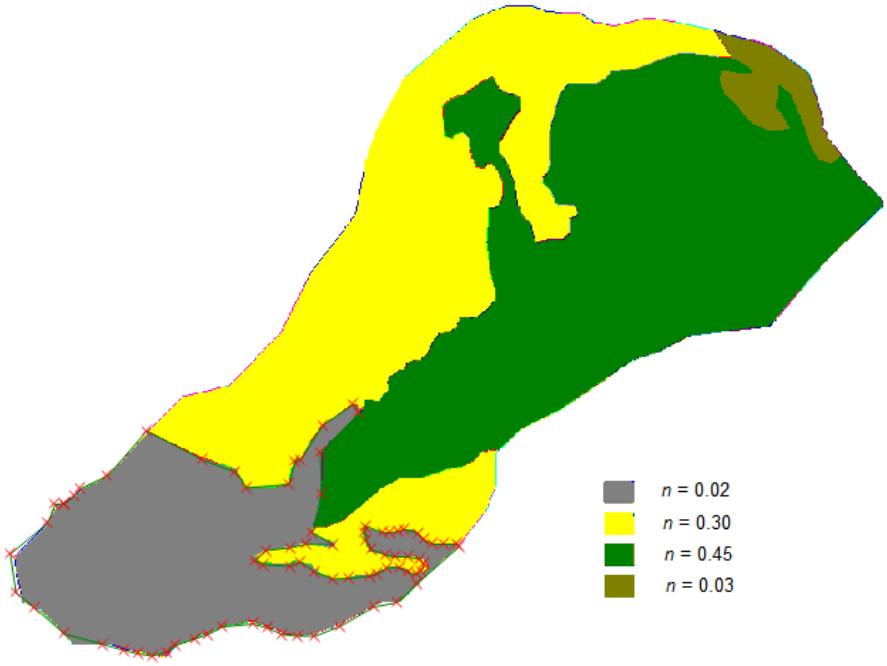


Figure 11.6 – The first polygon sketch.

4. Proceed to draw the second polygon. This will be the small yellow polygon to which corresponds to n = 0.30. To draw the polygon, follow the boundary with the green polygon as indicated by the raster, but in the boundary with the polygon already drawn (the gray) an overlap will be made so that the snapping tool takes the edge that already exists and completes the polygon. Try to make a path as shown in the figure below:

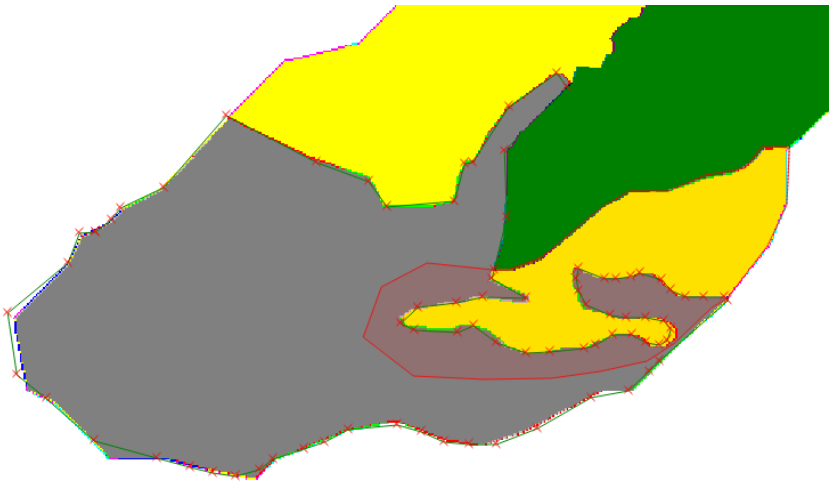


Figure 11.7 – Drawn of the second polygon.

5. Right-click to finish the drawing.

6. Enter the Manning coefficient, n = 0.30 and you will have an image like the one shown below where you can see how the final drawing of the second polygon took the vertices of the first adjacent polygon:

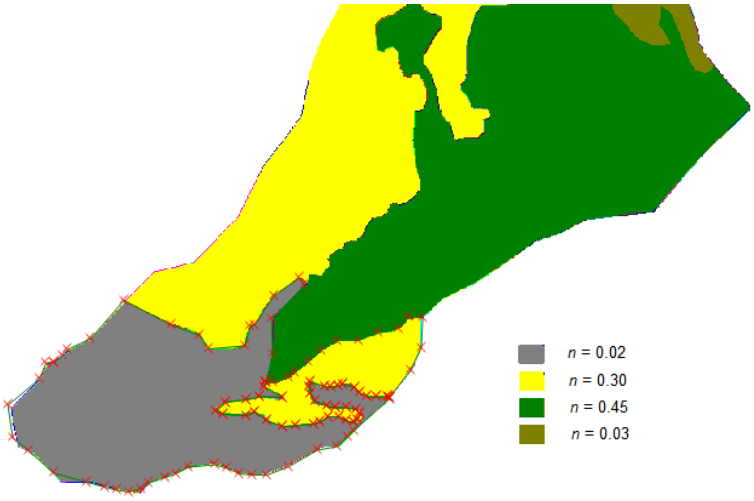


Figure 11.8 – Final draw of the second polygon.

7. The third polygon to be drawn will be the large yellow polygon. Just as in the previous case, the contour of the polygon according to the raster is faithfully followed, but in the adjacent polygons it is already drawn overlapped. The figure below shows the initial drawing, with an approach in the area where the overlap is made with the existing polygon:

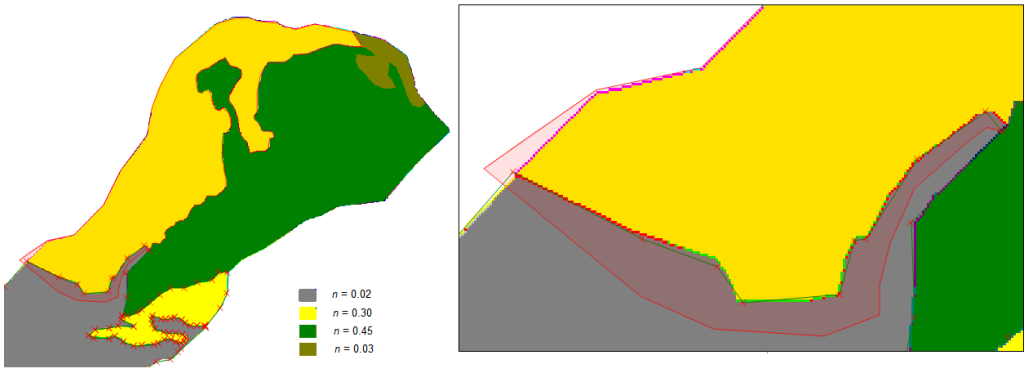


Figure 11.9 – Preliminary draw of the third polygon.

8. After finishing the drawing and assigning the *Manning N* umber to the polygon, the final drawing of the third polygon will be shown as shown below:

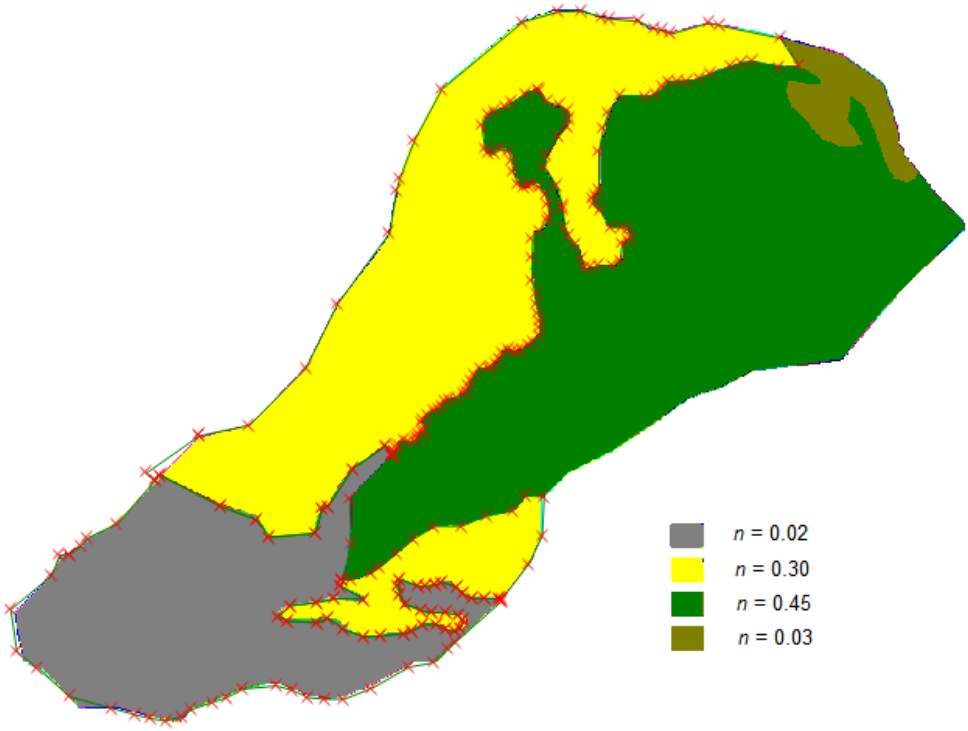


Figure 11.10 – Final draw of the third polygon.

9. The fourth polygon to draw will be the green polygon which corresponds to n = 0.45. To do it will follow the outer boundary of the same and the boundary with the brown polygon. Then a thick overlap will be made within the polygon already drawn, the yellows and the gray, as shown in the figure below:

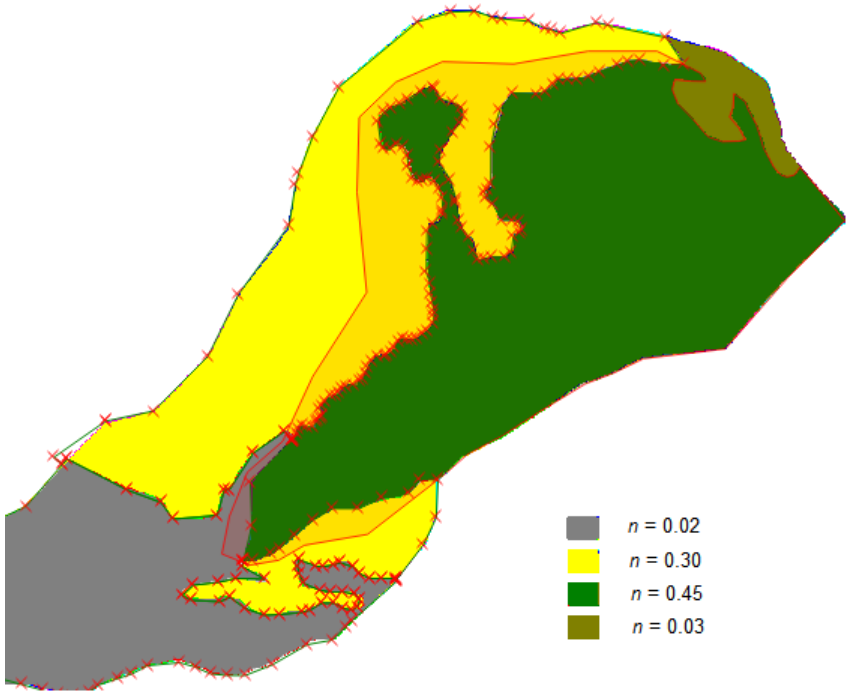


Figure 11.11 – Preliminary draw of the fourth polygon.

10. After finishing the drawing and assigning the Manning's n value to the polygon, you will have the final layout of the third polygon as shown below:

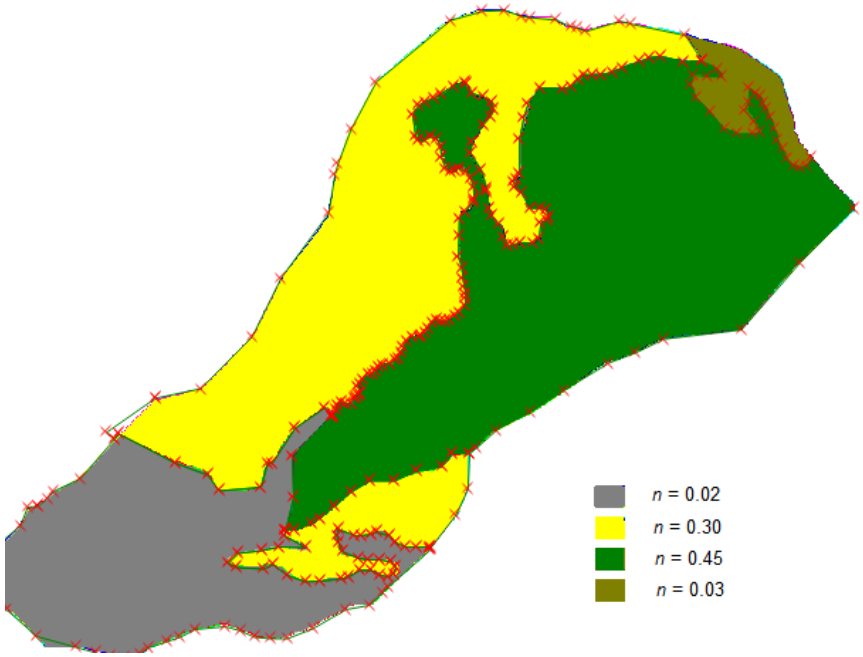


Figure 11.12 – Final draw of the fourth polygon.

11. To finish, draw the fifth polygon of brown color, which corresponds to n = 0.03. For this the outer limit is drawn following the contour of the raster and for the shared limits an overlap is made inside the neighboring polygons as shown in the figure below:

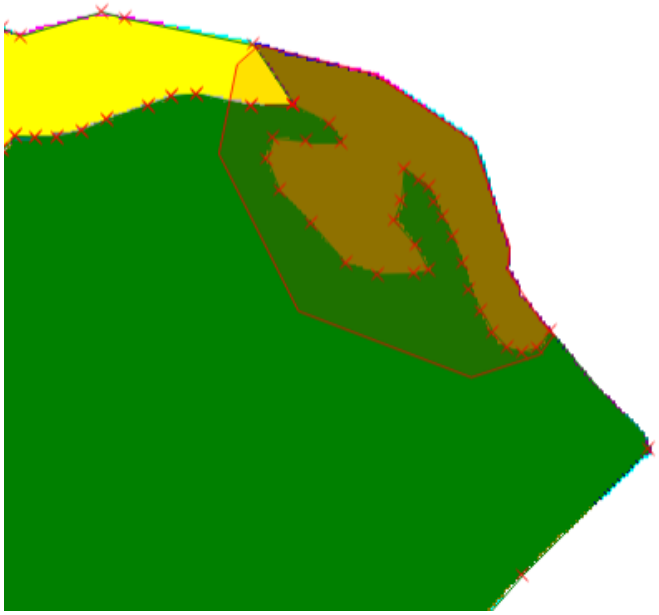


Figure 11.13 – Preliminary draw of the fifth polygon.

12. After finishing the drawing and inputting the Manning coefficient to the polygon, you will have the final drawing of the third polygon as shown below:

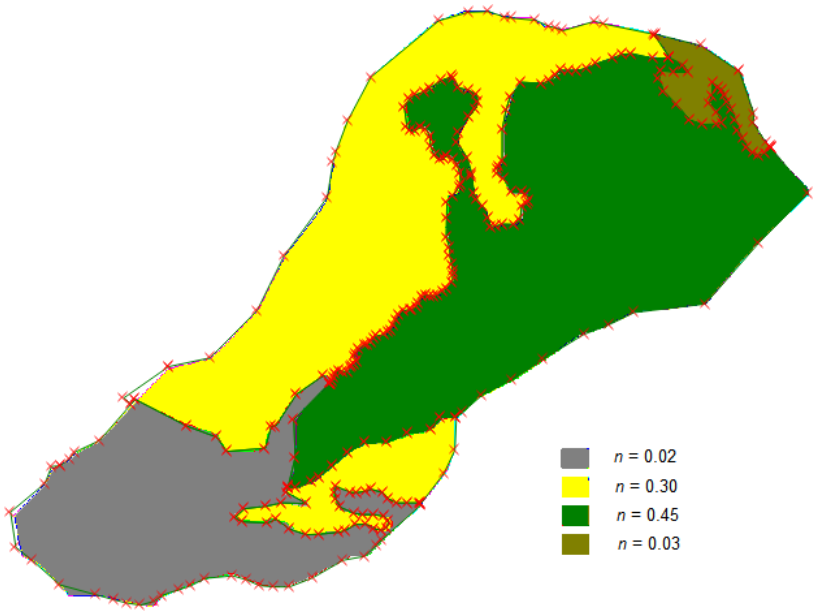


Figure 11.14 – Final draw of the fifth polygon.

13. To finish, click on the *Toggle Editing* button

changes are accepted and saved. Then the layer of the raster Manning_Map_n is deactivated, and the *Manning N* vectorized layer can be observed as shown in the Figure below:

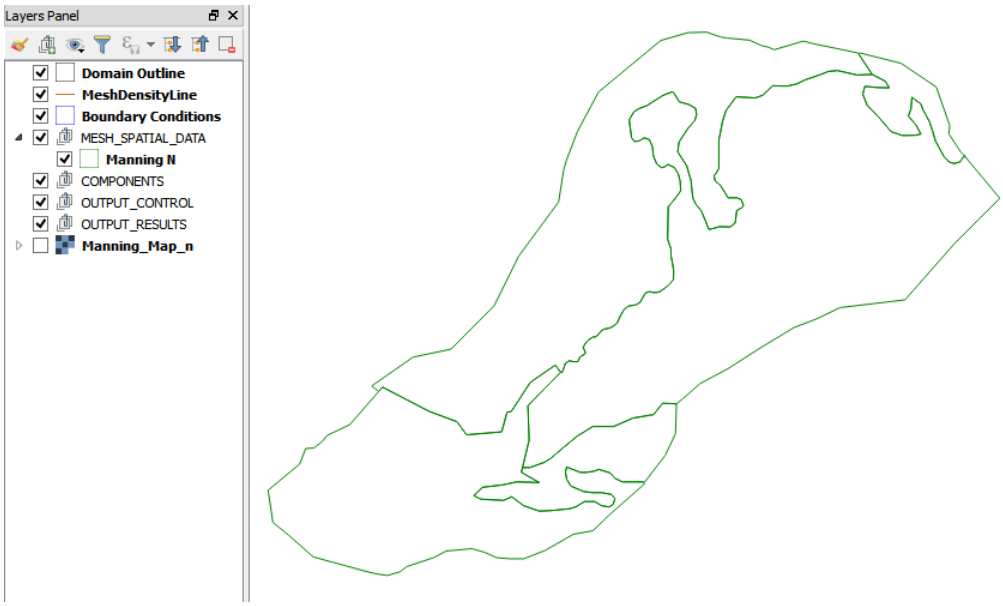
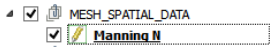


Figure 11.15 – Final draw of the *Manning N* layer.

11.5 Delete a polygon

Suppose you want to delete the last polygon created because you are not satisfied with the result. In that case you can proceed doing as follows:

1. The *Manning N* layer is edited using the *Toggle Editing* button



2. Then the polygon to be eliminated is selected using the *Select Feature* button

3. With the active select Feature tool, click on the polygon to be deleted and it will be highlighted in yellow to indicate that it is selected (Figure 11.16).

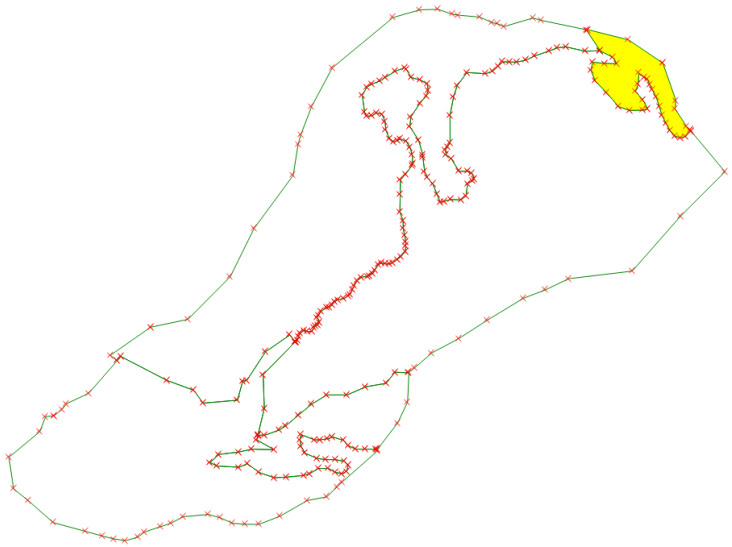


Figure 11.16 – Polygon to be deleted selected.

4. Then the polygon is deleted either with the Delete key on the keyboard or with the *Delete selected* button

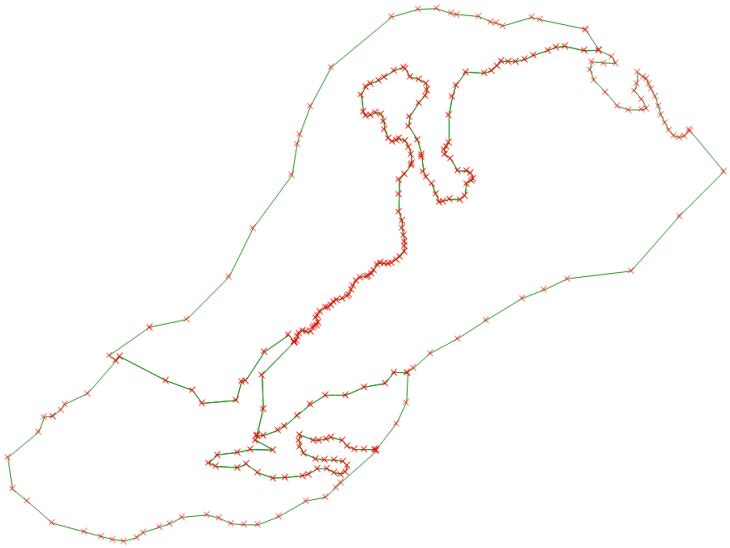


Figure 11.17 – Manning N layer with the polygon removed.

5. Finally, the changes are saved and the editing mode of the layer is deactivated by clicking on the *Toggle Editing* button.

12 — Creating raster elevations from X Y Z data sets

HydroBID Flood uses elevation data in raster format. For instance, to load an ASCII grid file, from the *Layer* menu, click *Add Layer*, and then click *Add Raster Layer...* You may also click the *Add Raster Layer* button:

In this example we will use elevation data available in a tab delimited X Y Z file, and create an interpolated raster layer.

The X Y Z file can be formatted as comma, space or tab delimited data, where the first two columns correspond to point X and Y coordinates and the third column is the point elevation Z as shown in the following figure.

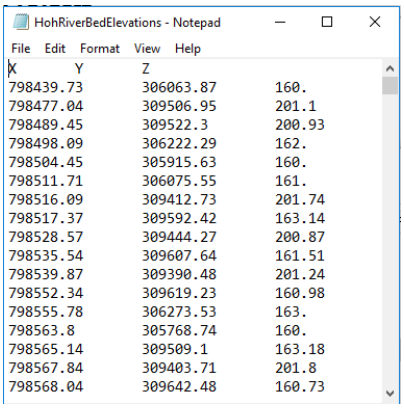


Figure 12.1 – File containing terrain elevation points.

First, the X Y Z file is loaded to create an event layer. This can be accomplished as follows:

1. From the *Layer* menu, click *Add Layer*, and then click *Add Delimited Text Layer...* Alternatively, you may also click the *Add Delimited Text Layer* button:

the following dialog is presented:

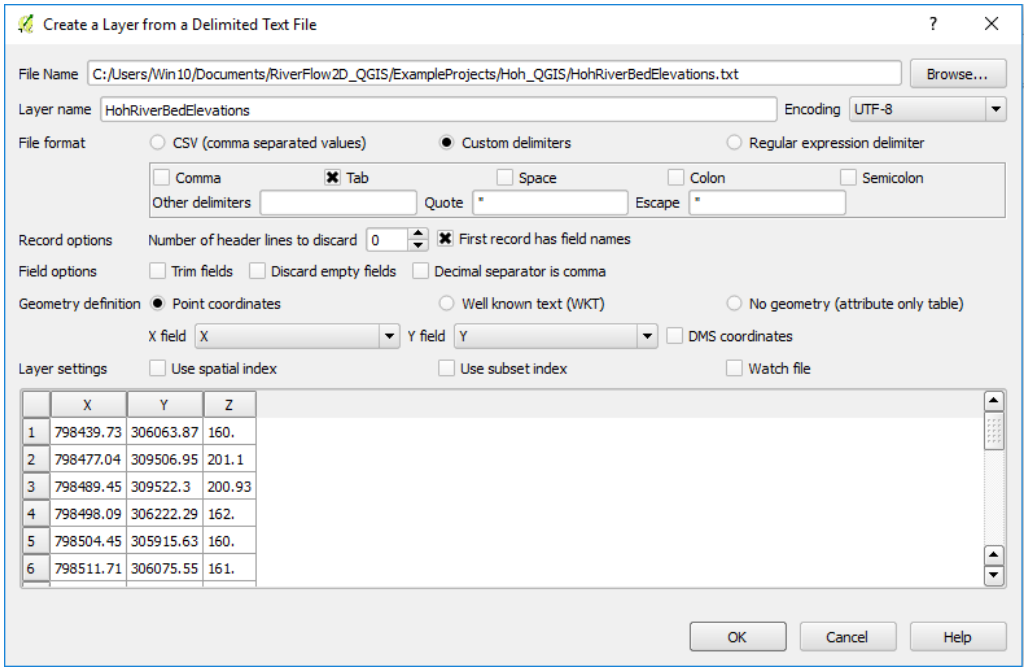


Figure 12.2 – Dialog to create a layer from delimited data text file.

2. Here you must indicate the location and name of the file with the X Y Z data.

3. Make sure that the first row has the names of the fields, and this is assigned to *X field*, *field_1* and *Y field*, *field_2*, then click OK to load the points in the project window. The Coordinate Reference System Selector dialog box appears again and we can click OK to continue making sure that the CRS is the appropriate one.

4. As surely the points are not within the extension of the window, you will have to click with the right mouse button on the label of the created layer and select the option Zoom to the layer.

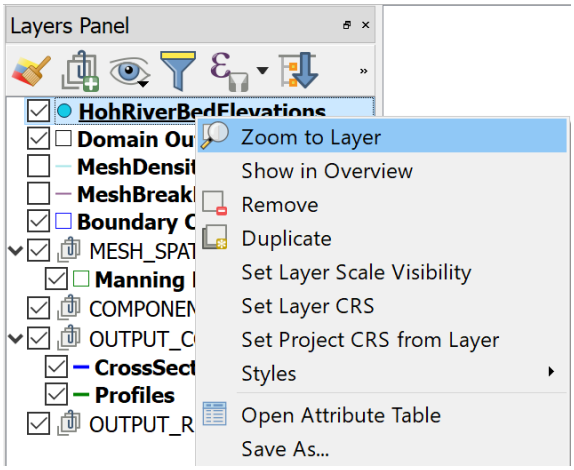


Figure 12.3 – How to zoom a layer.

5. The points should appear in the project window:

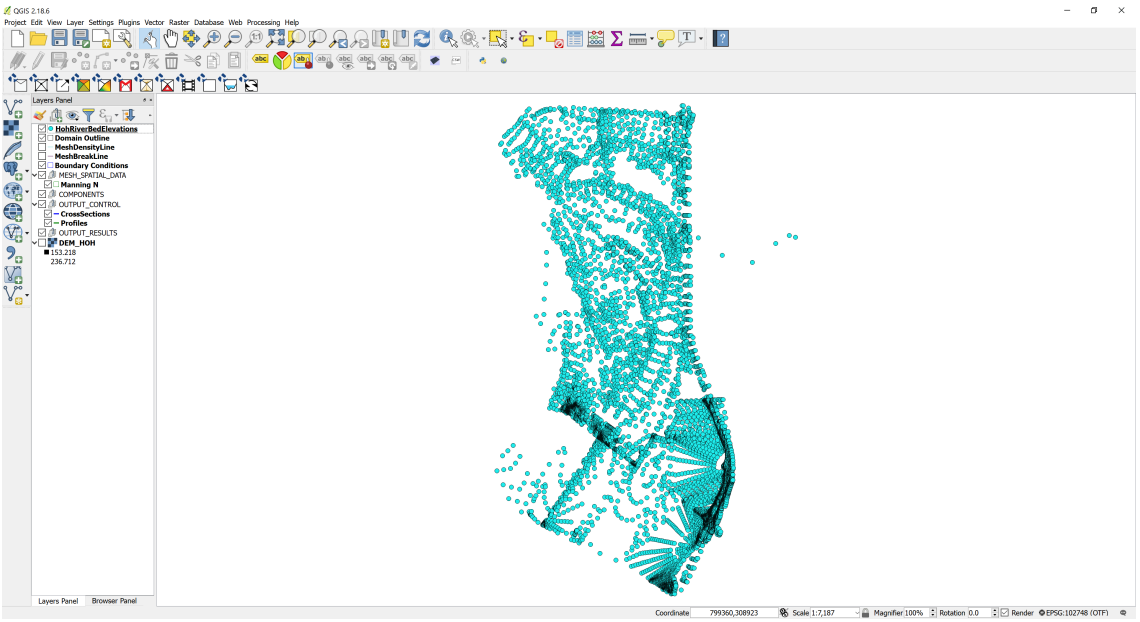


Figure 12.4 – Layer of points created from delimited data file.

6. The raster is created interpolating from the data in the point layer. For that we will use the *Interpolation* command which is found on the *Raster* menu under *Interpolation*. This command is a plugin that is installed by default in QGIS. If for any reason it does not appear, on the *Plugins* menu, click *Manage and Install Plugins...*, type *Interpolation* on the search field, and then then select the *Interpolation Plugin* checkbox to enable it.

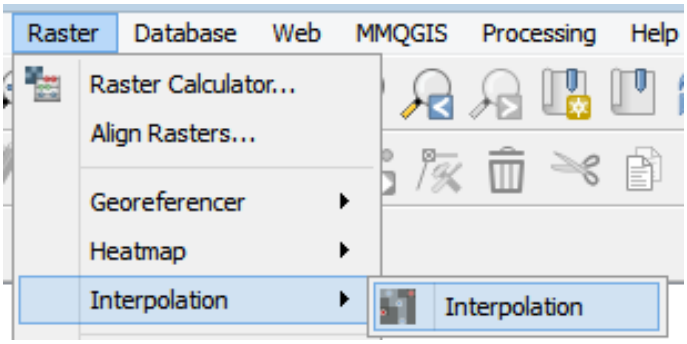


Figure 12.5 – Loading to the QGIS interpolation plugin.

When executing it, the following dialog window appears:

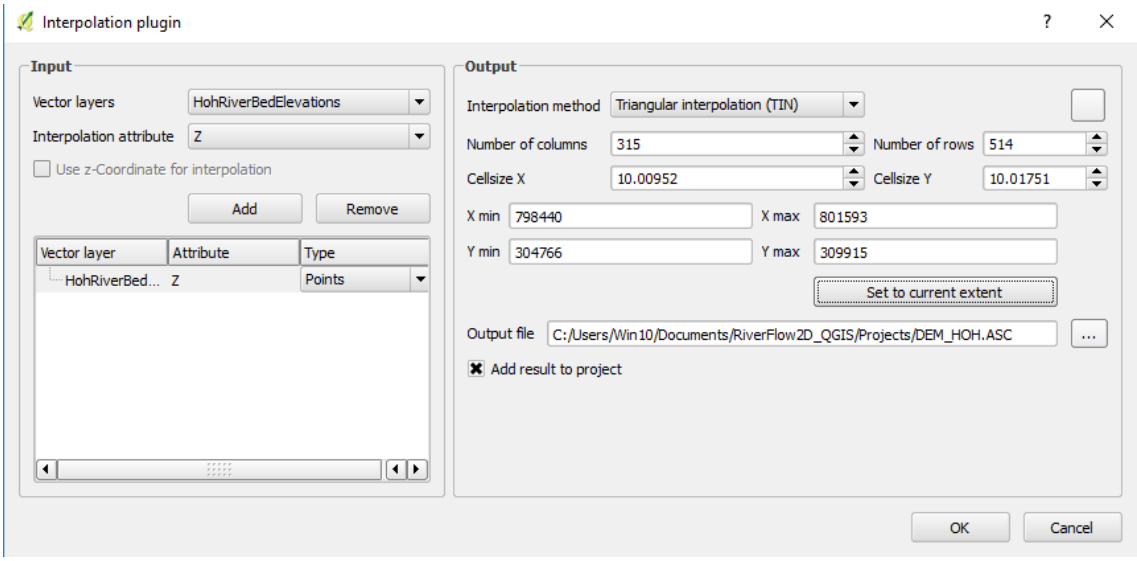


Figure 12.6 – Interpolation plugin window.

7. In this window select the layer to be interpolated to create the raster, in this example we assume it is a point layer that we just created named *HohRiverBedElevations*. Change it by selecting the dropdown for Vector Layers. We select the field that has the attribute that was used as the value to interpolate, in this case the z field. This is done by clicking the dropdown for Interpolation Attribute, selecting Z, then clicking the Add button.

8. Once added, go to select the *interpolation method*. There are 2 options: *triangular interpolation (TIN)*, or *Inverse Distance Weighting (IDW)*. In this case the TIN method is selected, then the resolution of the raster is defined, which is fixed to cells of $10ft \times 10ft$, which generates a raster of 315 columns x 514 rows.

9. Select the path to the folder where the raster layer is to be created. It is recommended to point to the project folder directory chosen earlier in the tutorial.

10. Click OK.

Once the process is completed, the raster resulting from the interpolation will be displayed on the screen, by default it is rendered in gray gradient as shown in Figure 12.6.

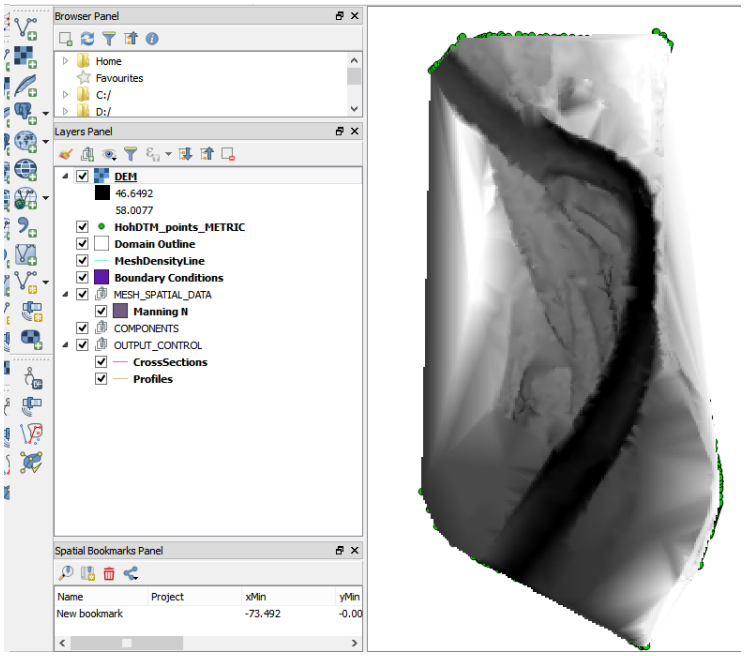


Figure 12.7 – Digital elevation model in raster format created by interpolation.

Note: Right-clicking on the label of the created layer and selecting *Properties* allows you to change the rendering style for a more informative color palette.

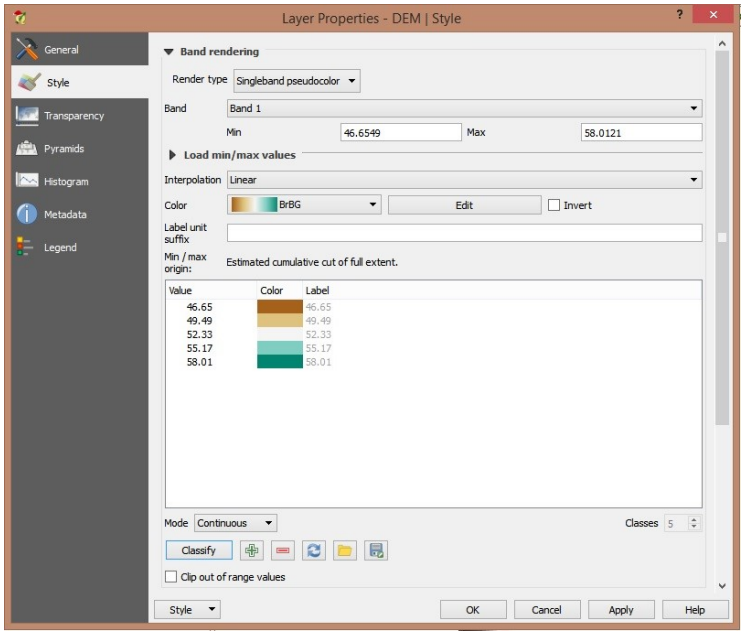


Figure 12.8 – Window to change the render style of a raster layer.

And now the raster layer is displayed with the new color palette selected:

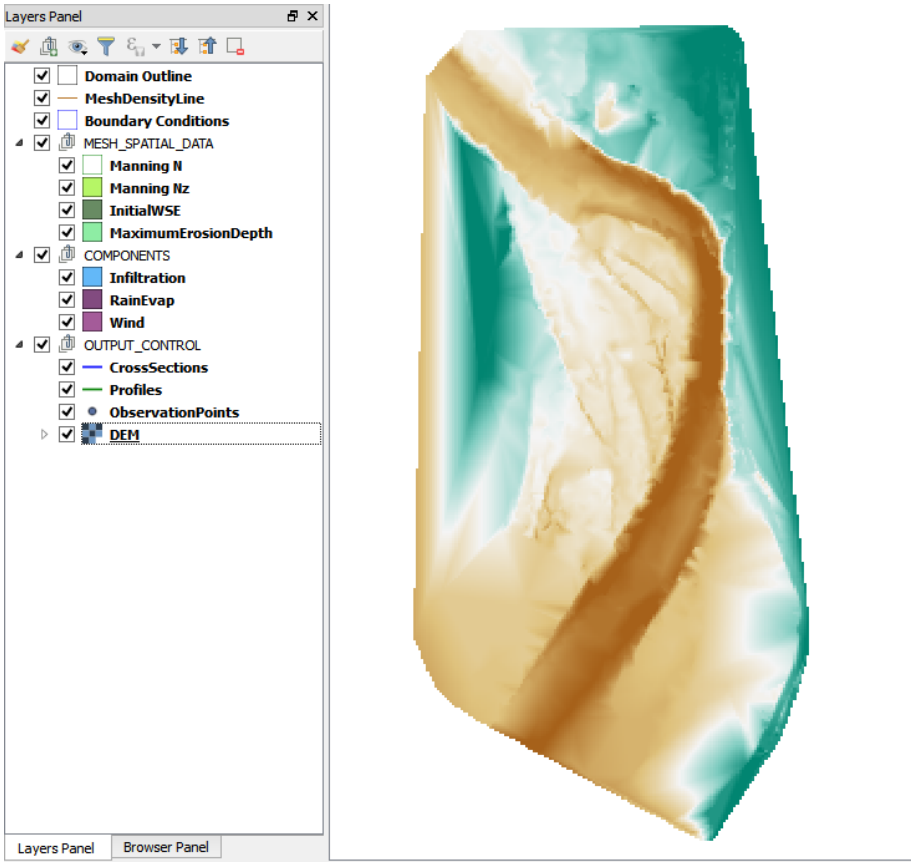


Figure 12.9 – Digital elevation model with color render.

It is convenient to move the raster layer created to the end of the list of layers, thus it does not interfere with the display of other layers.