FLO-2D®

TWO-DIMENSIONAL FLOOD ROUTING MODEL

FLO-2D QGIS PLUGIN

USER'S MANUAL

JUNE 2018

FLO-2D Software, Inc.

Table of Contents

Overview

The FLO-2D Plugin for QGIS is a tool that is used to develop the data input for the FLO-2D Model. The Plugin can import, export and create FLO-2D *.DAT files. QGIS is a free and open source geographical information tool that is available worldwide. The FLO-2D Plugin is also free and open source. The Technical Reference Manual that accompanies this document outlines the specific functions of the Plugin. This manual will provide instructions on how to install the Plugin, outline the Plugin features and demonstrate how to apply it through tutorials.

GIS Background and Experience

The user should be familiar with the Data Input Manual and FLO-2D Reference Manual in order to help validate data that is generated by QGIS and the FLO-2D Plugin.

It is assumed the user is familiar with general GIS concepts (e.g. data type, coordinate reference systems) and has experience in using the QGIS program. Reviewing the following topics will assist the user to become proficient with using the FLO-2D QGIS Plugin:

- Loading vector/raster data;
- Styling layers;
- Working with attribute tables;
- Data management using the field calculator;
- Data management including merging and cropping vector/raster data;
- Spatial analysis;
- Spatial queries;
- Manipulating the Processing Toolbox:
  - ✓ Creating a surface from points;
  - ✓ Converting rasters to vectors and vectors to rasters;
  - ✓ Raster calculation;
  - ✓ Creating slope/aspect from DTM points;

Installation Instructions

QGIS Installation – Stand Alone

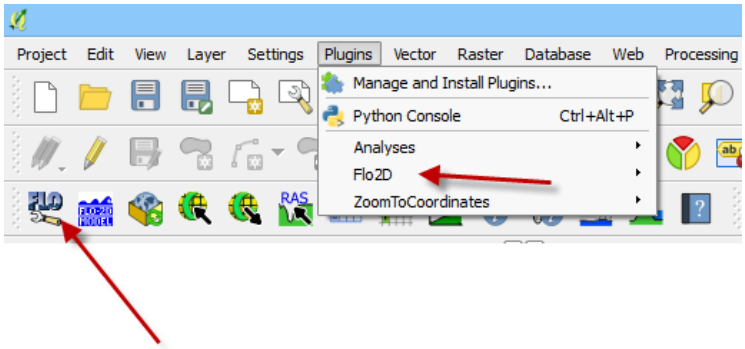
Version 3.2 Download the Installer from the QGIS website. www.flo-2d.com/qgis-plugin-flo-2d

Run the installer and follow the installation instructions. Use the default for all options.

Installing FLO-2D Plugin

1. Download the plugin zipped file www.flo-2d.com/qgis-plugin-flo-2d
2. Open Manage and install Plugins
3. Install the plugin using the Install from zipped file
4. The plugin is installed here:
   C:\Users\user\AppData\Roaming\QGIS\QGIS3\profiles\default\python\plugins\flo2d
5. In QGIS, from the main menu, select **Plugins** >> Manage and Install Plugins
6. FLO-2D plugin is listed and the toolbar will be visible

If your QGIS window looks like this, you've installed the plugin correctly.



Opening QGIS

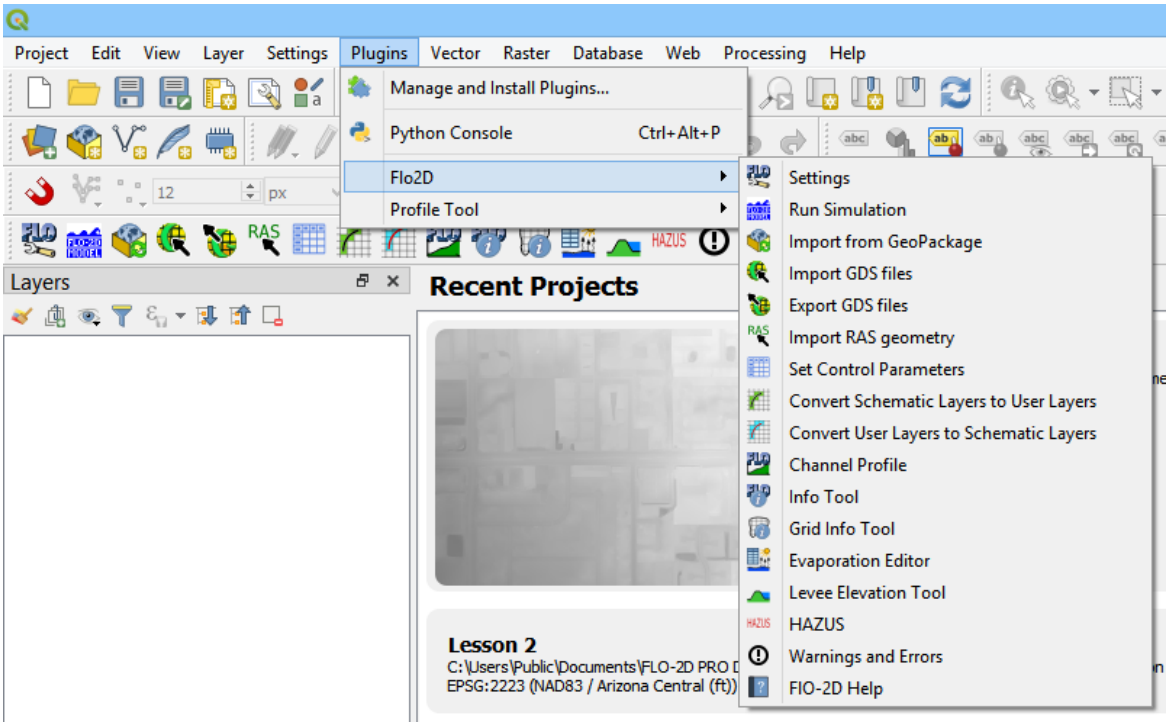
Run the program QGIS Desktop. It might also be called QGIS 3.4.2 Desktop with Grass.

Note: Be careful to call the correct version. QGIS may be installed many times on a computer. The older versions won't be replaced by the current build. That is common with open source programs.

Plugin Features

Plugin Menu

The FLO-2D Plugin menu is in the Plugins drop down menu, *Plugins>FLO-2D*. If the Plugin is not listed, follow the installation instructions to install the Plugin.



List of Additional Plugins

Plugins for suggested download that may come in helpful include:

- Profile tool
- QuickMapService

Note: For QuickMapService, it is necessary to install and add maps. Click on the QuickMapServices icon>Settings>More Services> Get Contributed Pack> Save.

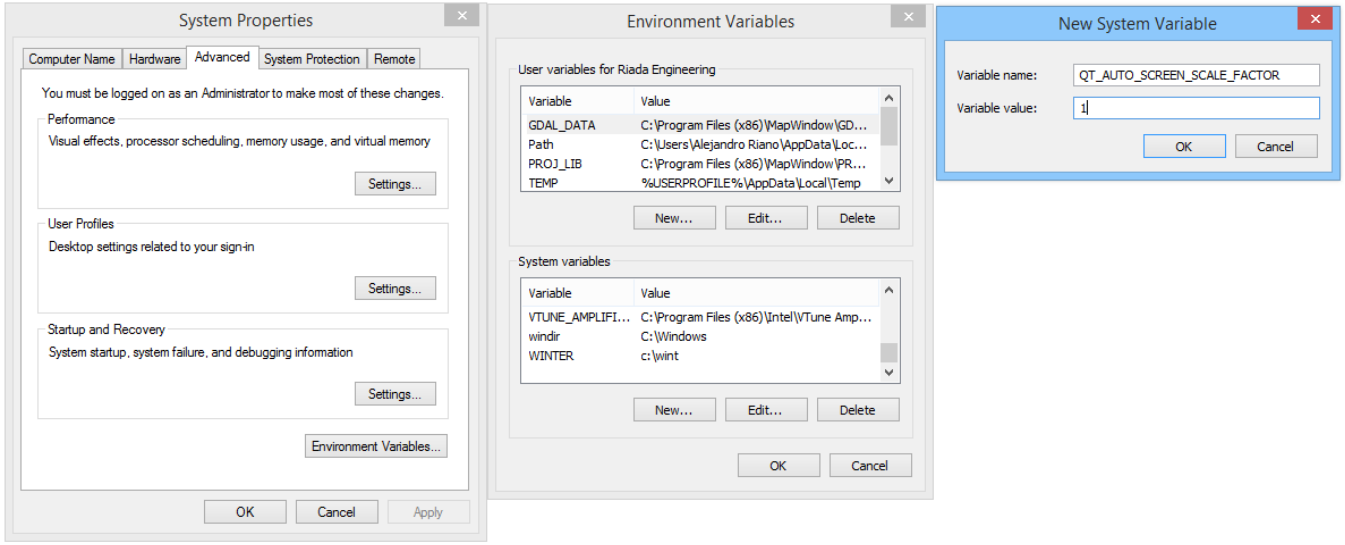
View Menu

High Definition Screens

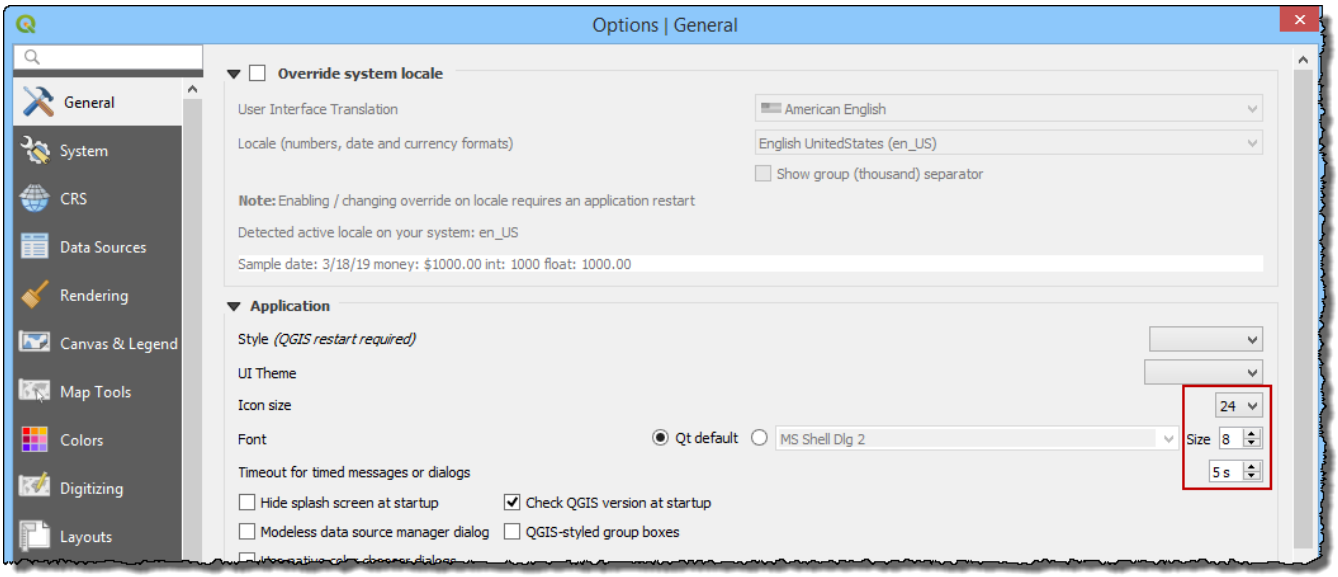
If the computer has a high definition resolution or ultra-high definition resolution, QT creator and QGIS may need an adjustment so all windows can be loaded correctly.

For QT Creator Windows

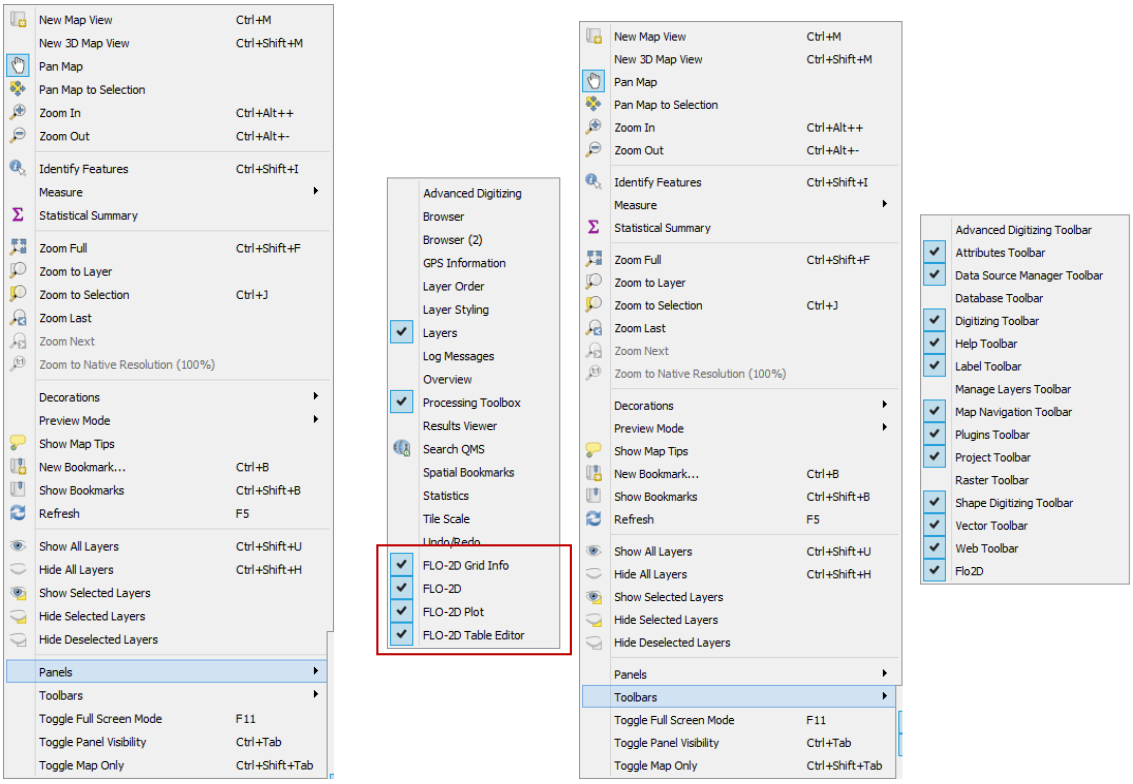
If the data in Widgets and Dialog Boxes is squeezed or not readable, set the *Environmental Variables* for QT Creator scaling factor, on the *Advanced* tab of the *System Properties* window, following the figures shown below. Set the *Environmental Variables* for the System not the user.



For QGIS, navigate to *Settings* > *Options* > *General* and set the icon size to 48 and the text size to 11.

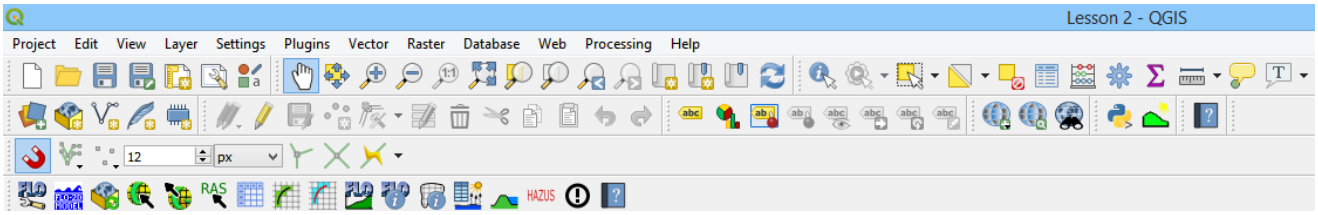


If any parts of the Pugin are missing, they can be retrieved using the View menu.



Plugin Toolbar

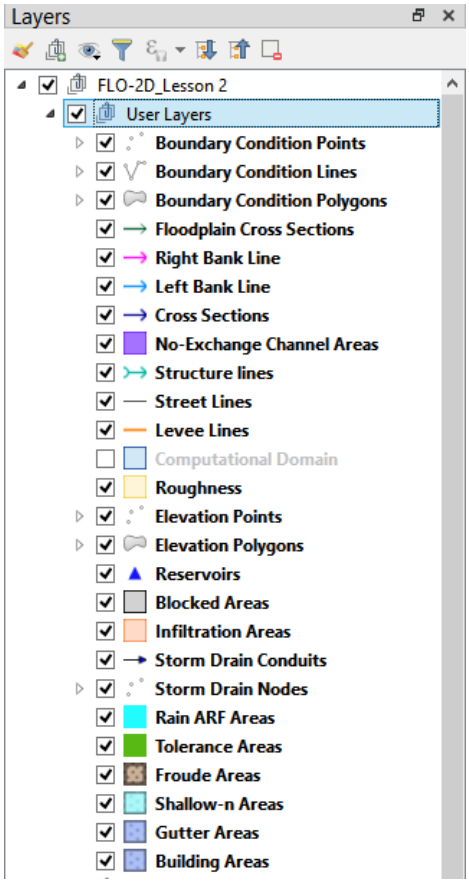
The FLO-2D Plugin toolbar is located on the QGIS toolbar lower tier. If the toolbar is missing, follow the FLO-2D Plugin installation instructions or use the *View* menu to load it onto the *QGIS Toolbar* menu.

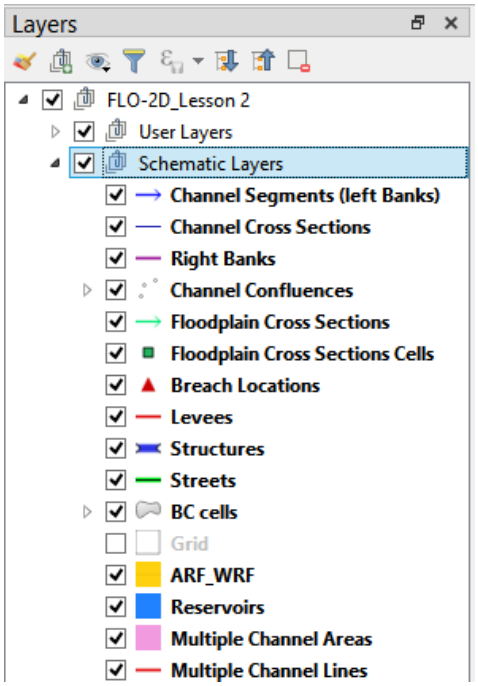


Plugin Layers

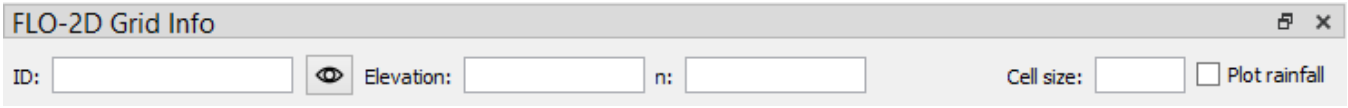
The *FLO-2D Layers* are loaded into the QGIS program when the user creates a GeoPackage. These layers are grouped into categories such as *User Layers* that can be edited and *Schematic Layers* that are written when data is schematized. There are also *Storm Drain Layers*, *Infiltration Layers*, *Data Tables* and *Channel Layers*. All the data is stored as SQLite data in a GeoPackage file.

If the Layers Panel is missing, use the *View* Menu to display it. This window can be docked or undocked by clicking and dragging the title area.





FLO-2D Grid Info Panel



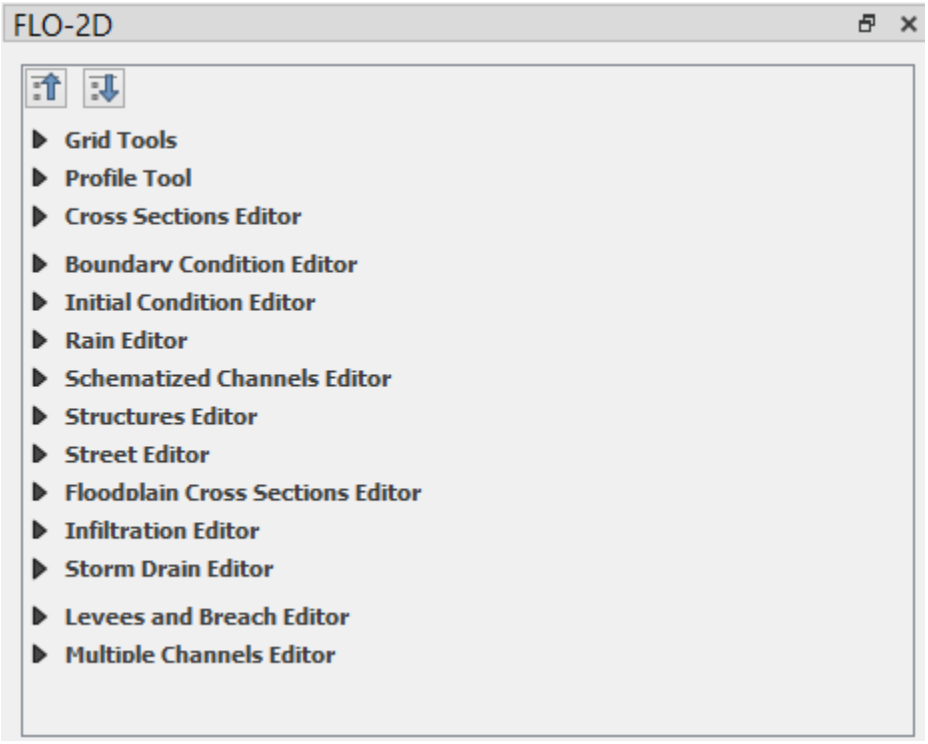
The FLO-2D Grid Info button is used with the FLO-2D Grid Info Panel. Click the button and then click any cell to fill the grid element information.

Enter a grid element number and click the Eye button. The map will zoom to that element and highlight it with red.

If the windows are missing, show them using the View>Panels menu.

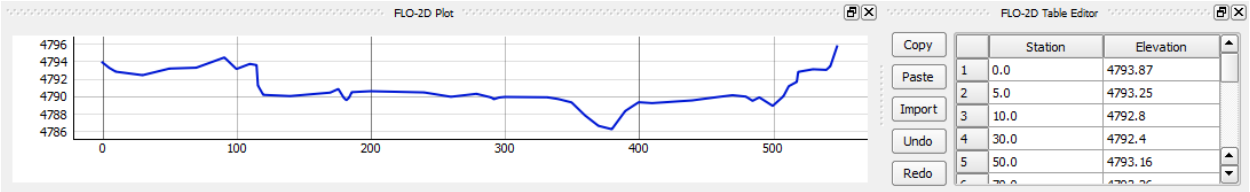
FLO-2D Editor Widget

The editor widget is used to digitize the User Layers. If the windows are missing, show them using the View>Panels Menu.



FLO-2D Plot and Table Editor Windows.

The plot window and table editor window are used to assign table data to various components such as inflow, rain, hydraulic structures and channels. The user can copy and paste from excel or ASCII text files. If these windows are missing, they can be restored using the View Menu and they can be docked or shifted to other monitor screens for easy editing and viewing.



Data Storage

Database Format

The Plugin uses a [GeoPackage](http://www.GeoPackage.org/spec/) for data storage. It is an SQLite (http://www.sqlite.org/) database with spatial extensions for storing vector and raster data. The GeoPackage or *.gpkg file is a binary file that stores data tables in SQLite format. SQLite is a public domain data base engine that is used and supported worldwide. More information about SQLite can be found at the website https://www.sqlite.org/about.html. The GeoPackage encoding system is approved by an Open Geospatial Consortium, a standard that is deemed sustainable by the U.S. Library of Congress (Library of Congress, 2017).

The FLO-2D Plugin uses the GeoPackage to store the data in a series of layers and tables. The Plugin requires a single GeoPackage file for each project. The project units and coordinate reference system are defined in the GeoPackage at the start of the project. The QGIS program has an extensive geodetic registry. It also supports “on the fly” mapping so layers of various coordinate systems can be viewed. The Plugin requires that all data used in the processing routines must be in the baseline coordinate system defined in the GeoPackage.

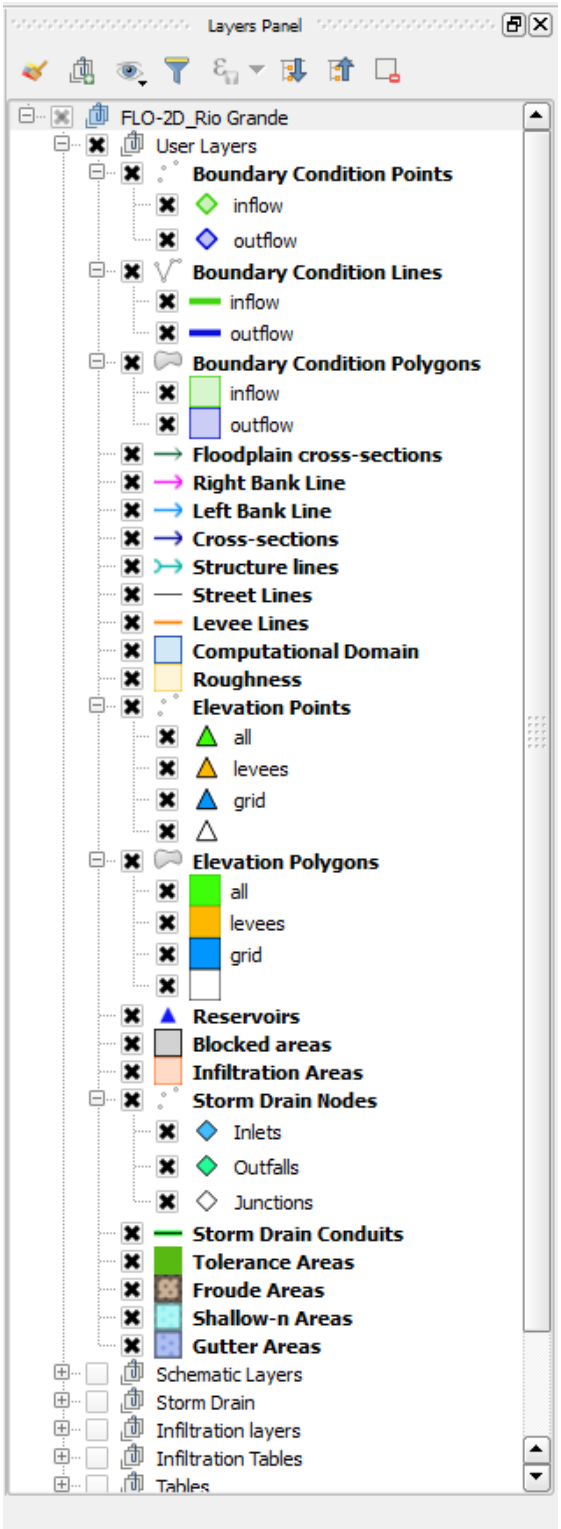
See the Technical Reference Manual for an outline of the GeoPackage data.

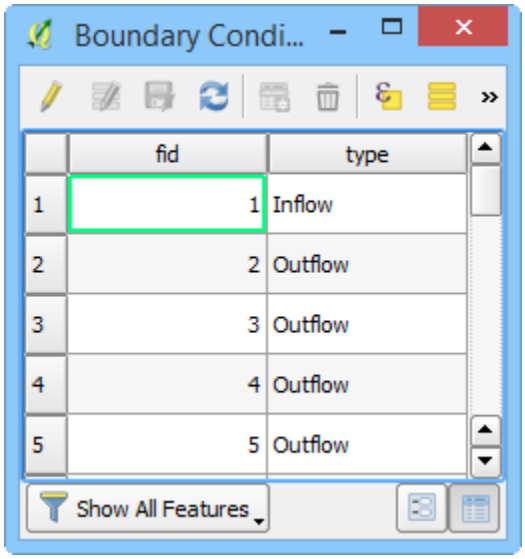
User Layers

Data is arranged as vector layers and tables in the Plugin. The vector layers are organized as follows:

- Points
- Polylines
- Polygons

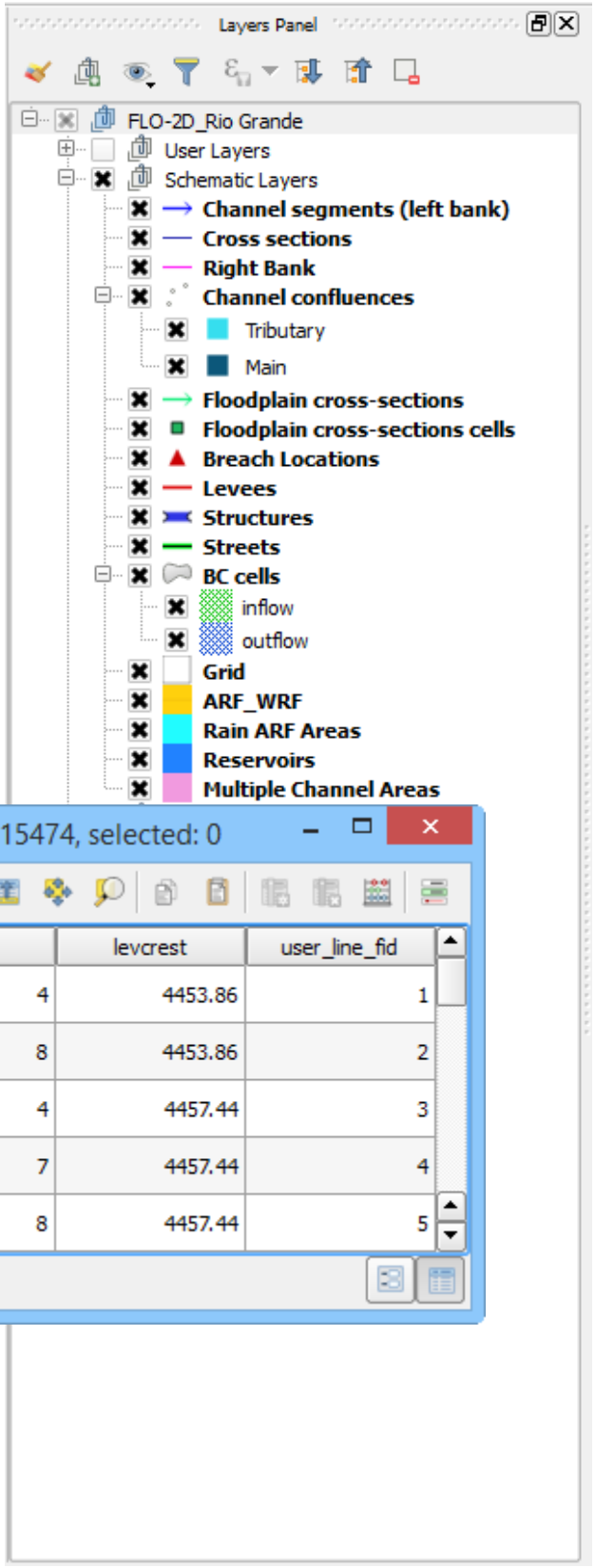
There is a layer for each FLO-2D model component that is digitized by the user and schematized into the "schematic" layers and tables. These layers are used to define the schematized layers that are FLO-2D format. The layers can be directly edited using the general QGIS editor tools and the *FLO-2D Editor Widgets*. They can also be edited using the *Attribute Table Editor* and the *Field Calculator*.

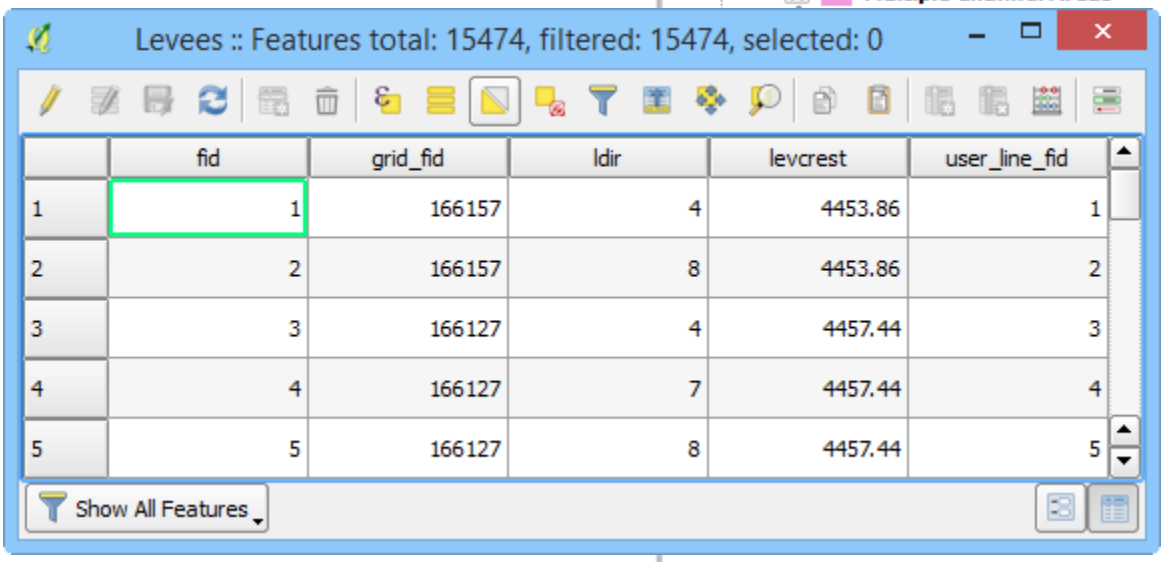




Schematic Layers

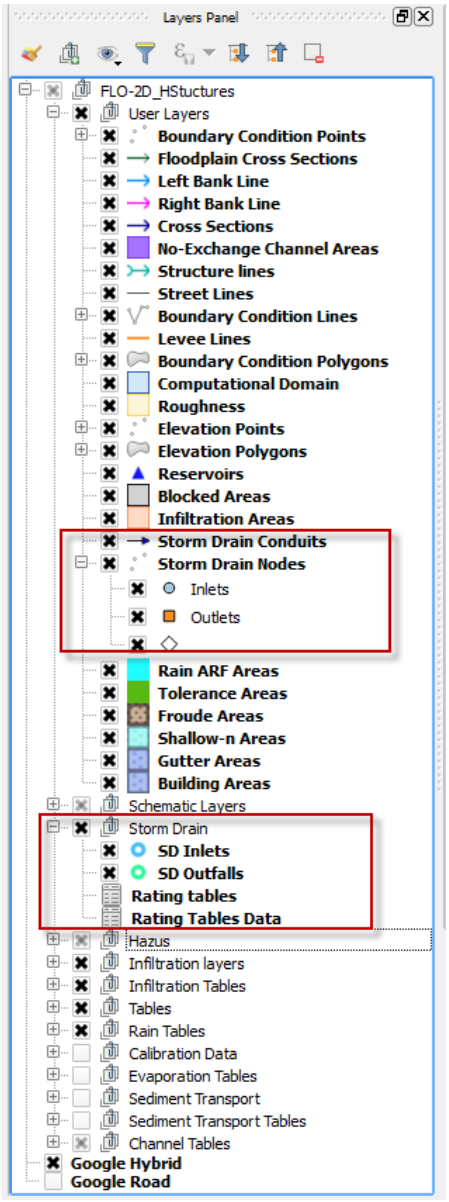
The *Schematic Layers* are organized such that the Plugin can generate the FLO-2D *.DAT files from them. These layers, created by the schematization tools, should not be manipulated by the user. The layers are vector layers with attribute fields that fill the FLO-2D data files.





	fid	grid_fid	ldir	levcrest	user_line_fid
1	1	166157	4	4453.86	1
2	2	166157	8	4453.86	2
3	3	166127	4	4457.44	3
4	4	166127	7	4457.44	4
5	5	166127	8	4457.44	5

Storm Drain Layers



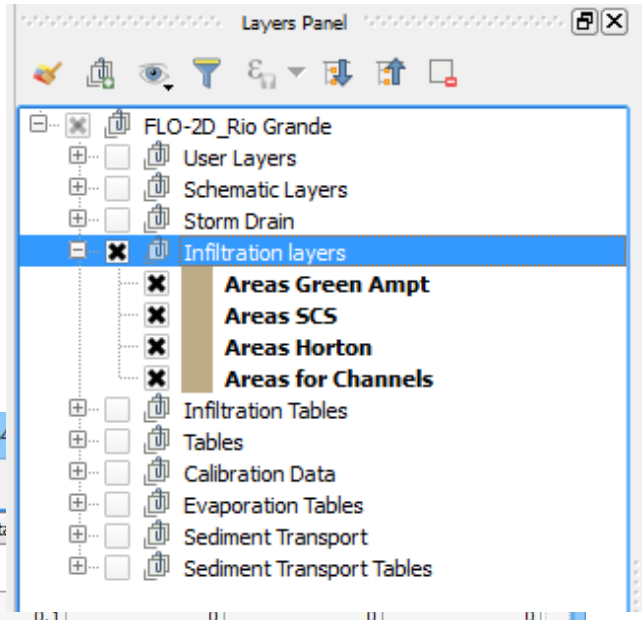
The *Storm Drain Layers* are used to create and store data for the storm drain system.

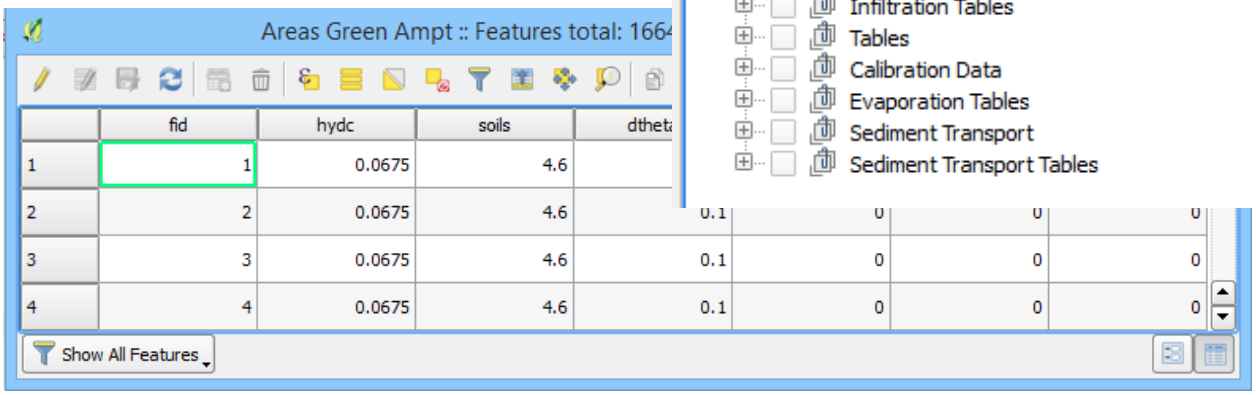
Data is written to these tables in the following methods.

- Importing from FLO-2D Project;
- Importing from an INP file;
- Importing from a shapefile;
- Digitizing directly.

Infiltration Layers

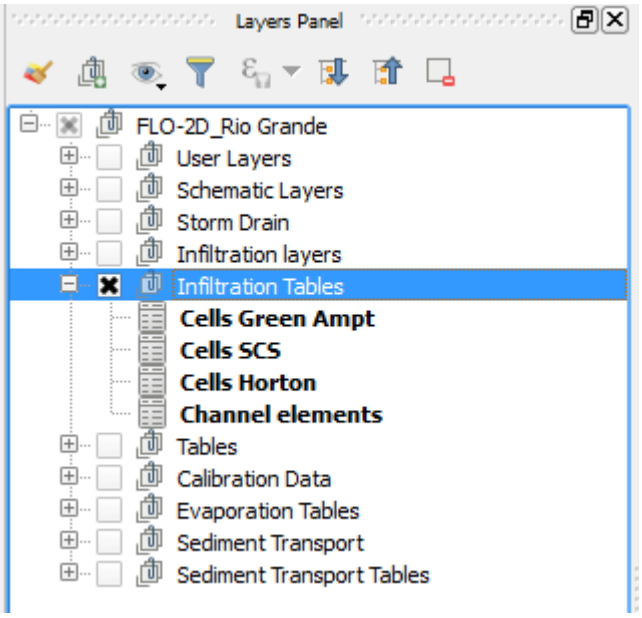
The schematic *Infiltration Layers* are polygons that store infiltration data computed from the infiltration tables.

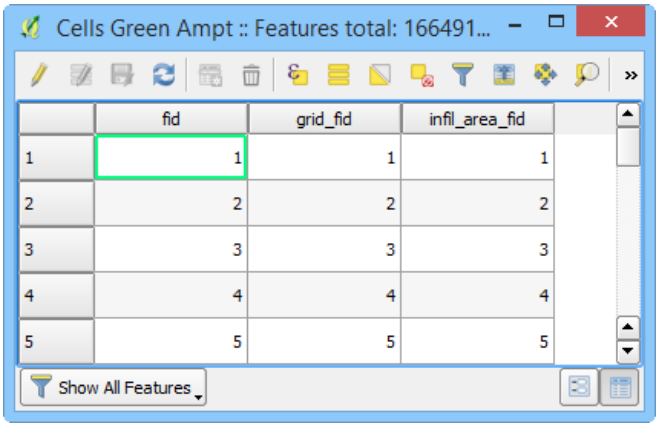




Infiltration Tables

The *Infiltration Tables* join grid elements to the *Infiltration Layers* and are produced by the *Infiltration Editor* Widget.



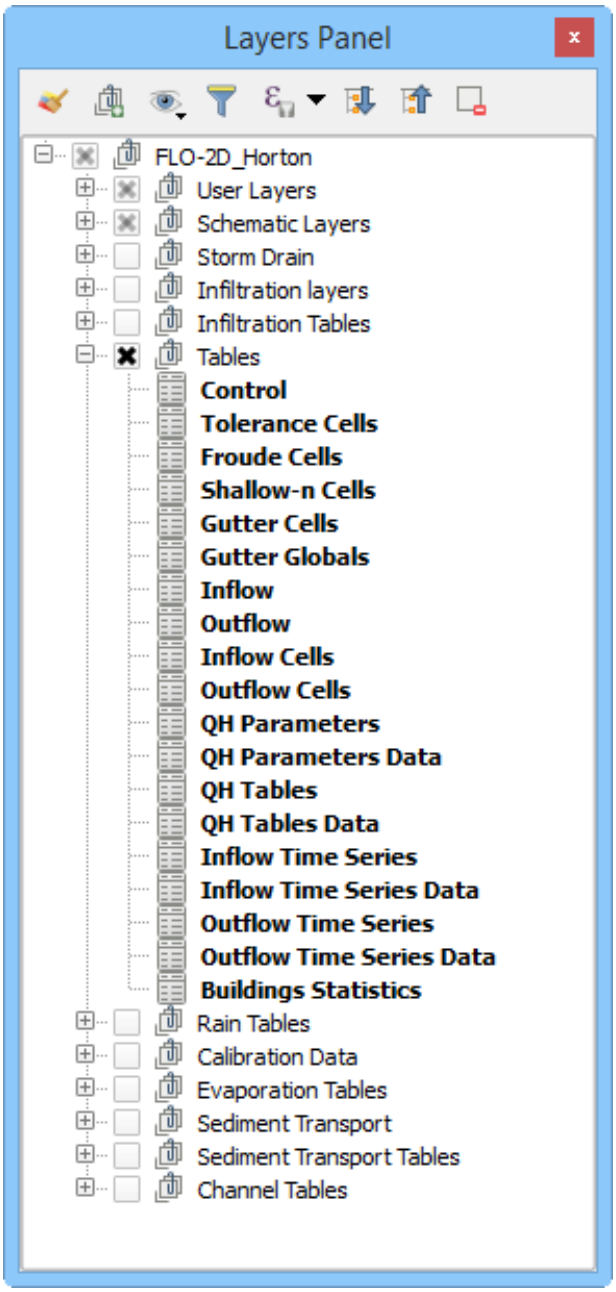


Tables

The *Tables* layer stores the data that is converted into the *.DAT files.

Data is written to these tables in the following methods.

1. Importing from FLO-2D Project;
2. Schematizing data.

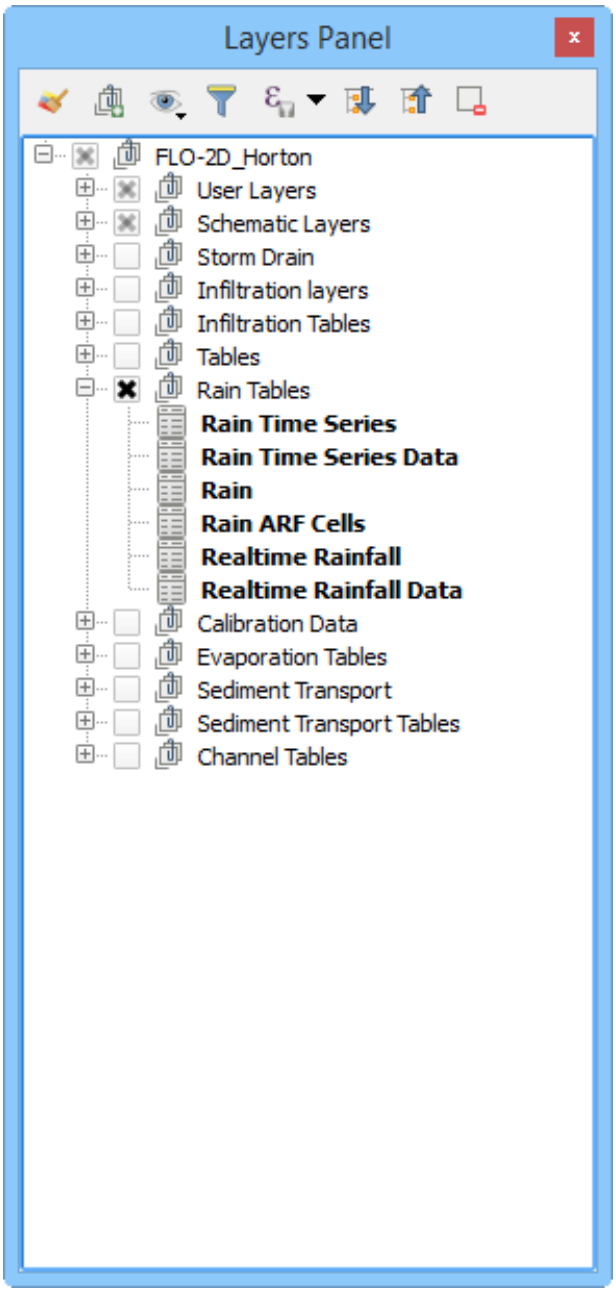


Rain Tables

The *Rain Tables* layer stores the data for uniform rainfall, spatially variable rainfall, and spatially and temporally variable rainfall.

Data is written to these tables in the following methods.

- Importing from project;
- Schematizing uniform rainfall;
- Calculating spatially variable RainARF from NOAA Atlas data;
- Calculating spatially and temporally variable data from NEXRAD radar and realtime storm data.



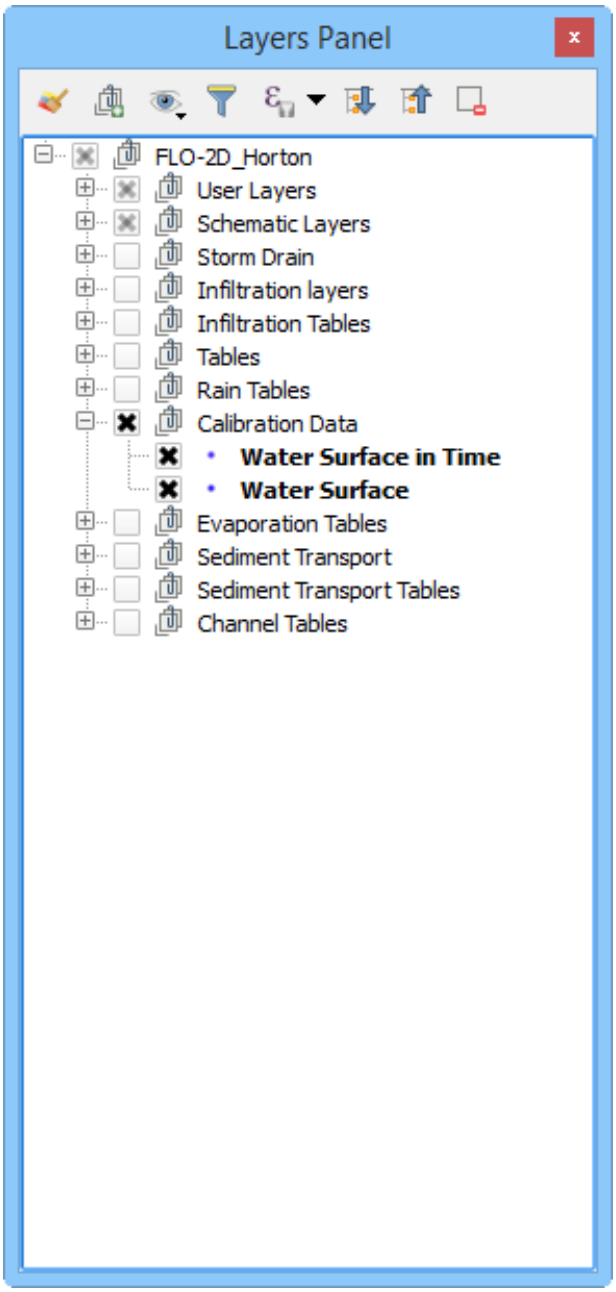
Calibration Data

The *Calibration Data* layers are point vector layers that store the water surface elevation at a known point and the time to peak water surface elevation at a known point.

Data is written to these tables in the following methods.

- Importing from project;
- Directly digitizing layer.

The data is written to the WSURF.DAT file and WSURFTIME.DAT file.

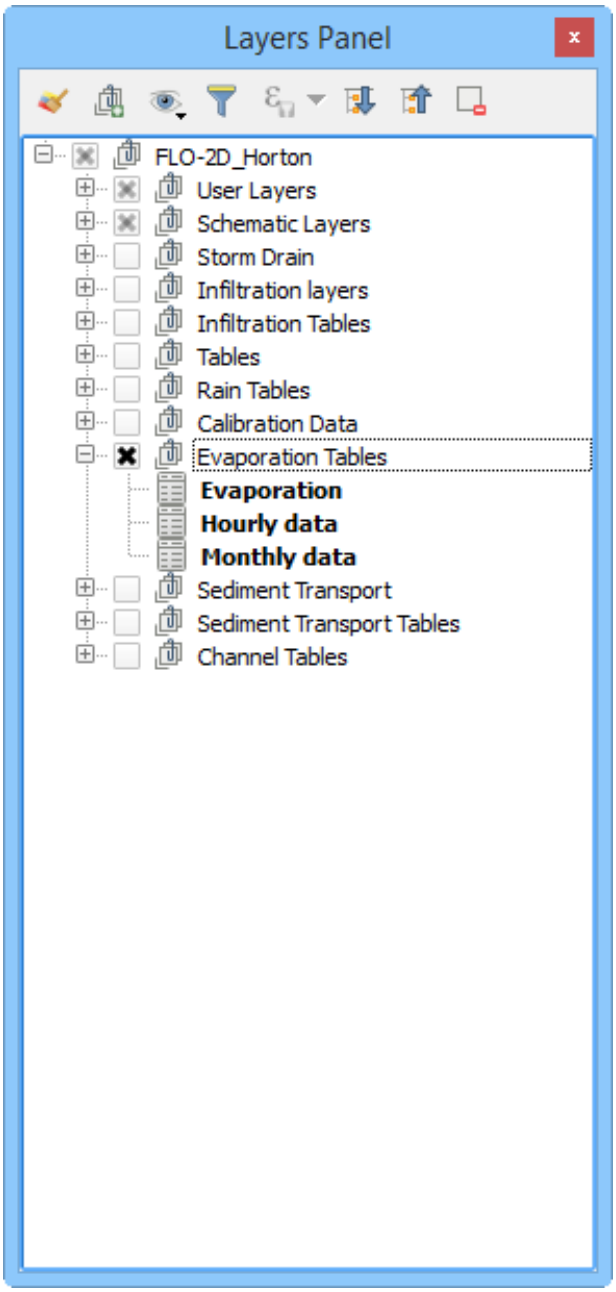


Evaporation Tables

The *Evaporation Tables* layer store spatially variable data for estimating evaporation at runtime.

Data is written to these tables in the following methods.

- Importing from project;
- Evaporation tool.

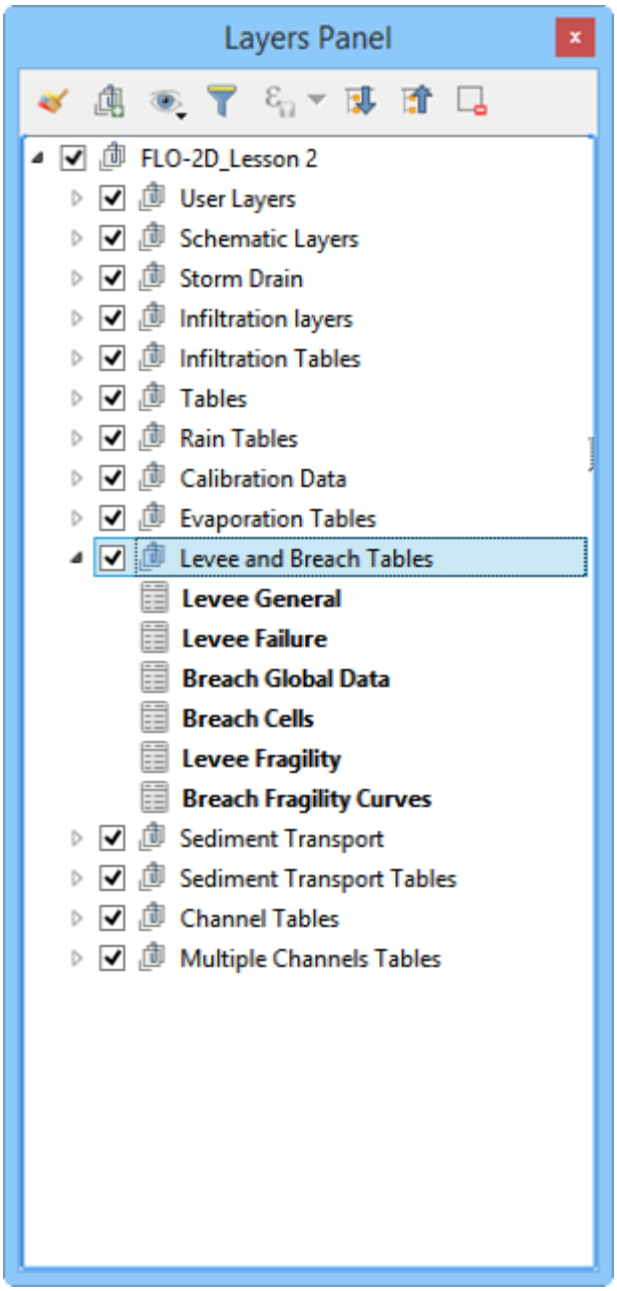


Levee and Breach Tables

The *Levee and Breach Tables* layer is used to store the prescribed breach, erosion breach, and levee fragility data tables. These labels are filled by the Breach Widget.

Data is written to these tables in the following methods.

- Importing from project;
- Schematic edits;
- *Levee Breach Editor* widget.

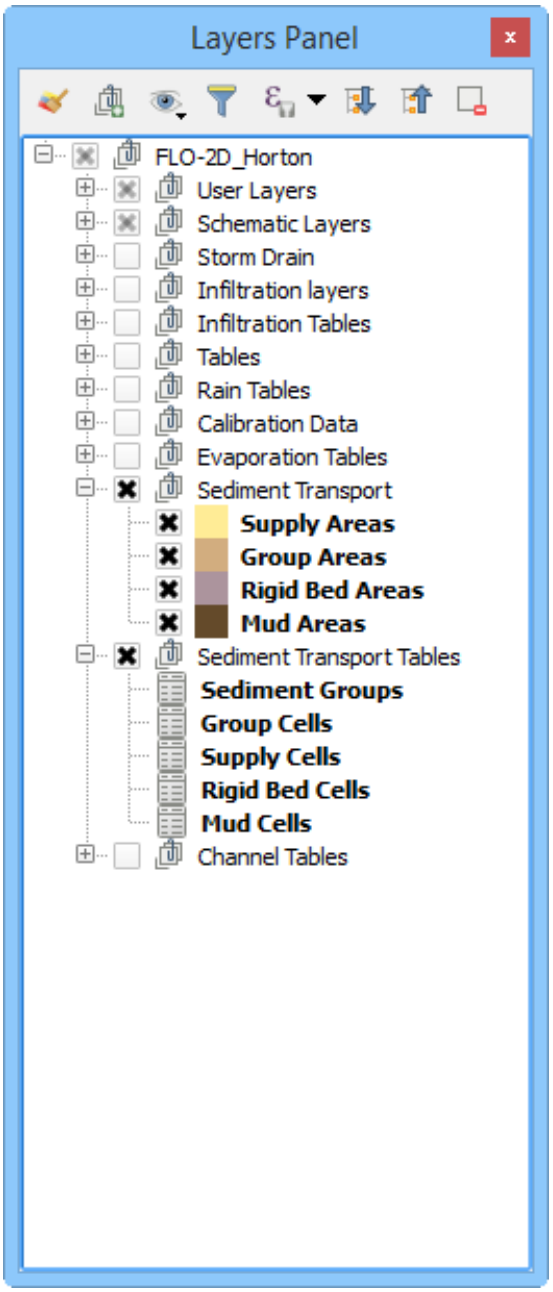


Sediment Transport Layers and Tables

The *Sediment Transport* layers, and *Tables* layer store the spatial and global data for sediment transport and mudflow.

The data is written to these tables in the following methods.

- Importing from project;
- Directly digitizing the data into the sediment transport areas;
- Calculating the data from the Areas and assigning it to the grids.

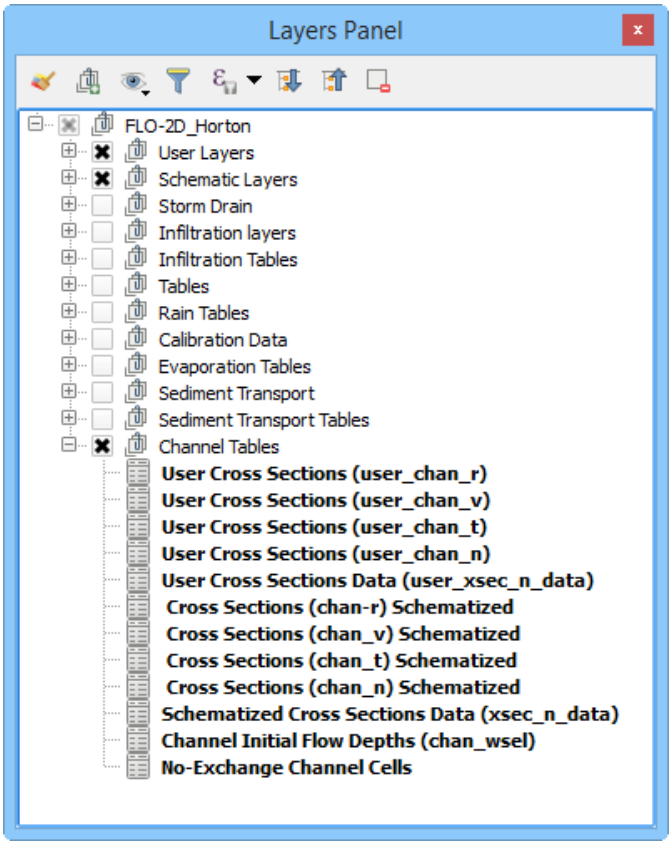


Channel Tables

The *Channel Tables* layer stores data for user layers and schematic layers.

The data is written to these tables in the following methods.

- Importing from project;
- Running Import RAS tool;
- Digitizing Channels;
- Calculating right banks;
- Interpolating cross sections;
- Schematizing channel data.

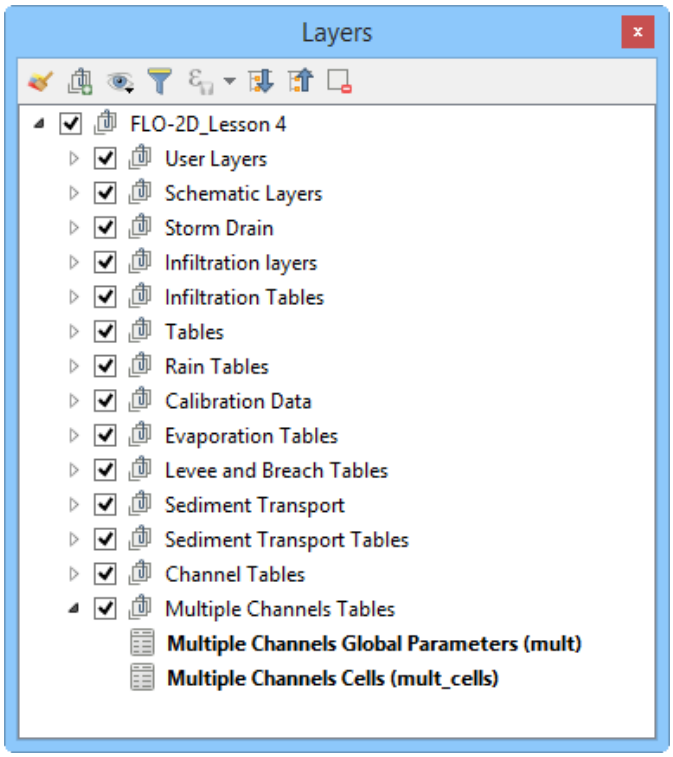


Multiple Channel Tables

The Multiple Channel Tables layer stores data for user layers and schematic layers.

The data is written to these tables in the following methods.

- Importing from project;
- Multiple Channel Layers;
- Multiple Channel Widget.



FLO-2D Tools

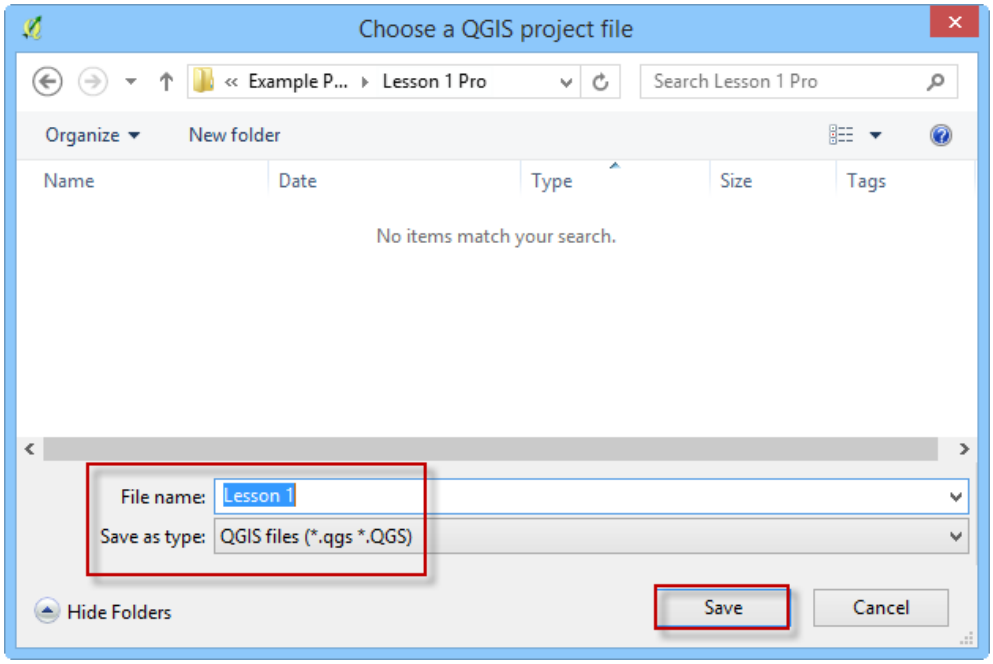
The following toolbar descriptions are organized by the location of the tool on the QGIS window starting with some basic QGIS tools and then the *FLO-2D Toolbar* command icons and finally the *FLO-2D Widgets*. Each section is broken into three parts:

- Tool overview
- Instruction guide
- Troubleshooting

QGIS Tools

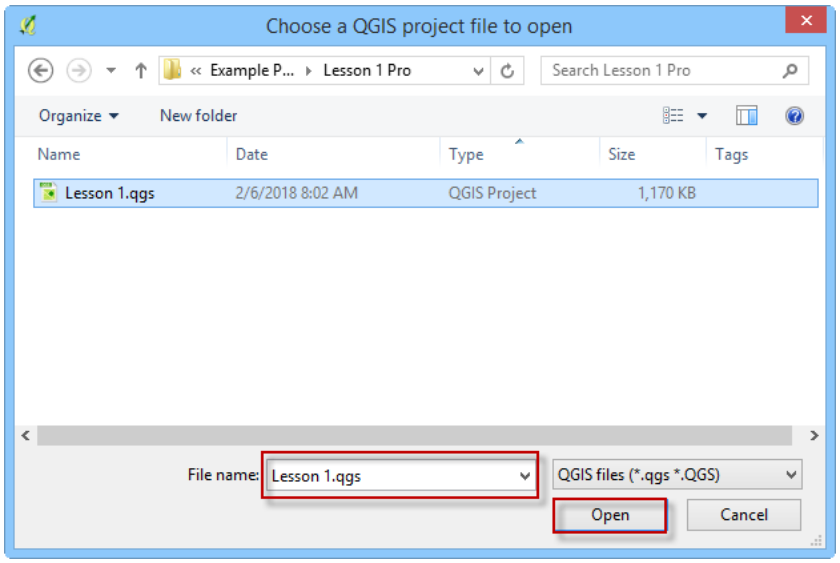
Save Project

Click the *Save* Icon on the QGIS toolbar. Save the file to the project directory.

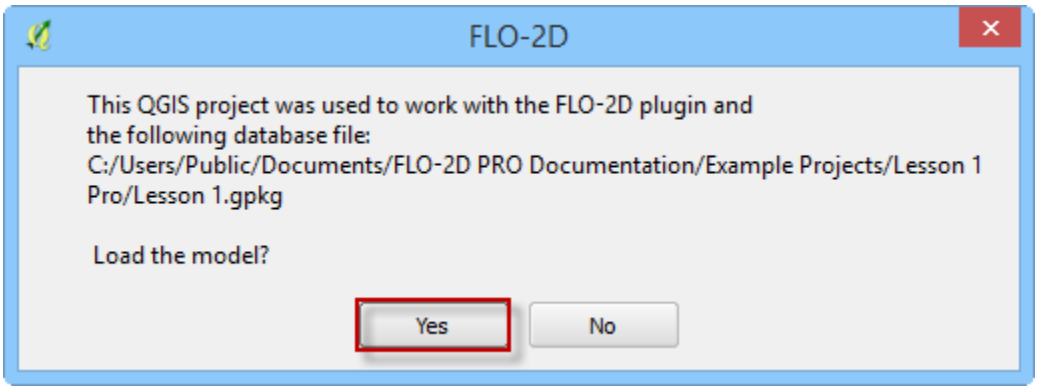


Open a Project

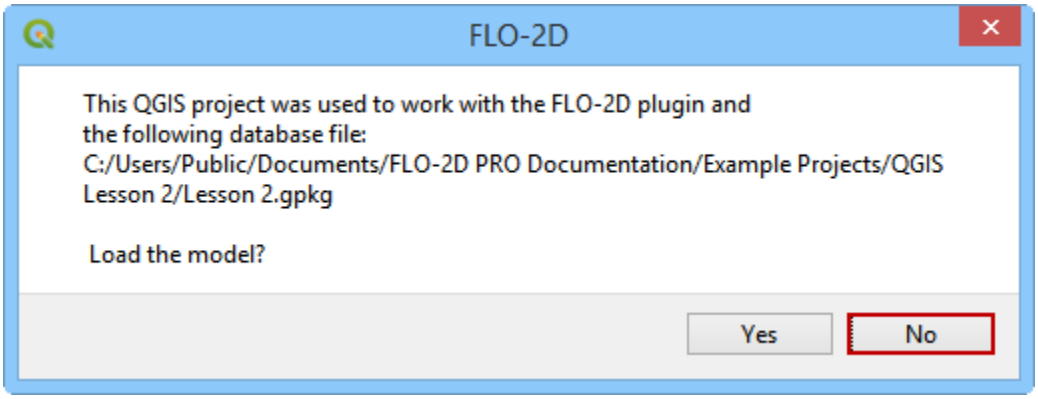
Click the open icon on the QGIS toolbar. Navigate to the project folder, select the *.qgs file and click *Open*.



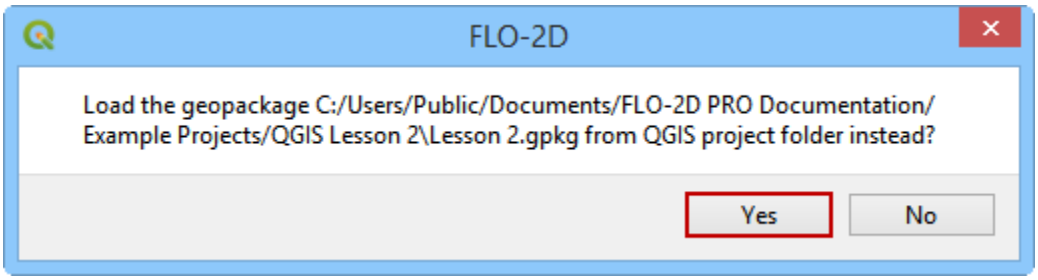
Click *Yes* to load the GeoPackage in the FLO-2D format.



Or Click *No* to add a GeoPackage from the project directory.



Click *Yes* and the project will load from the project directory.



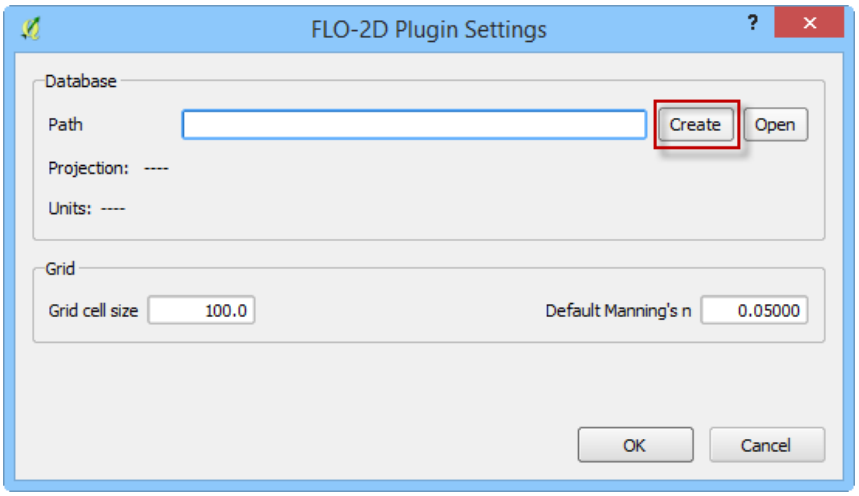
FLO-2D Toolbar

The *FLO-2D Toolbar* contains many tools to setup, import, and export FLO-2D data into QGIS.

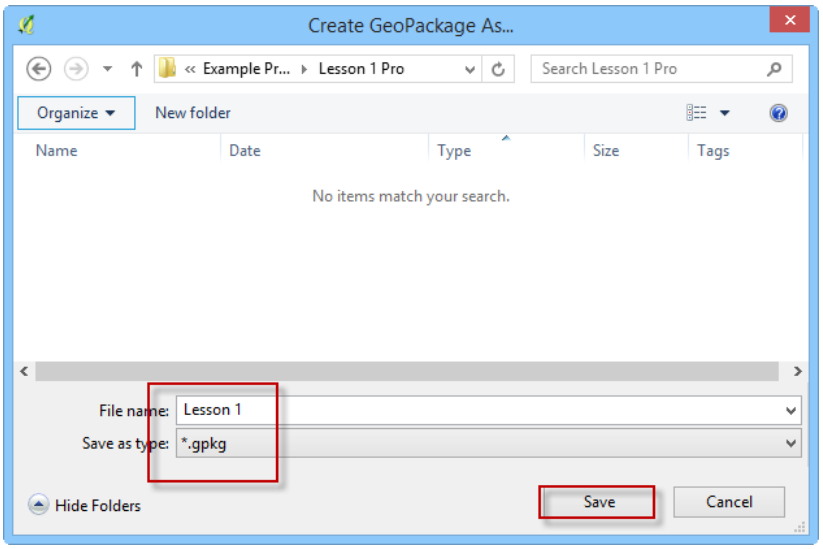
FLO-2D Plugin Settings

The *FLO-2D Plugin Settings* tool is used to set up the FLO-2D layers, set the coordinate system and create the GeoPackage file.

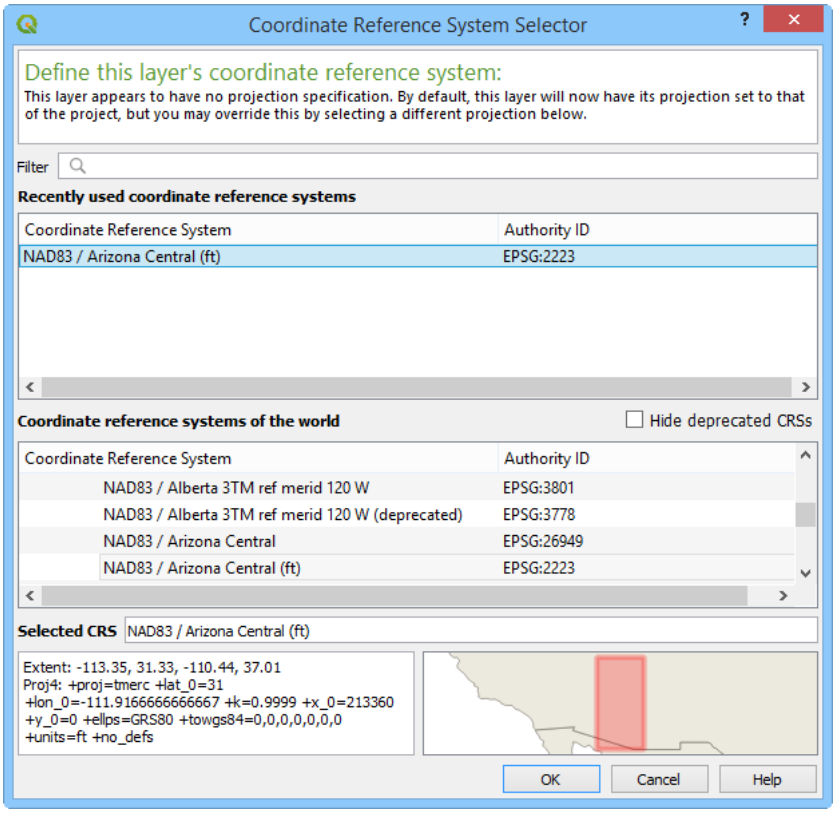
Click on the *FLO-2D Plugin Settings* tool icon to assign the desired grid element size and baseline n-value. Click the *Create* button to set up the GeoPackage location.



Navigate to the project location: Project name is the user's choice. Name the file (*).gpkg and click *Save*.



Set the coordinate reference system (CRS) for the project and click *OK*.



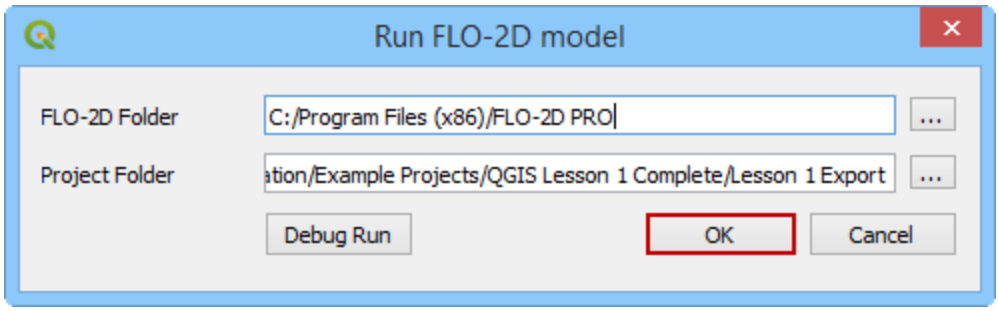
Run FLO-2D

Click the Run FLO-2D icon on the FLO-2D Plugin toolbar. Set the Project Export folder location and the location of the FLOPRO.EXE file.

Click *OK* to start a simulation.

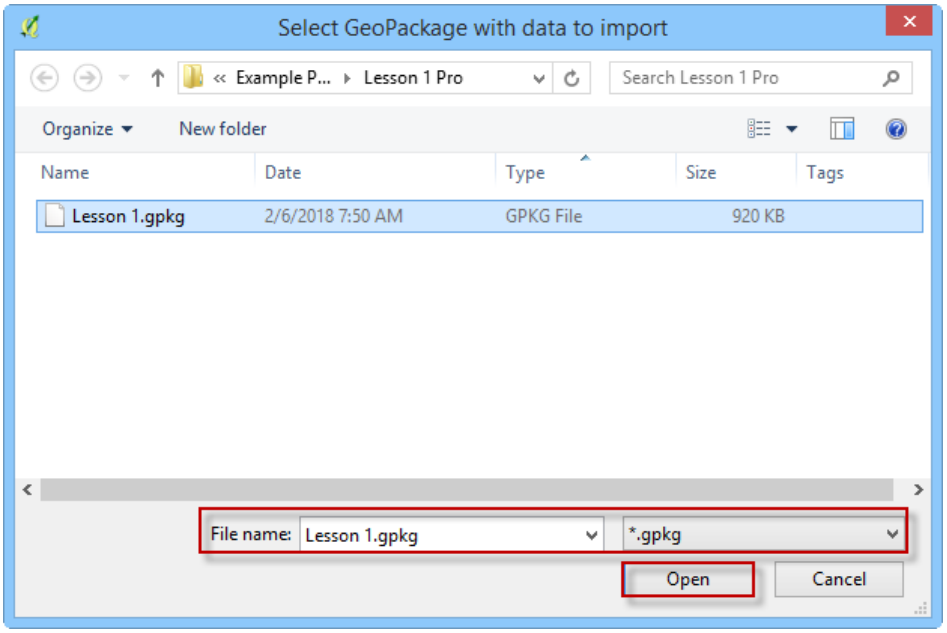
Or

Click *Debug Run* to conduct a Data Review and generate a DEBUG.OUT file. A Debug Run will limit the simulation time and run a set of data checks that are reported to DEBUG.OUT. This file can be imported back onto QGIS for project review using the Warnings and Errors button. 



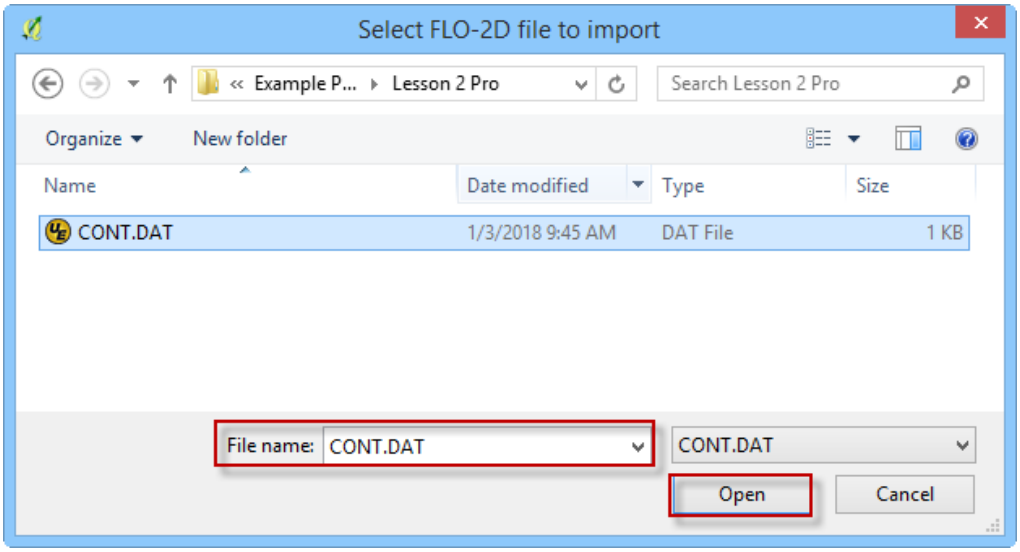
Import a GeoPackage from a Previous Build

When the FLO-2D Plugin is updated, the GeoPackage structure changes. The project can no longer be opened in the standard method described in the previous section. To load a GeoPackage from a previous build, set up the project (see *FLO-2D Plugin Settings*). Select the *Import from GeoPackage* icon from the *FLO-2D Toolbar* and navigate to the location of the GeoPackage and click *Open*.



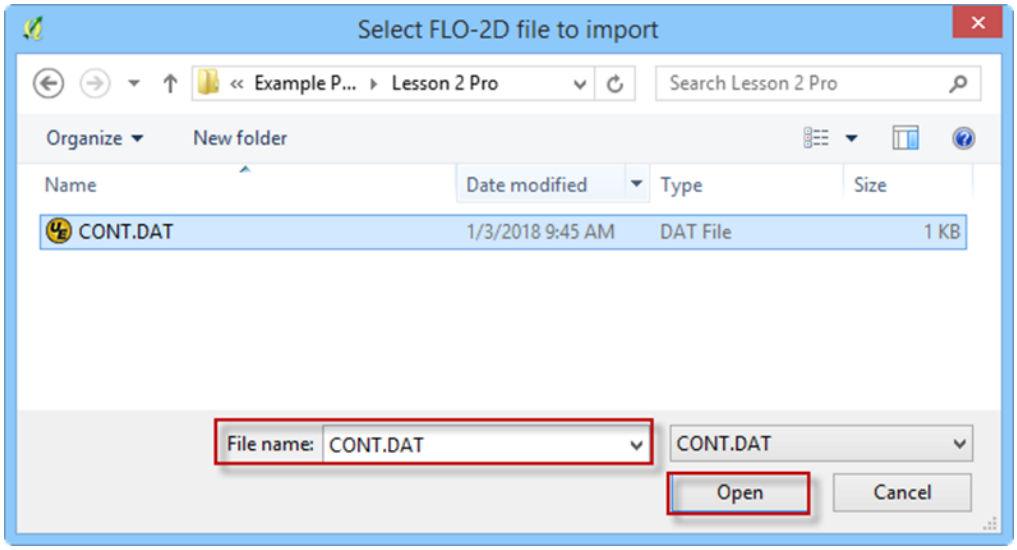
*Import the FLO-2D *.DAT files into the Project*

Click the *Import GDS Files* icon. Navigate to the FLO-2D *.DAT files, Select CONT.DAT and click *Open*. All FLO-2D files are imported to the Plugin with this option.

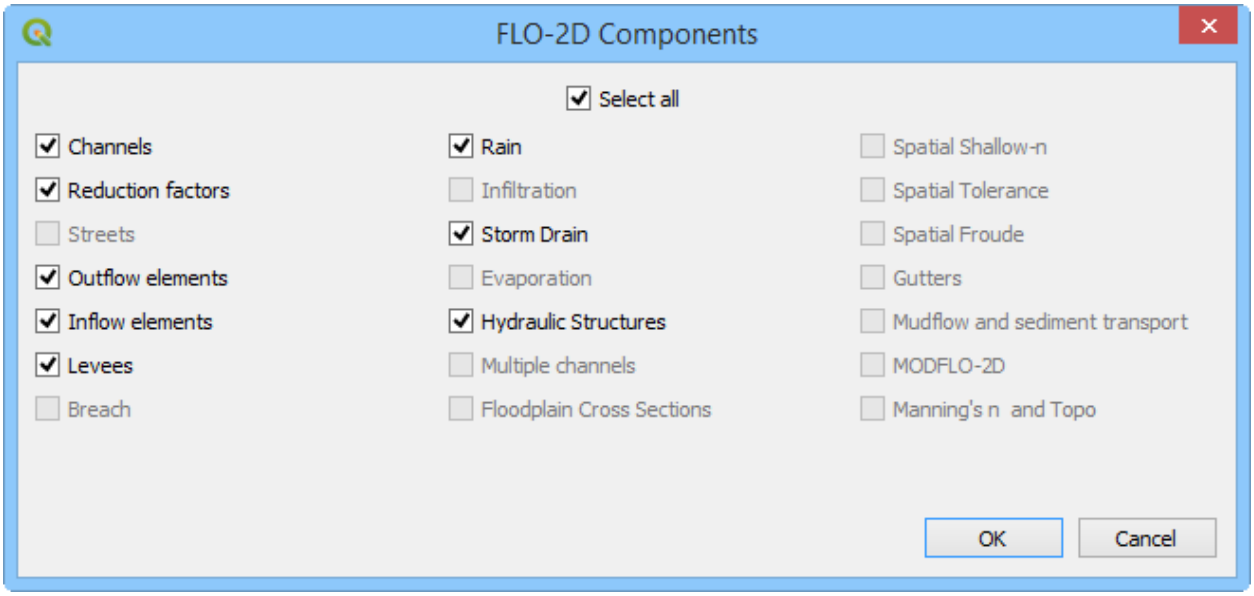


Import Selected Component Files

Click the *Import GDS Files* icon. Navigate to the FLO-2D *.DAT files, Select CONT.DAT and click *Open*. Only selected components will be added using this feature. The GeoPackage will not be overwritten.

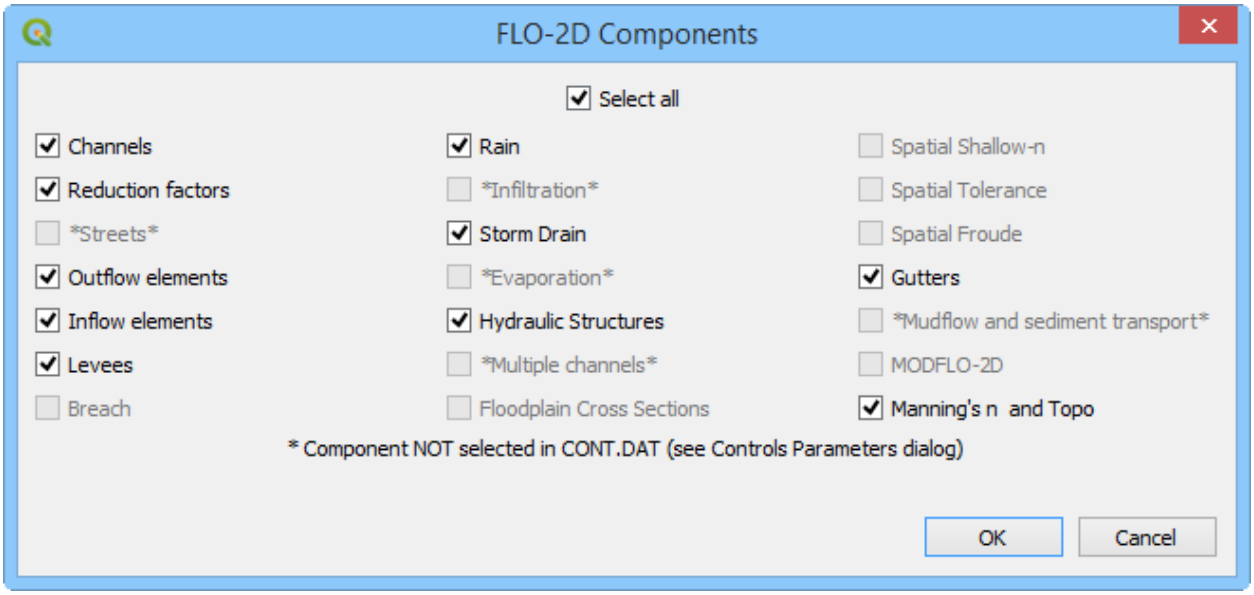


Select the FLO-2D Components to load into the project.



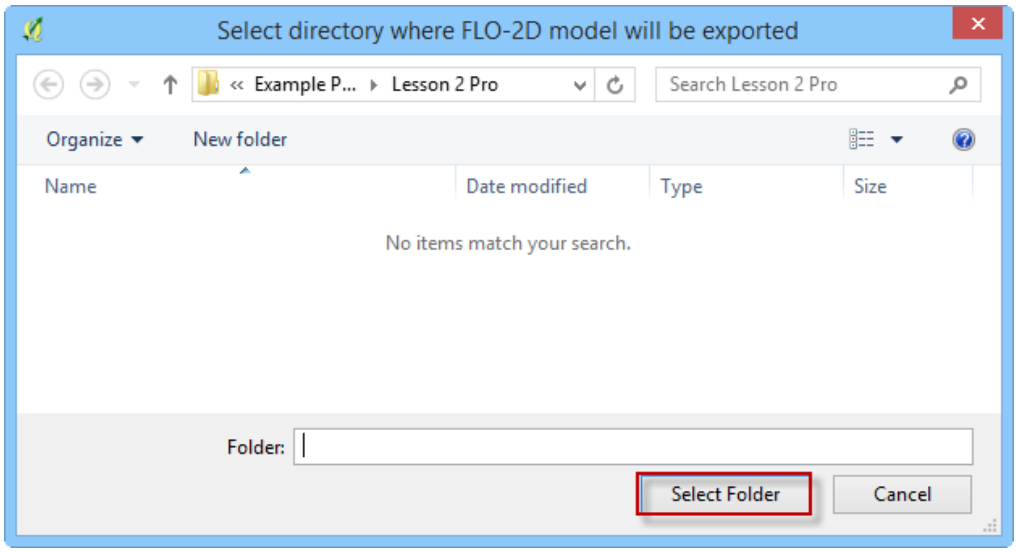
*Export FLO-2D *.DAT Files*

Click the *GDS Export* icon. Select the components that require an export. It isn't necessary to export all files every time. Export large files like IN-FIL.DAT orTOPO.DAT only when needed.



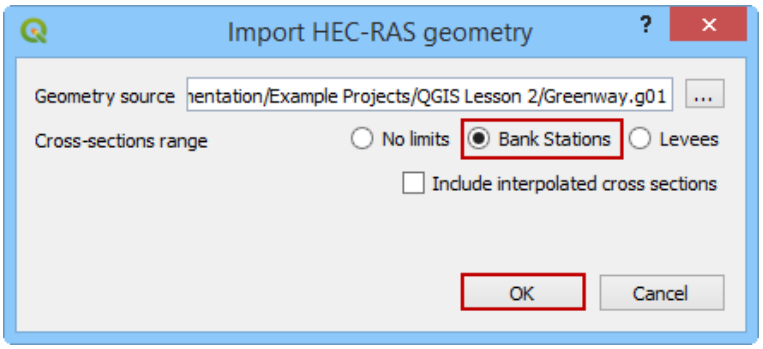
Navigate to the project folder and click Select Folder.

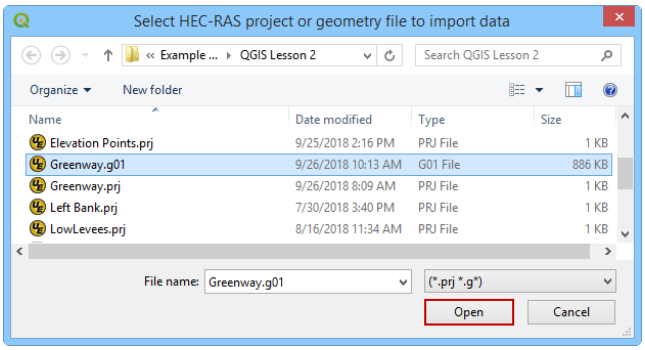
*** Set the Control Data switches before running the Export tool. See Setup the FLO-2D Control Data step. ***



Import HEC-RAS GeoRAS Channel

Click the *Import RAS Geometry* icon to import channel data from a GeoRAS project. The tool will import the project from the *.g0 geometry file or the *.prj project file.



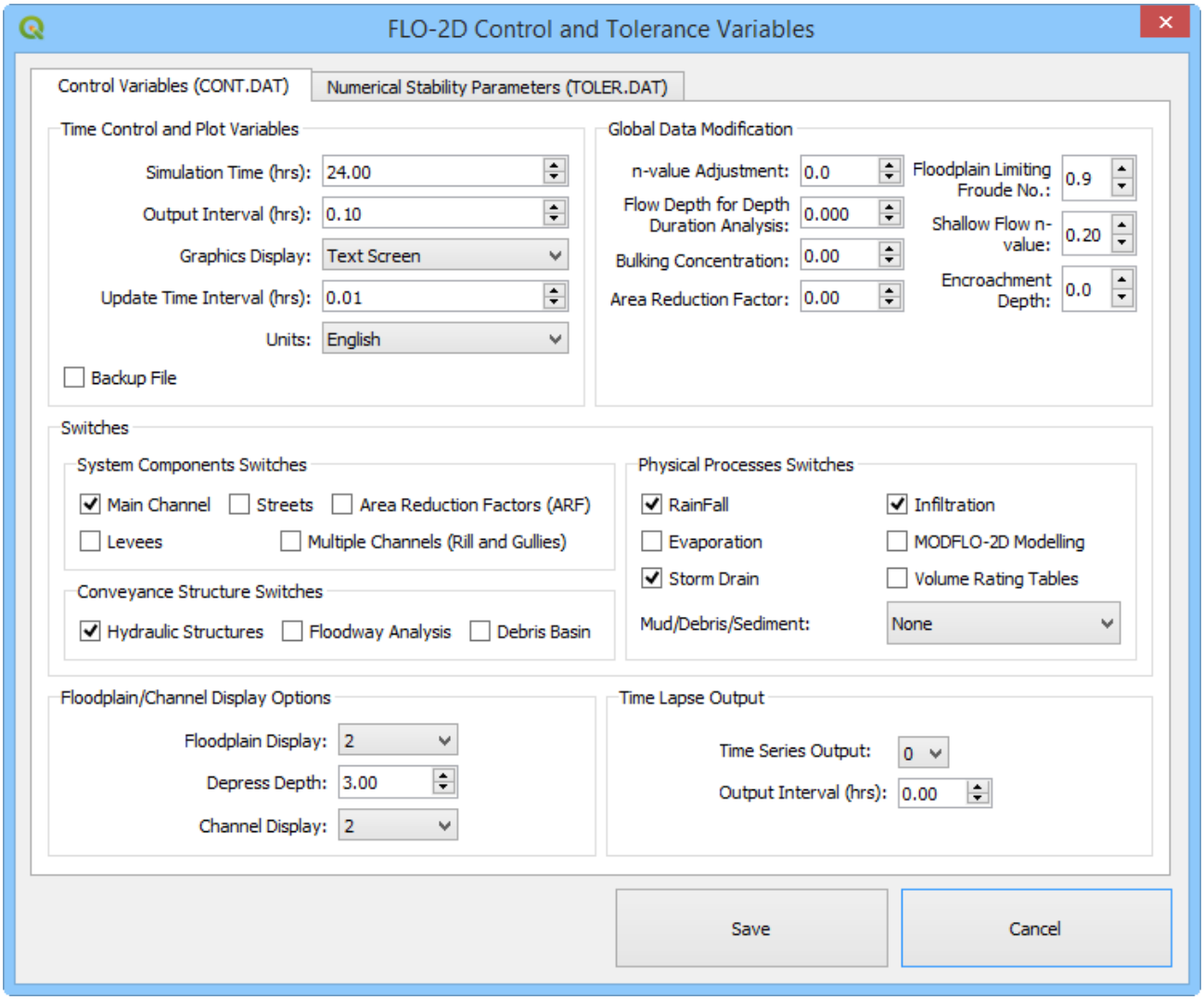


The imported RAS channel is saved to the *User Layers*. The user can edit the channel placement or cross sections so the channel will be schematized correctly. See the *Cross Section Editor* in the FLO-2D *Widgets* section below for further instructions.

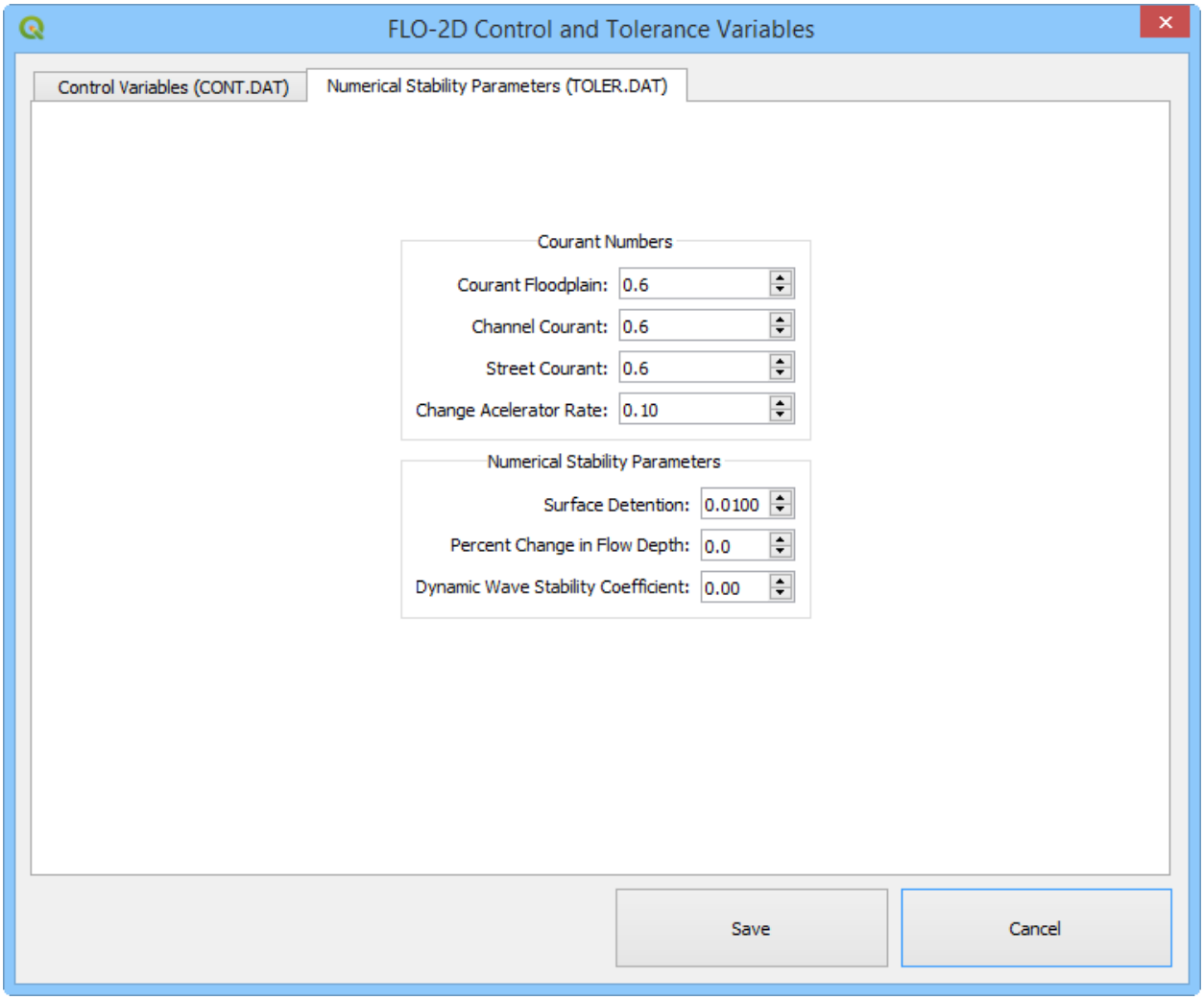
Setup the FLO-2D Control Data

Click the *Set Control Parameters* Icon. The dialog box is used to set the control data for the CONT.DAT and TOLER.DAT files. Save the data to the GeoPackage with the *Save* button. The variable descriptions and instructions are available in the Data Input Manual.

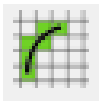
Important – If the user unselects a *Component Switch*, that data file will not be written when the data is exported. Use this method to speed up data export when large data files like Infiltration are present. The original data will be preserved.



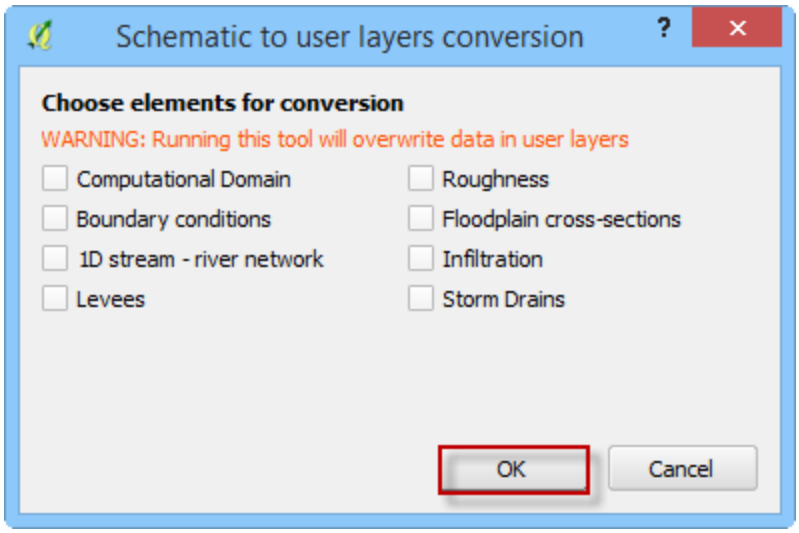
Numerical stability parameters are defined on the second tab.



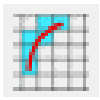
Convert Schematic Layers to User Layers



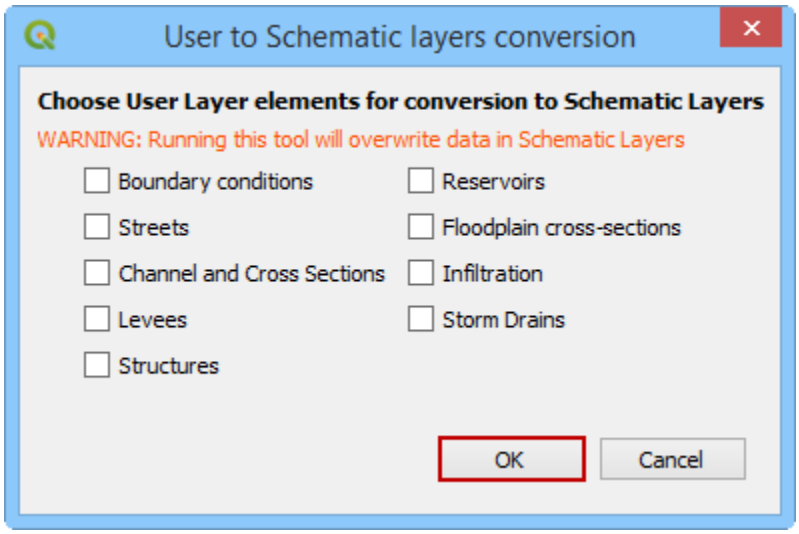
Click the *Convert Schematic Layers to User Layers* icon. This tool will convert the *Schematic Layers* to *User Layers* so they can be edited.



Convert User Layers to Schematic Layers

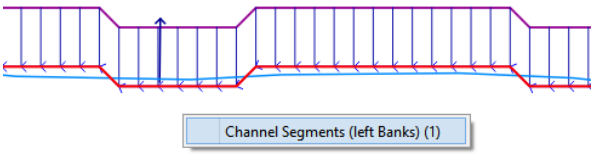


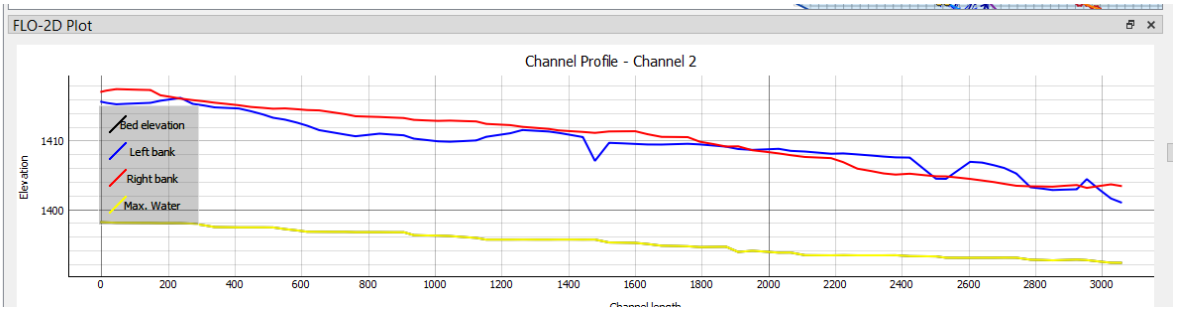
Click the *Convert User Layers to Schematic Layers* icon. This tool will write the *User Layers* to *Schematic Layers*.



Channel Profile Tool

Click the *Channel Profile* icon and then click a channel of interest and select the left bank.



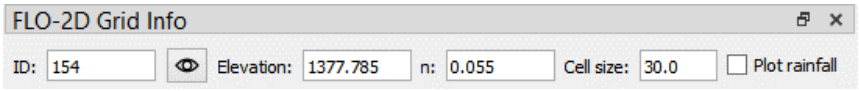


FLO-2D Info Tool

Use the *Info Tool* to identify data in the *User Layers*. The layers must be active and checked on in the *Layers* Panel. Click the *Info Tool* icon, then select the component on the map. The data will load into the editor widget, *FLO-2D Plot*, and *FLO-2D Table Editor*. See a description of how to use the tool in the Profile Tool section. The two tools are coupled.

Grid Info Tool

This tool loads the FLO-2D Grid Info panel. Click the icon and then click any grid element.



Evaporation Editor

This tool loads the evaporation editor.

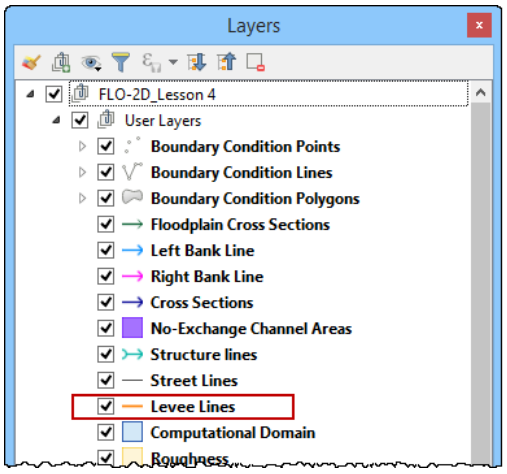
Levee Elevation Tool

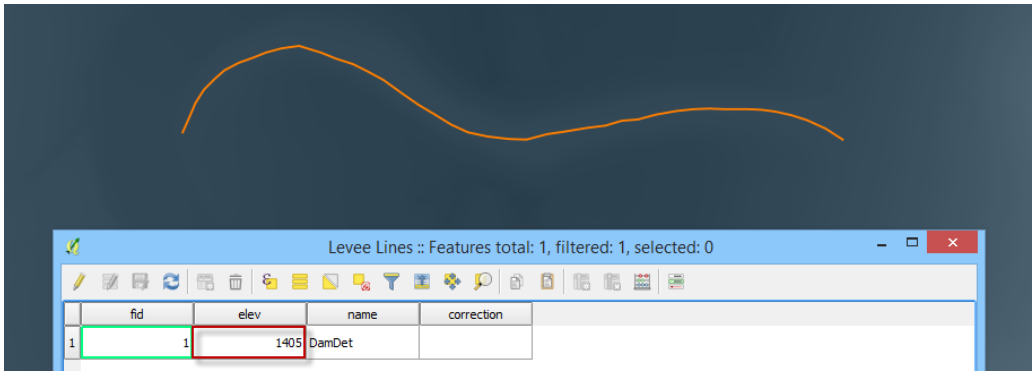
The *Levee Elevation* Tool will create levees, berms, walls and dams. It uses data from 4 sources:

1. User Elevation Polygons
2. User Elevation Points
3. Levee User Lines*
4. External 3D Lines

The method to create levees from each data source is outlined below. * The Levee User Line features are required when using the Polygon, Point and Levee Line selections.

Start by creating a Levee Line any place where a levee should be positioned. Do this by digitizing the lines in the *Levee Lines* layer of the *User Layers*. Select the layer and click the *Toggle Editing* pencil. Add a polyline and assign a levee crest elevation if it is a uniform levee. It is not necessary to assign a crest elevation if points or polygons will be used to define the crest elevation.



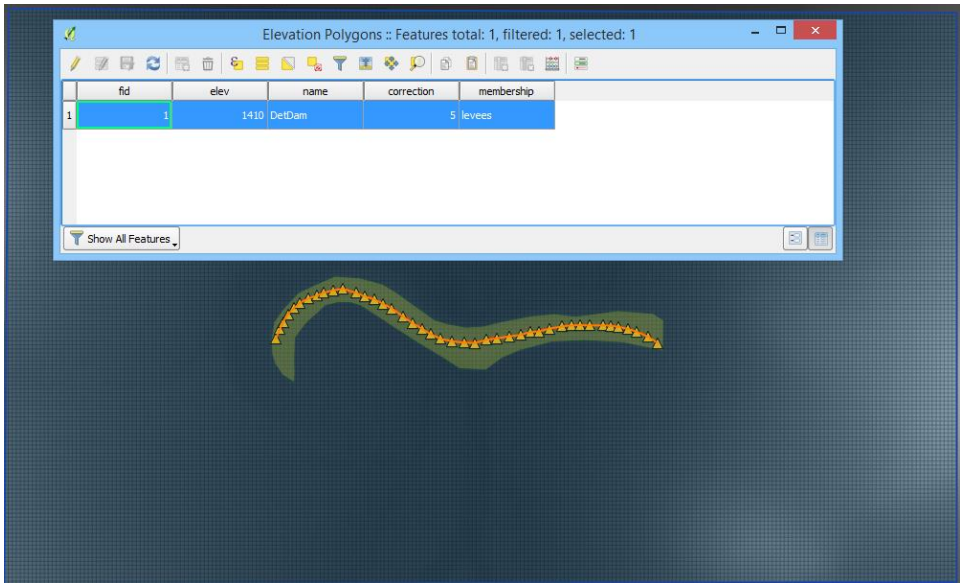


Levee from Elevation Polygon

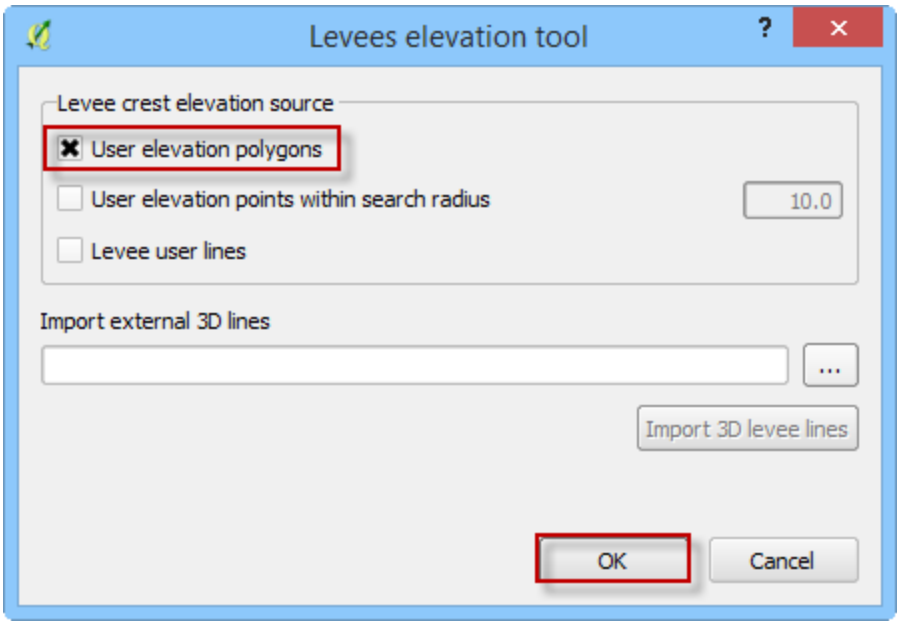
Digitize a polygon feature in the *Elevation Polygons* layer. Select the layer and click the *Toggle Editing* pencil. Add a polygon and assign the levee crest elevation. *The levee line feature must also be present for the levee position to be applied. See the instructions above.

The attributes include:

- Fid – Field ID automatically entered
- Elev – crest elevation
- Name – feature name
- Correction – adds to the crest elevation (optional)
- Membership – Set this to “levees”



Click the *Levee Elevation Tool* icon and select *User elevation polygons* and click *OK*. The Plugin will build the levee into a schematic layer, set the elevation data for the crest and apply a correction if used. The data can be reviewed in the attribute table of the levee layer. If the Levee user lines is also checked, that layer will be used to set crest elevations where polygons are not covering the levee.

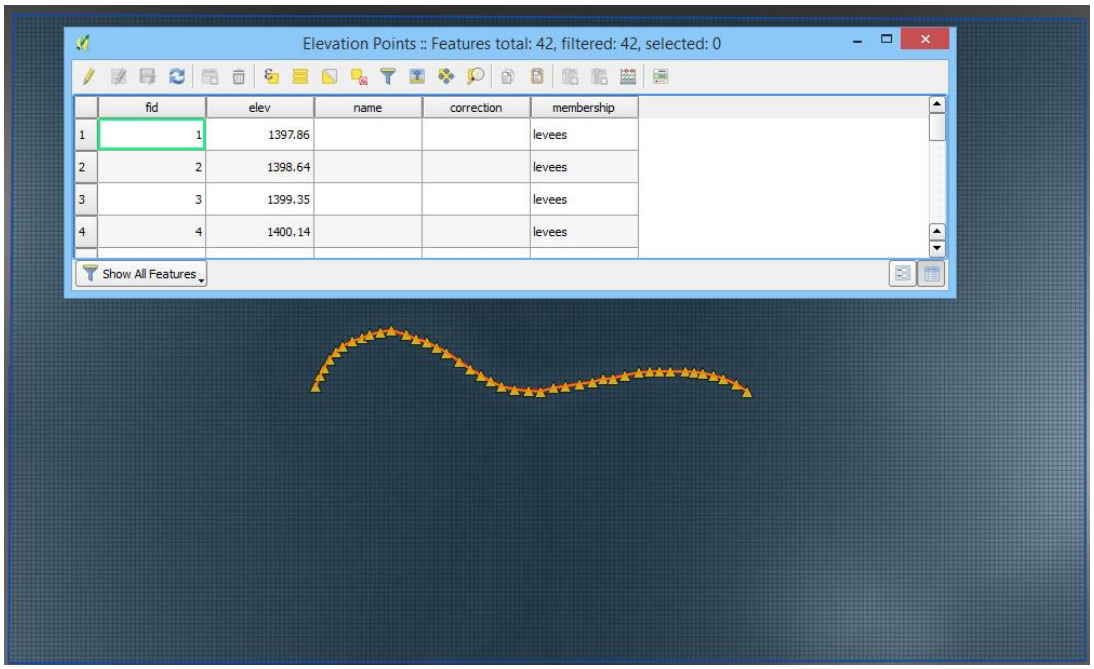


Levee from Elevation Points within Search Radius

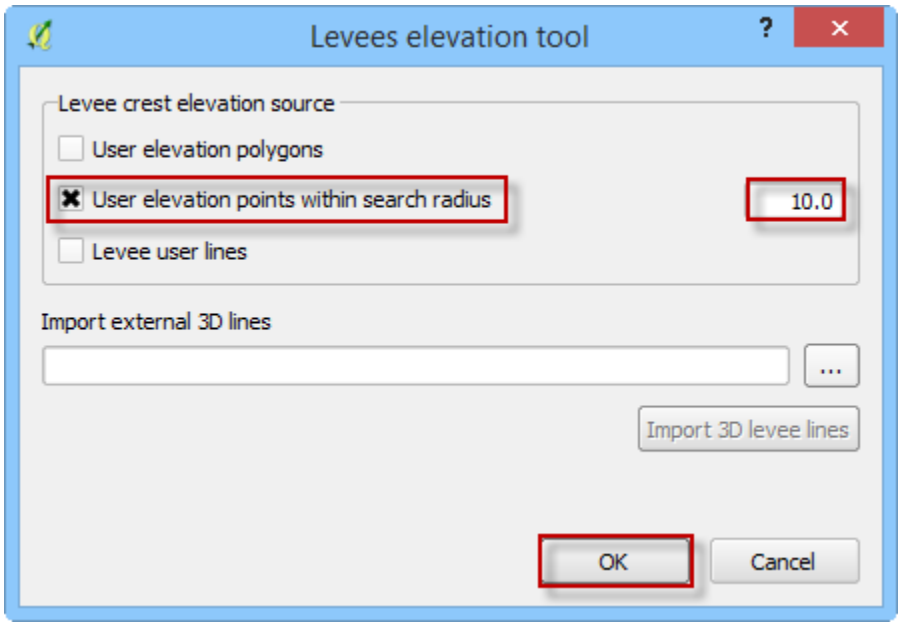
First digitize points in the *Elevation Points* layer, then select the layer and click the *Toggle Editing* pencil. Add points along the crest of the levee and assign the levee crest elevation or an elevation adjustment (correction). The points should be within the search radius buffer from the *Levee Line*. The points can be snapped to the *Levee Line*.

The attributes include:

- Fid – Field ID, automatically assigned
- Elev – crest elevation
- Name – feature name (optional)
- Correction – adds to the crest elevation (optional)
- Membership – Set this to “levees”



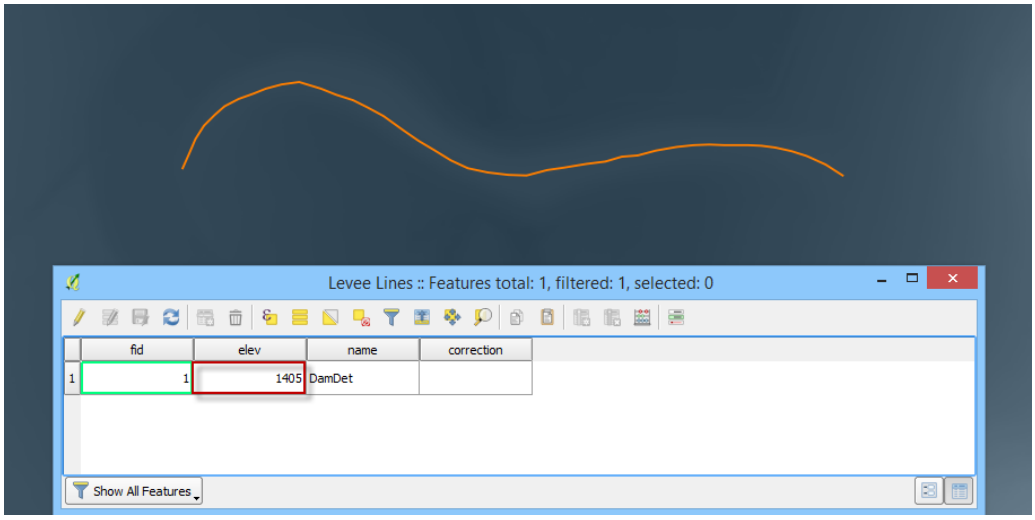
Click the *Levee Elevation Tool* icon and select *User elevation points within search radius* and click *OK*. The plugin will schematize the *Levee Lines* layer, set the elevation data for the crest and apply a correction if used. The levee crest elevation is interpolated between the two points. If the Levee user lines is also checked, that layer will be used to set crest elevations where points are not connected to the levee.



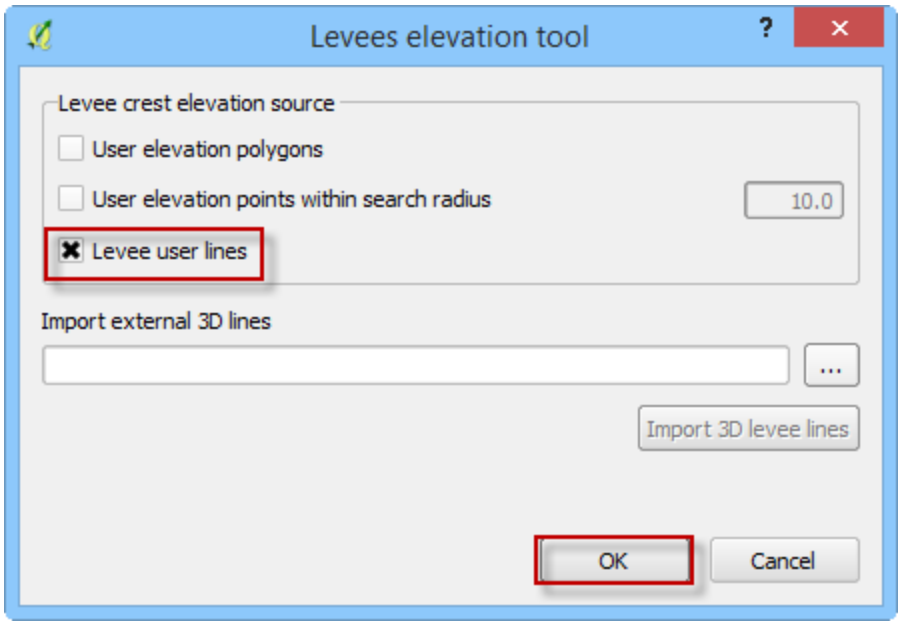
Levee from Levee User Lines

Digitize the levee line in the *Levee Lines* layer. Select the layer and click the *Toggle Editing* pencil. Add a line feature along the center of the crest of the levee and assign the levee crest elevation or a correction. The attributes include:

- Fid – Field ID, automatically assigned.
- Elev – crest elevation
- Name – feature name (optional)
- Correction – adds to the crest elevation (optional)
- Membership – Set this to "levees"



Click the *Levee Elevation Tool* icon and select *User elevation points within search radius* and click *OK*. The plugin will schematize the *Levee Lines* layer, set the elevation data for the crest and apply a correction if used. The levee crest elevation is interpolated between the two points.



Levee from Import External 3D Lines

The levee data comes from an external point text file with a *.xyz extension:

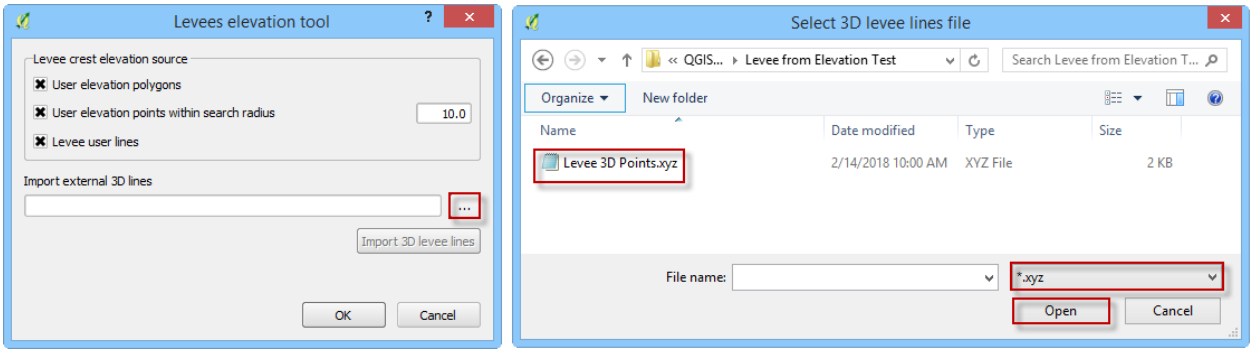
- Xcoord –x coordinate of the center of the levee crest
- Ycoord – y coordinate of the center of the levee crest
- Z – crest elevation of the levee

The levee points should be in order from one side of the levee to the other. The direction is not important. Two levees should be separated by a blank line (text file carriage return).

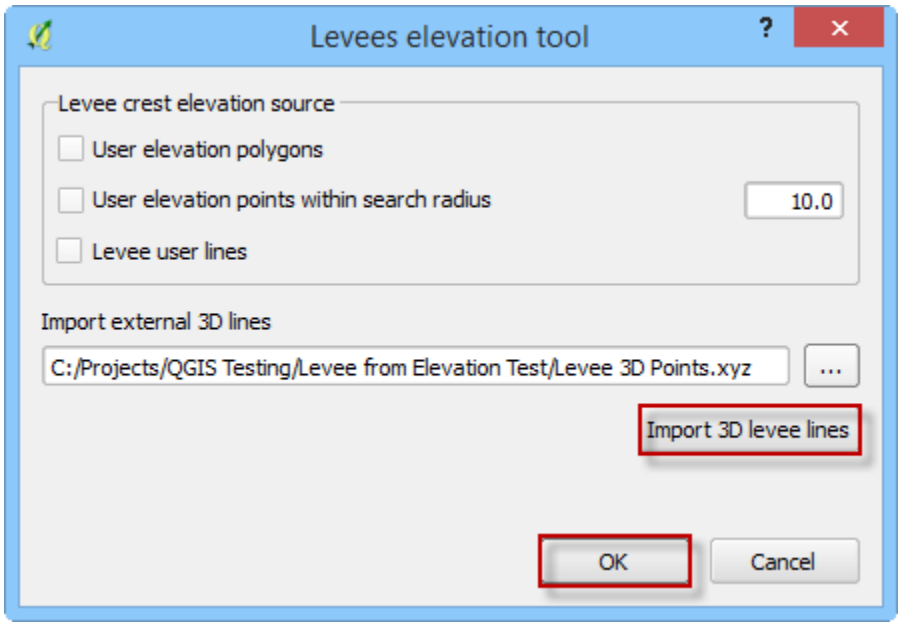
Levee 3D Points.xyz

1	652017.605426898	946357.211492341	1397.86
2	652028.138883151	946380.91176891	1398.64
3	652036.038975341	946399.345317353	1399.35
4	652048.328007636	946418.656653816	1400.14
5	652064.128192015	946434.456838196	1400.48
6	652078.172800352	946446.745870491	1400.67
7	652098.361924837	946457.279326744	1400.31
8	652120.306625364	946465.179418934	1399.27

Call the levee data from the *Levee Elevation Tool* by clicking the "…" button under *Import external 3D lines*.



Once the data is identified, click the *Import 3D levee lines* button. The imported levees are written to the elevation points and *Levee Lines User Layer*. Click *OK* to schematize the levee.



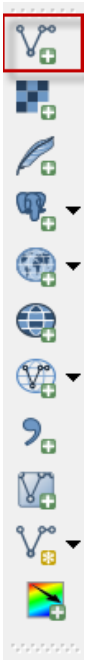
Hazus Tool

The Hazus tool is used to generate water surface elevation or flow depth rasters for the FEMA Hazus program.

Building Layer

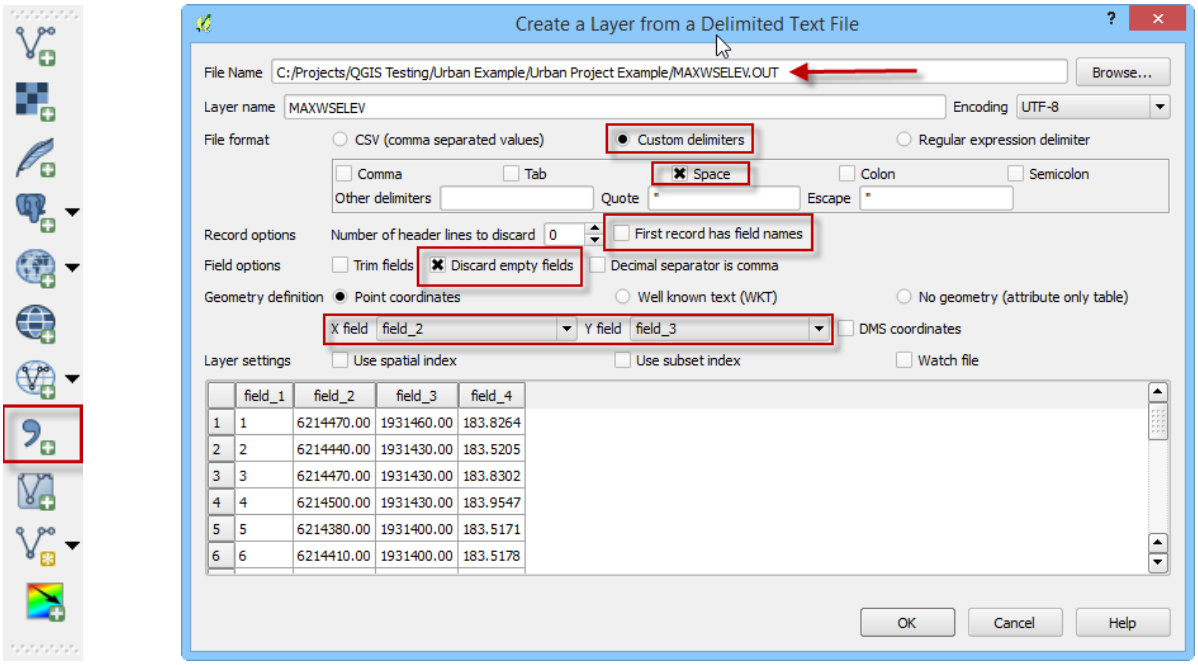
Add building layer from a shapefile.

Click the *Add a Vector Layer* button. Navigate to the building shapefile and add it to the map. If the layer does not have a CRS assigned, a prompt will allow it to be assigned.

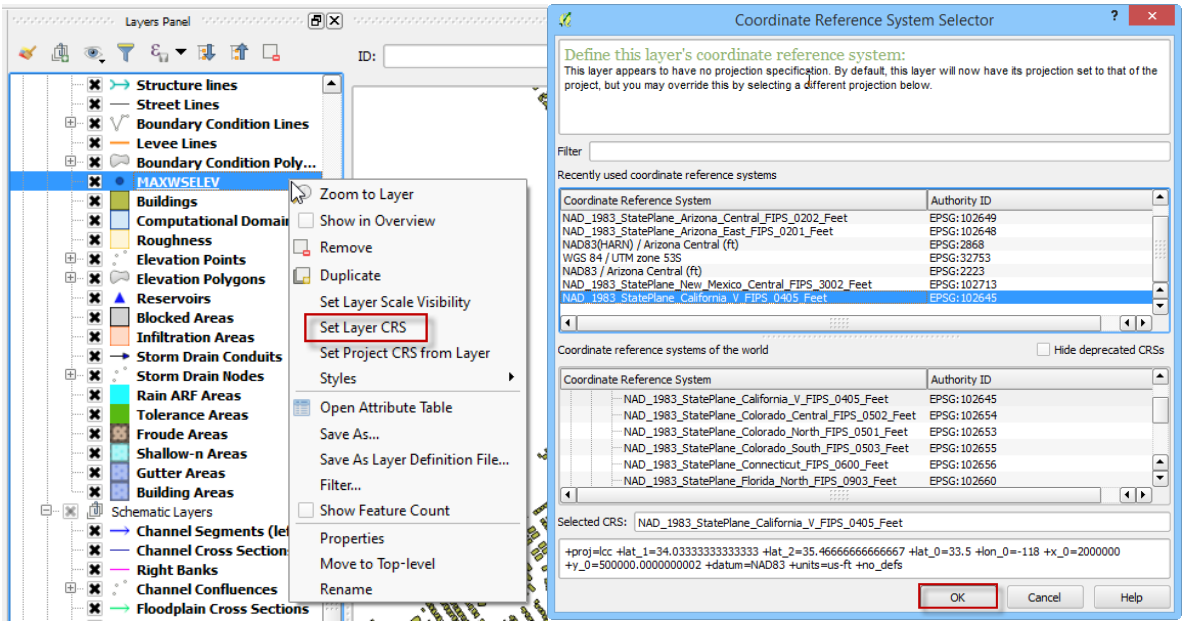


Import Depth and Water Surface Layers.

Add a depth layer and a water surface layer. Click *Layer>Add Layer>Add Delimited Text Layer* to import the files DEPTH.OUT and MAXWSELEV.OUT.



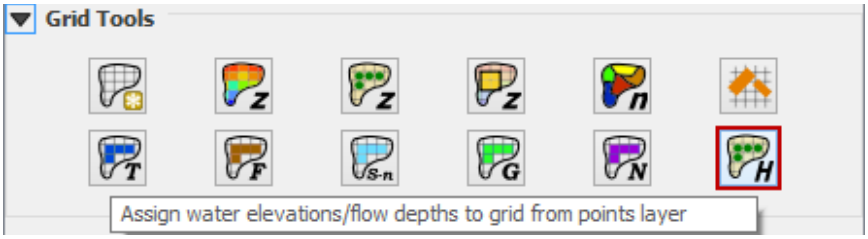
Right click the layer and assign the Coordinate Reference System.



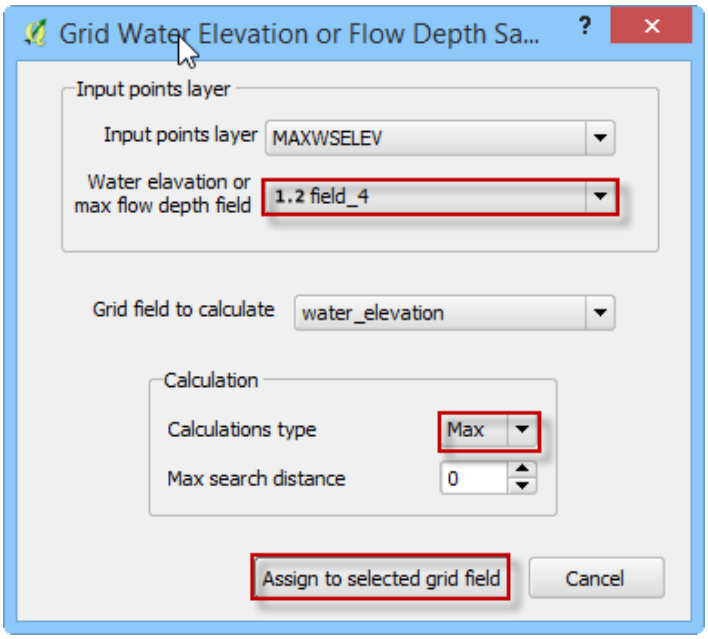
Repeat this process for the DEPFP.OUT file.

Assign Water Elevation and Depth to the Grid Layer

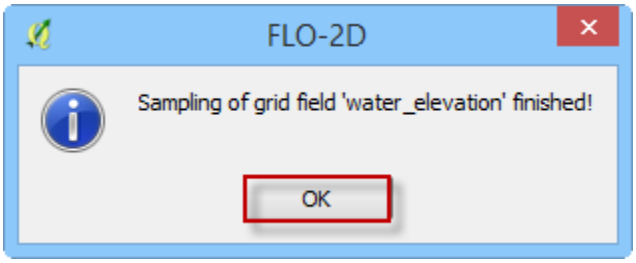
On the *Grid Tools* widget select the *Assign water elevations/flow depths to grid from points layer* icon.



Edit the dialog box as shown below and click the *Assign to selected grid field* button.

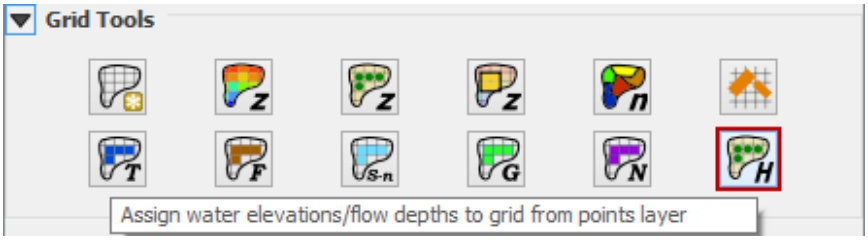


Click *OK* to close the dialog box.

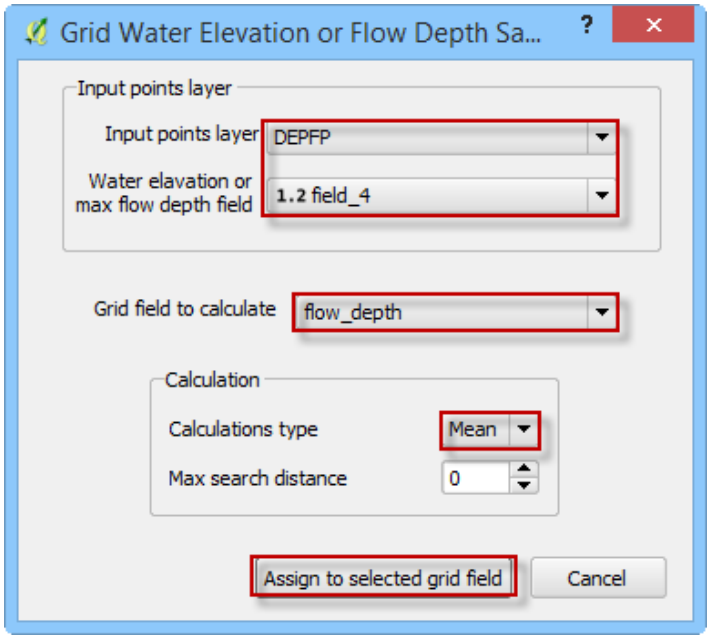


Repeat the process for the Depth layer.

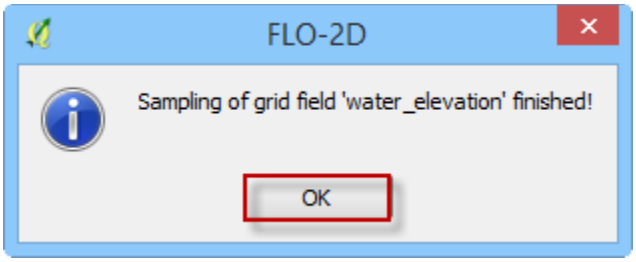
On the *Grid Tools* widget select the *Assign water elevations/flow depths to grid from points layer* button.



Edit the dialog box as shown below and click the *Assign to selected grid field* button.

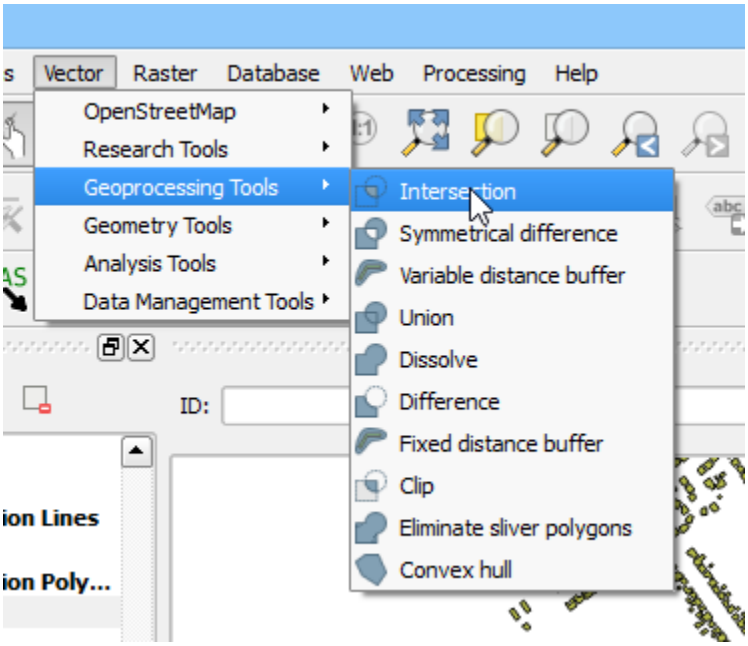


Click *OK* to close the dialog box.

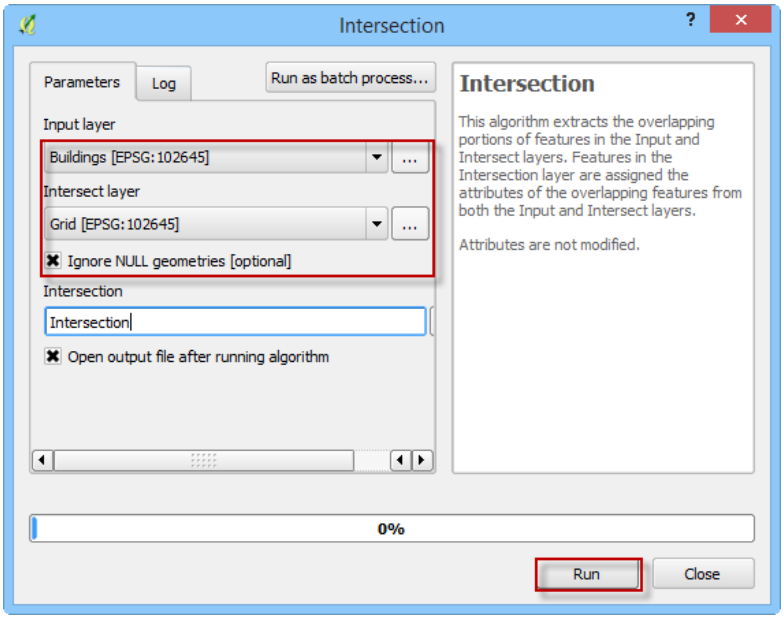


Intersect Building Layer to Grid

Use the *QGIS Vector* Menu to set up the intersection. Click *Vectors>GeoProcessing Tools>Intersection*.

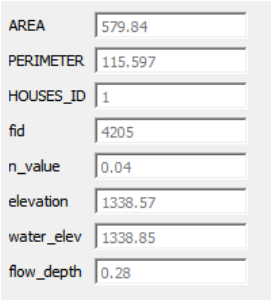


Set up the intersection dialog box as shown below. Click *Run* to make the intersection. This process adds the Intersection layer to the map automatically.

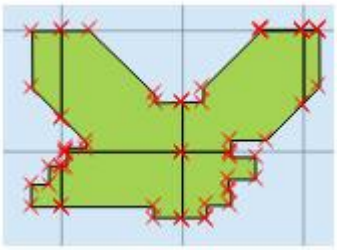


Review Intersection Layer

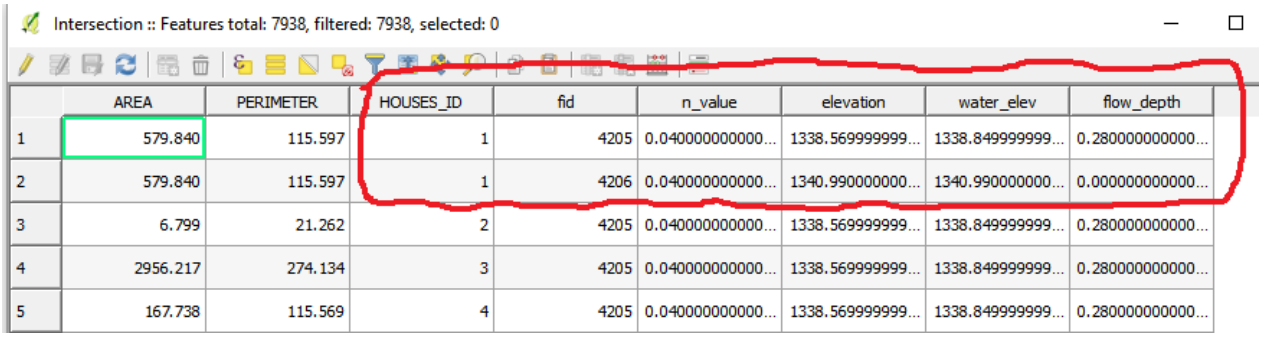
The new *Intersection* layer has fields from both the *Buildings* and *Grid* layers:



Each building polygon that intersects the grid has several partitions (polygons) with different elevations. The following building has 7 partitions with different data from each grid:



Each partition of the building has, different field values. For example, HOUSE_ID "1" in the Features Table, has different 'elevation', 'water_elev', and 'flow_depth':

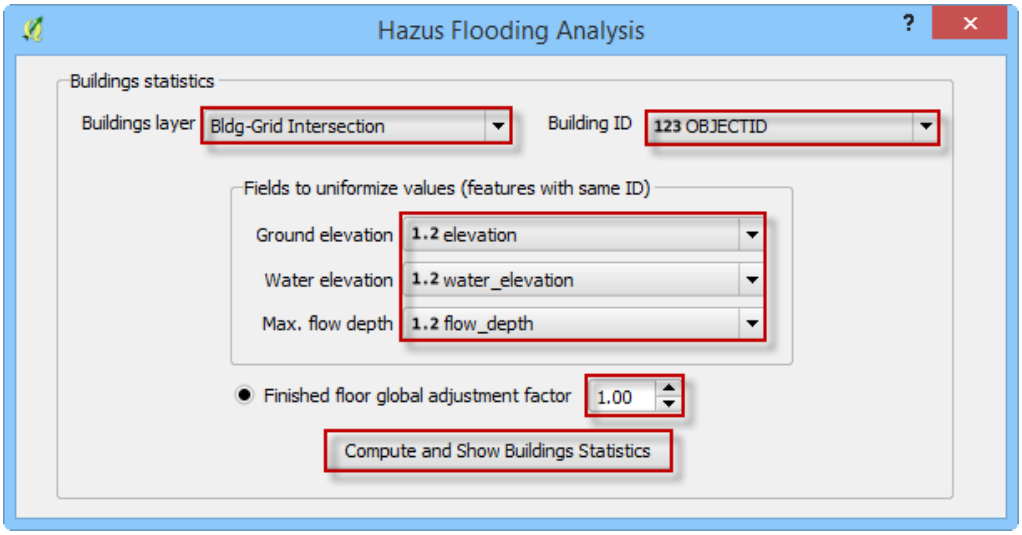


	AREA	PERIMETER	HOUSES_ID	fid	n_value	elevation	water_elev	flow_depth
1	579.840	115.597	1	4205	0.040000000000...	1338.569999999...	1338.849999999...	0.280000000000...
2	579.840	115.597	1	4206	0.040000000000...	1340.990000000...	1340.990000000...	0.000000000000...
3	6.799	21.262	2	4205	0.040000000000...	1338.569999999...	1338.849999999...	0.280000000000...
4	2956.217	274.134	3	4205	0.040000000000...	1338.569999999...	1338.849999999...	0.280000000000...
5	167.738	115.569	4	4205	0.040000000000...	1338.569999999...	1338.849999999...	0.280000000000...

Homogenize the Intersection layer.

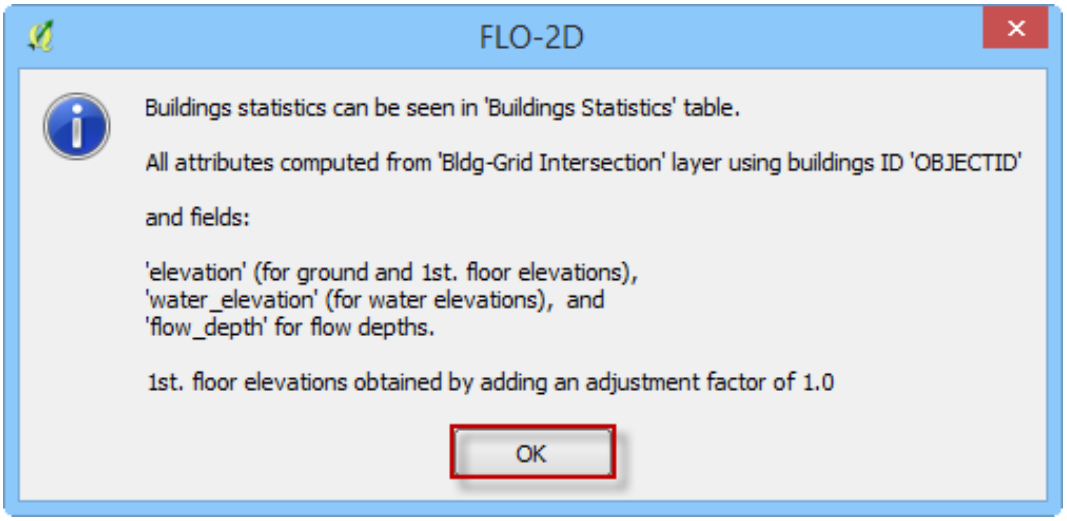
Select the HAZUS icon in the *FLO-2D Toolbar*.

Fill the dialog box as shown below and click the *Compute and Show Building Statistics* button.



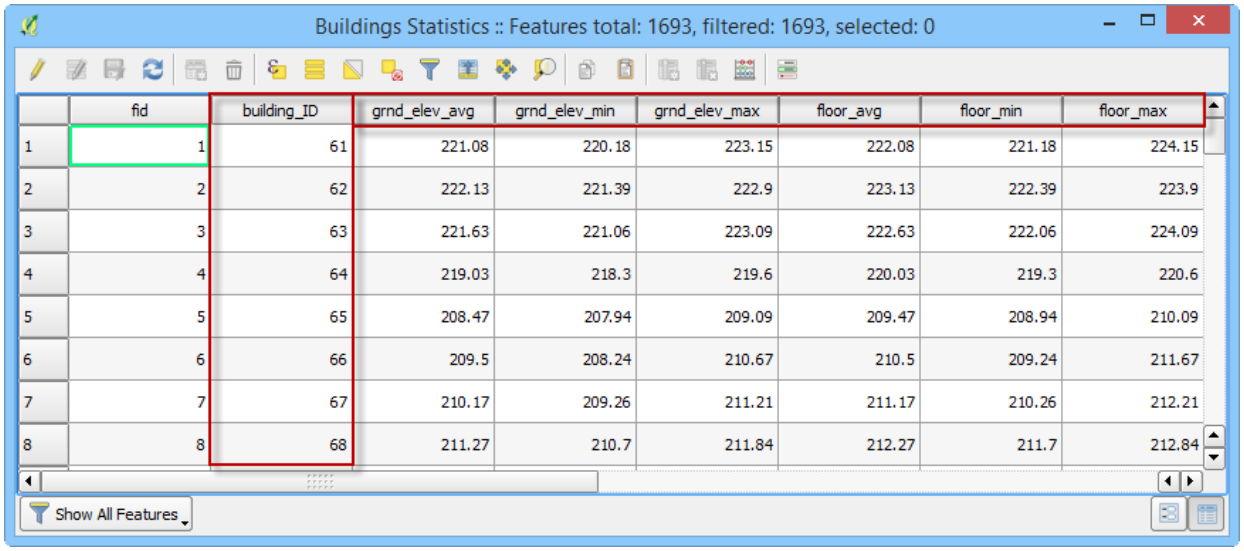
Note: The 'Finished floor global adjustment factor' value will be added to the ground elevations, if selected.

Click *OK* to close the message dialog box.

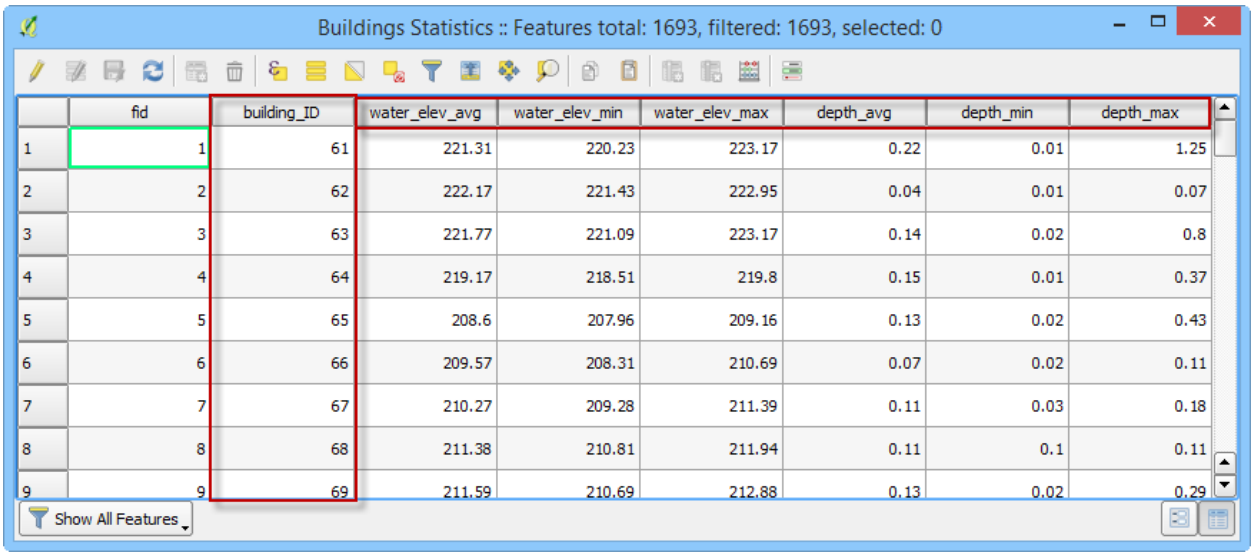


The Hazus tool calculates the statistics of the buildings polygons. It computes the following data for each building.

- Ground elevation (min, max, mean);
- First floor elevation (min, max, mean);
- Water surface elevation (min, max, mean);
- Depth (min, max, mean).



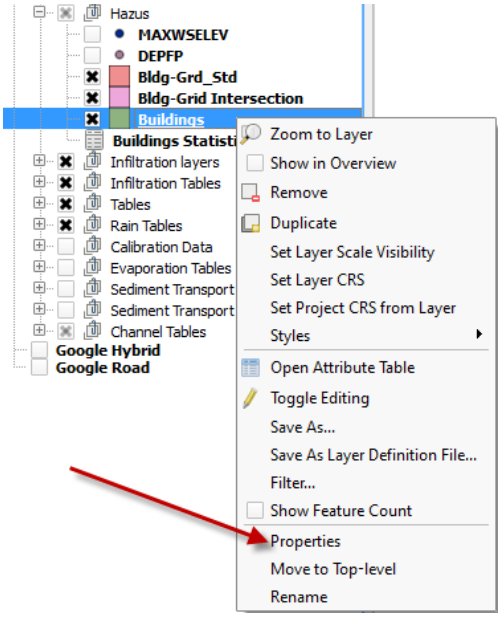
	fid	building_ID	grnd_elev_avg	grnd_elev_min	grnd_elev_max	floor_avg	floor_min	floor_max
1	1	61	221.08	220.18	223.15	222.08	221.18	224.15
2	2	62	222.13	221.39	222.9	223.13	222.39	223.9
3	3	63	221.63	221.06	223.09	222.63	222.06	224.09
4	4	64	219.03	218.3	219.6	220.03	219.3	220.6
5	5	65	208.47	207.94	209.09	209.47	208.94	210.09
6	6	66	209.5	208.24	210.67	210.5	209.24	211.67
7	7	67	210.17	209.26	211.21	211.17	210.26	212.21
8	8	68	211.27	210.7	211.84	212.27	211.7	212.84

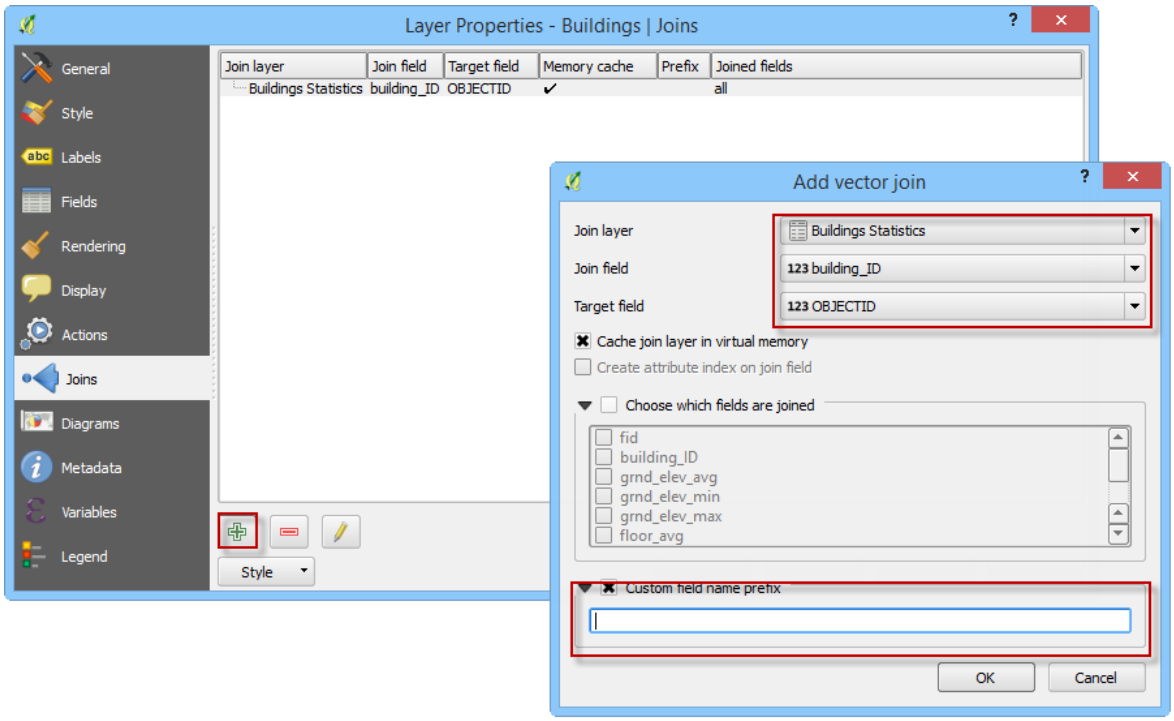


	fid	building_ID	water_elev_avg	water_elev_min	water_elev_max	depth_avg	depth_min	depth_max
1	1	61	221.31	220.23	223.17	0.22	0.01	1.25
2	2	62	222.17	221.43	222.95	0.04	0.01	0.07
3	3	63	221.77	221.09	223.17	0.14	0.02	0.8
4	4	64	219.17	218.51	219.8	0.15	0.01	0.37
5	5	65	208.6	207.96	209.16	0.13	0.02	0.43
6	6	66	209.57	208.31	210.69	0.07	0.02	0.11
7	7	67	210.27	209.28	211.39	0.11	0.03	0.18
8	8	68	211.38	210.81	211.94	0.11	0.1	0.11
9	9	69	211.59	210.69	212.88	0.13	0.02	0.29

Join Building Statistics Table to Building Polygons

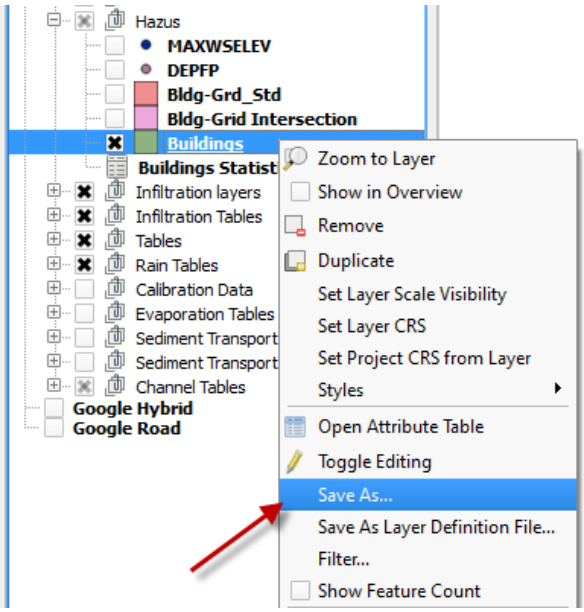
Right click the *Buildings* layer and click *Properties*. *Add a Join* to the layer. Click *OK* and Close the *Properties* window.

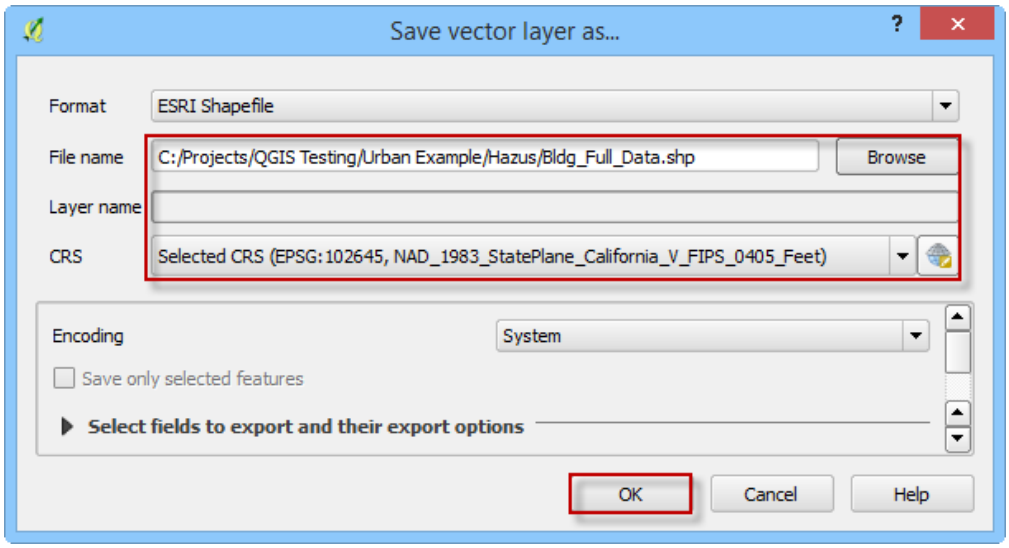




Save the *Buildings* Layer to a shapefile. Select the *Save As…* to a location and name the file.

Note: The style of this new layer can be edited to help the user review the data. The attributes can be sorted and arranged to help track outliers or bad data. Use the field calculator to perform additional statistical analysis on the data in this layer.



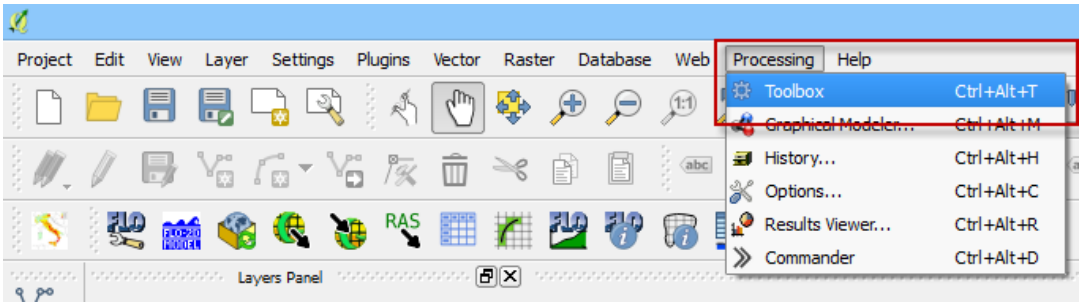


Now, the Buildings Shapefile has “join” fields from the Buildings Statistics table:

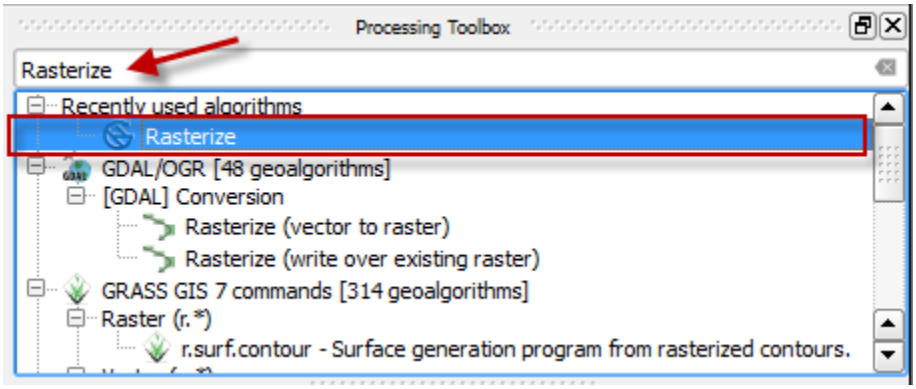
Buildings Statistics_fid	1489
Buildings Statistics_grnd_elev_avg	1290.96
Buildings Statistics_grnd_elev_min	1285.16
Buildings Statistics_grnd_elev_max	1293.24
Buildings Statistics_floor_avg	1291.96
Buildings Statistics_floor_min	1286.16
Buildings Statistics_floor_max	1294.24
Buildings Statistics_water_elev_avg	1292.13
Buildings Statistics_water_elev_min	1289.57
Buildings Statistics_water_elev_max	1293.39
Buildings Statistics_depth_avg	1.17
Buildings Statistics_depth_min	0.03
Buildings Statistics_depth_max	4.41

Rasterize the Buildings

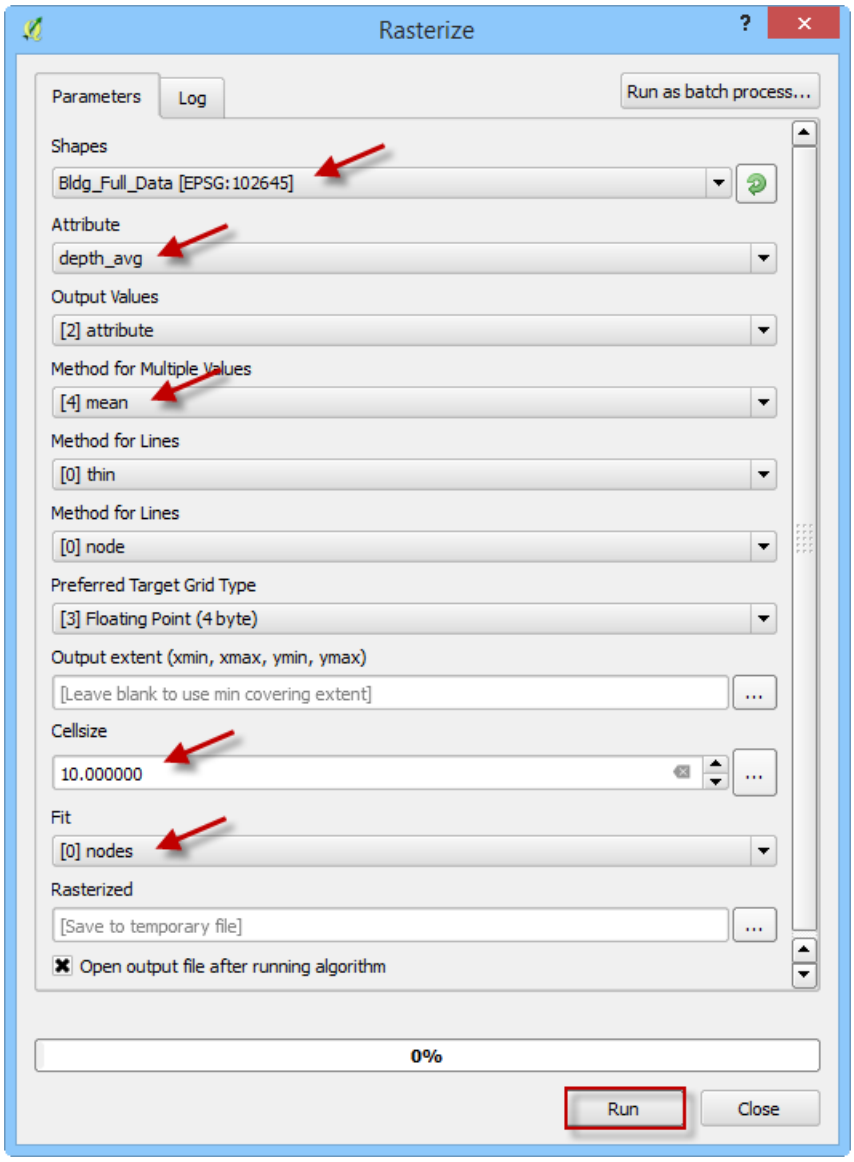
On the Main QGIS Menu, click *Processing>Toolbox*.



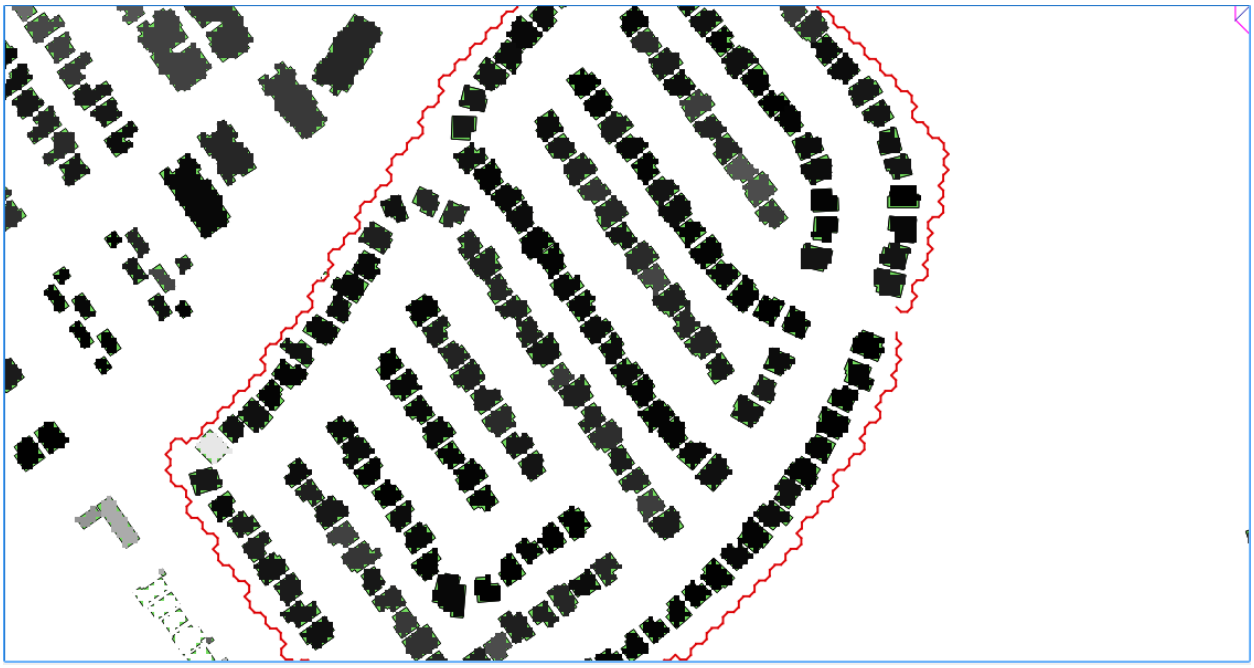
Enter the search term *Rasterize* in the Processing Toolbox search field. Double click the *Saga Rasterize* tool. Saga>Raster Creation Tools>Rasterize.



Change the dialog box as shown below and click *Run*.

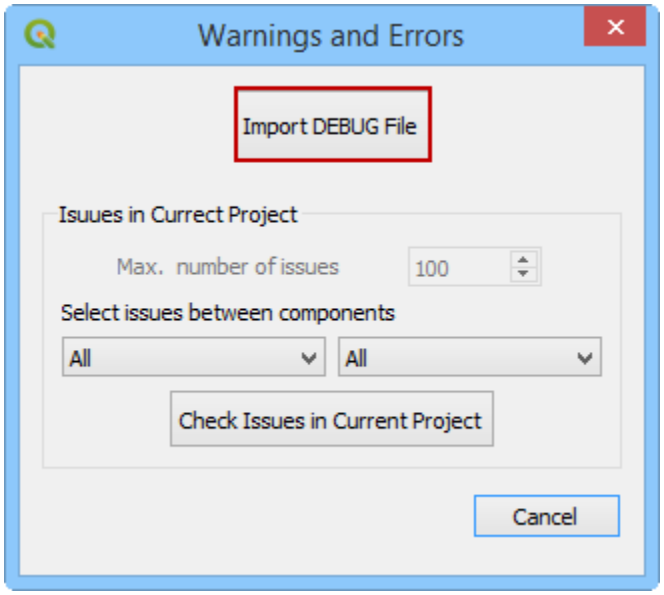


This example uses 10 ft. pixel resolution. The user can change this value to the desired resolution to better fit the buildings. This raster can be used with the FEMA Hazus software. Any other rasters that Hazus requires can be generated with the same methodology.

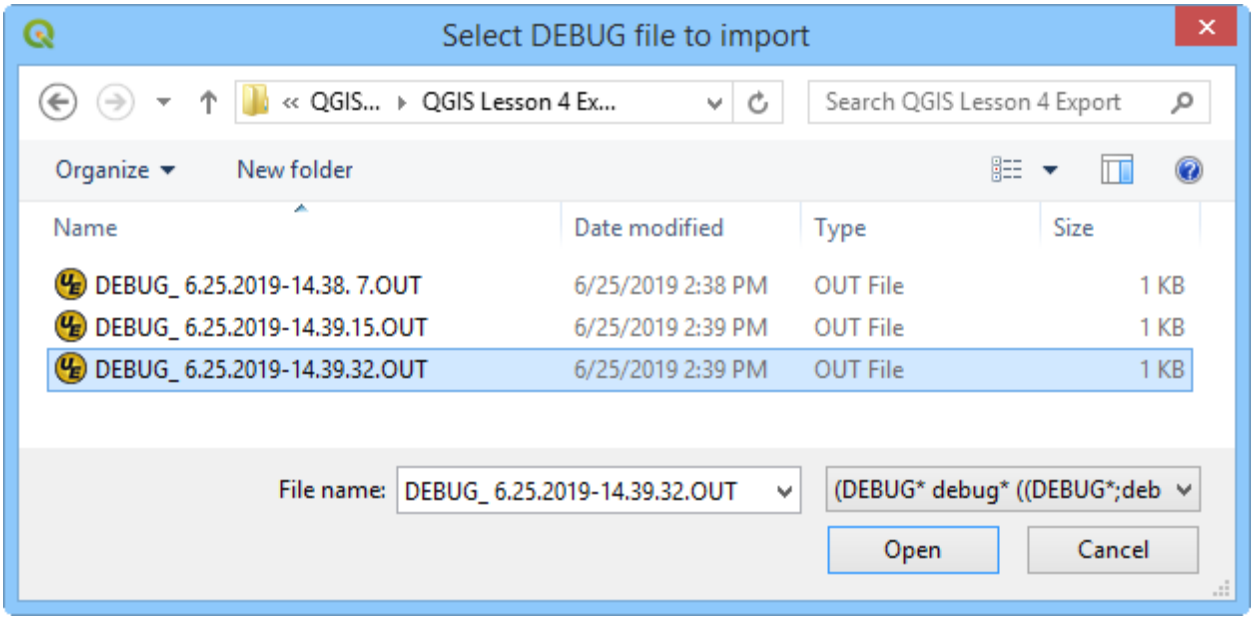


Data Warnings and Errors

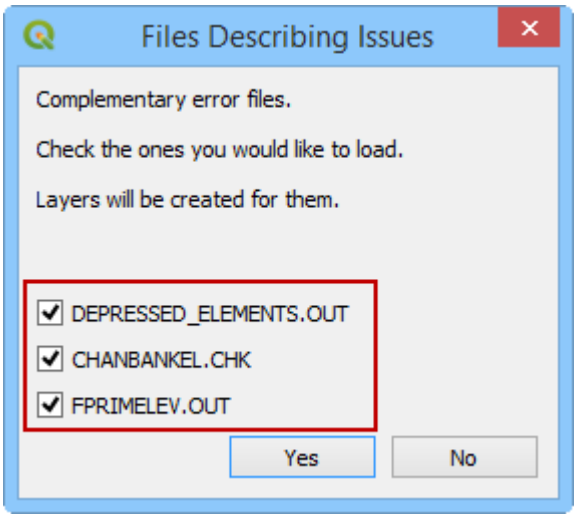
The Data Warnings and Error button opens a system that helps the user debug data files and search for data conflicts. Click the Error and Warning button to open the import dialog box. To import the Debug files, click the Import DEBUG File button.



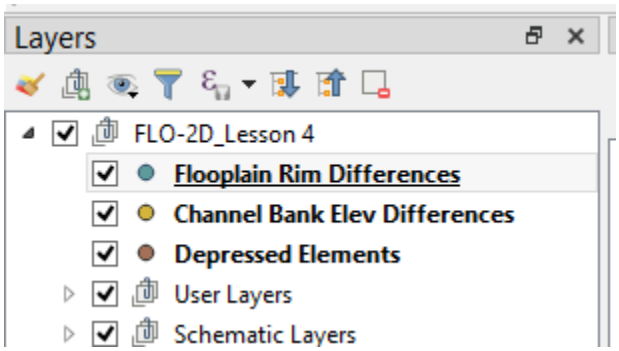
The DEBUG file will have a date and timestamp on the file so the user can find the most up to date file.



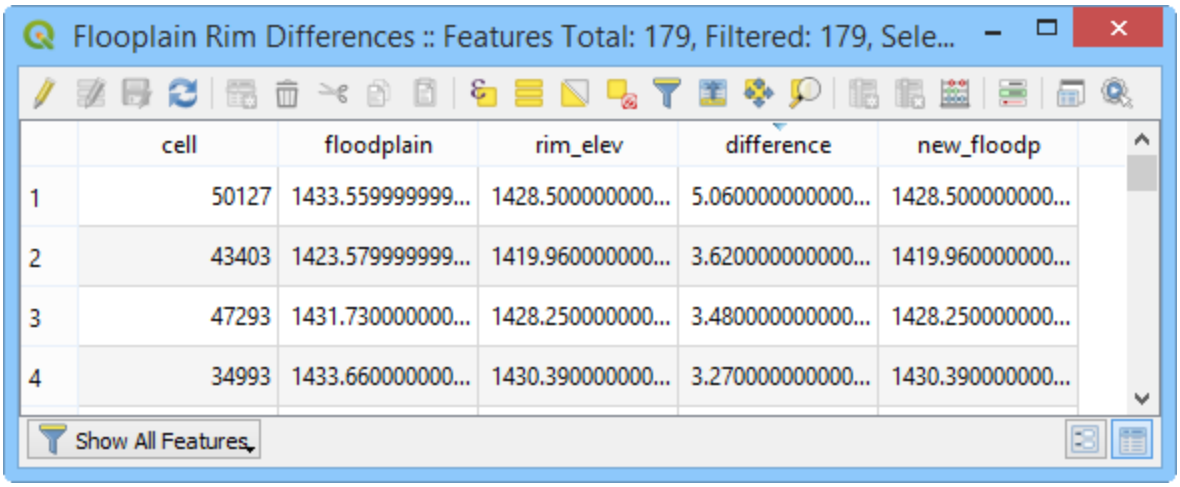
The import process will include several files that can be used to help users review surface features such as rim elevations, depressed elements and channel – floodplain interface. Click Yes to load the Errors and Warning Dialog box and import the review files.



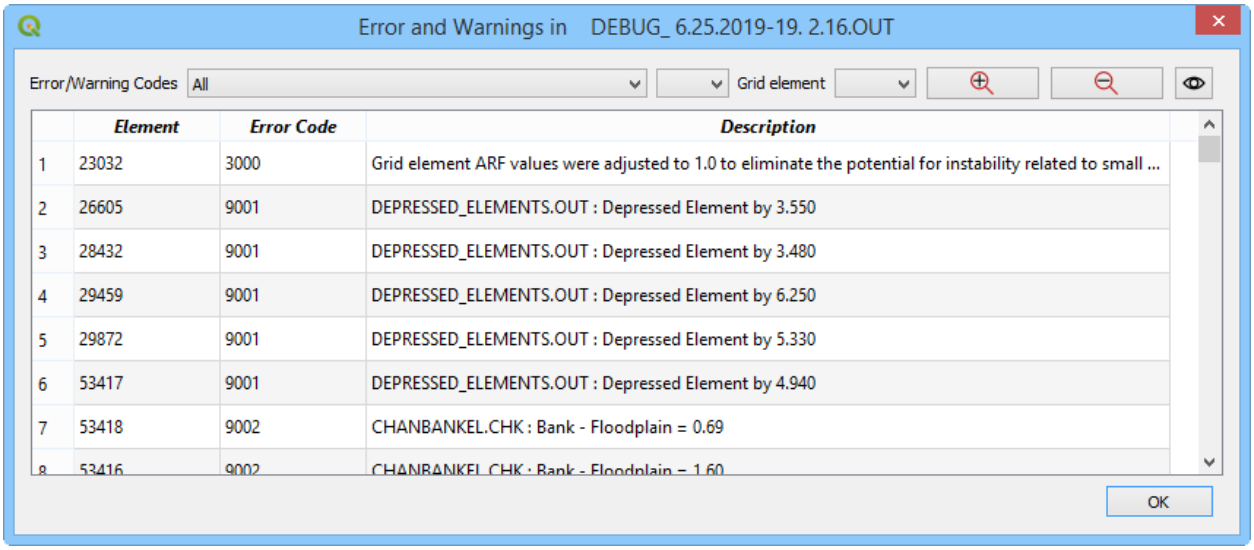
The layers show points where there are differences between channel bank and floodplain bank elevations, rim and floodplain inlet elevations, and depressed elements.



Each layer has an attribute table that can be sorted and used to find grid elements that may need elevation edits.



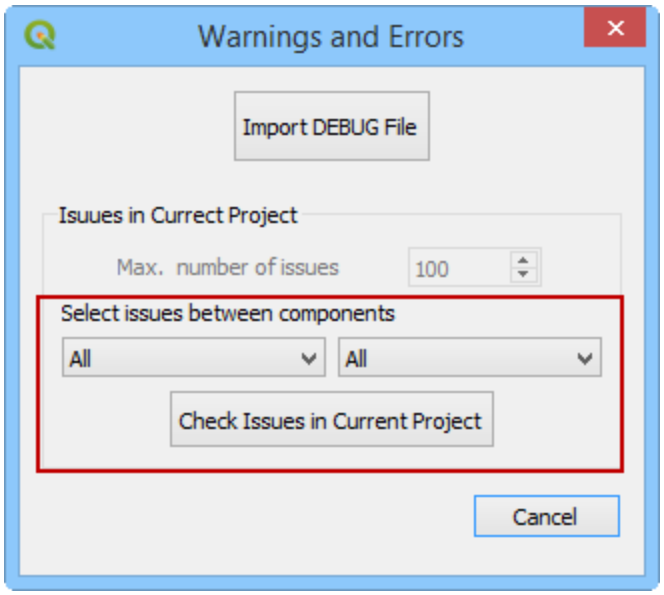
The Errors and Warnings Dialog box shows all Errors, Conflicts, and Warnings created by the file checking program.



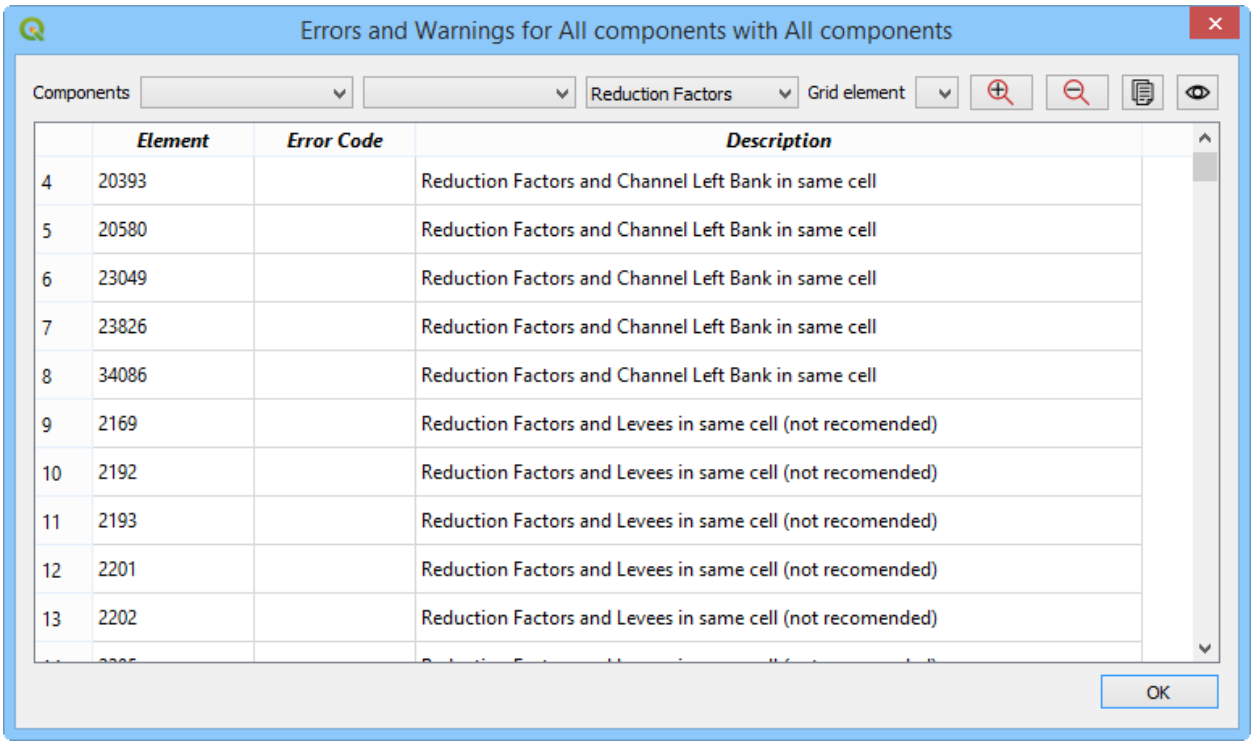
The Dialog box can be used to sort by codes or elements or error types. There are highlight and zoom options as well.

Click OK to close the box.

The Warnings and Errors tool can also be used to check for component conflicts in the QGIS Dataset. The conflicts can be compared individually or loaded completely for the full project.



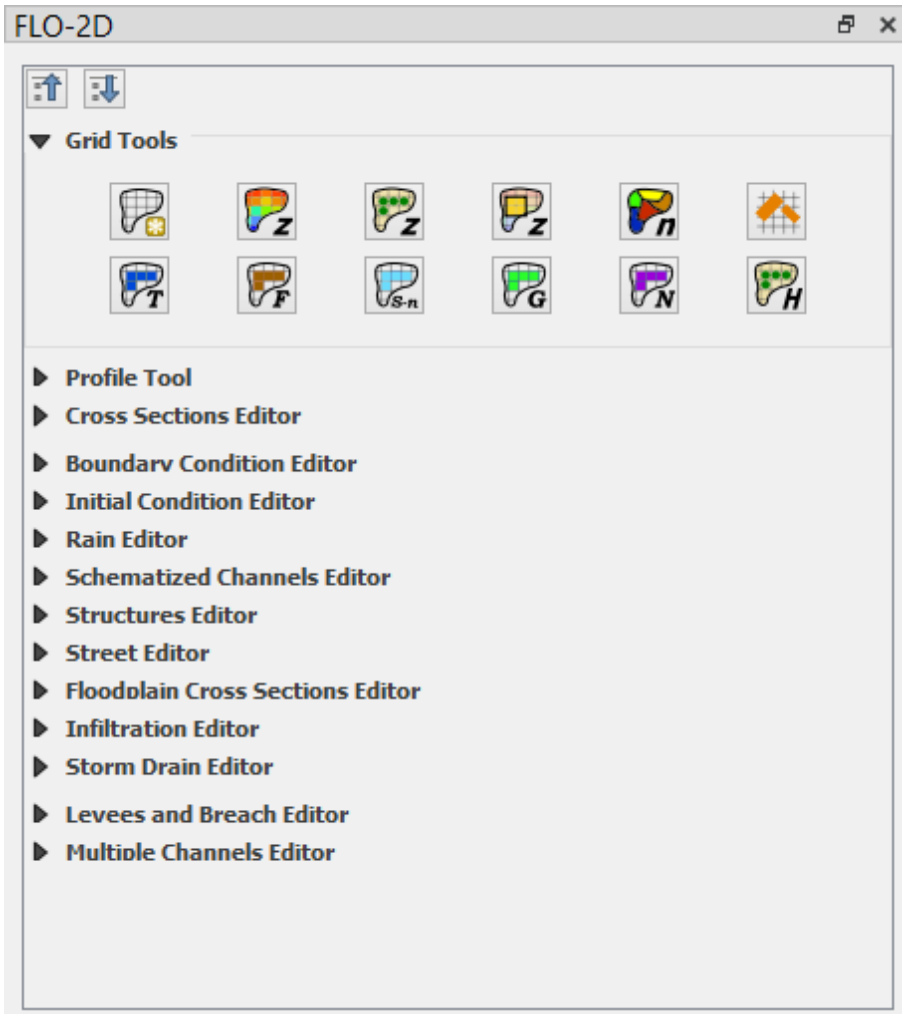
The Conflict checker is set up so that the user can sort the conflict issues by comparison tables.



FLO-2D Widgets

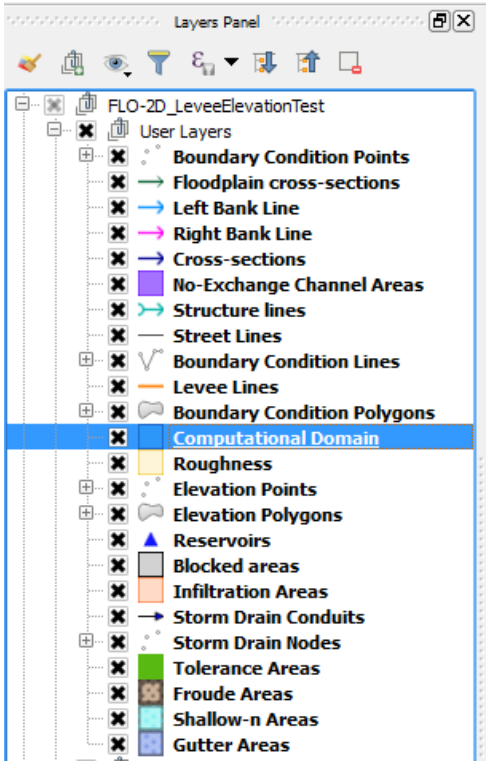
The *FLO-2D Widgets* are editor windows used to digitize features into the FLO-2D layers and edit and fill data tables. The QGIS sidebar is used to store the docked FLO-2D widgets. This widget panel can be un-docked and moved to a second window.

Note: If the docked panel is missing, reset it from the View/Panels menu.

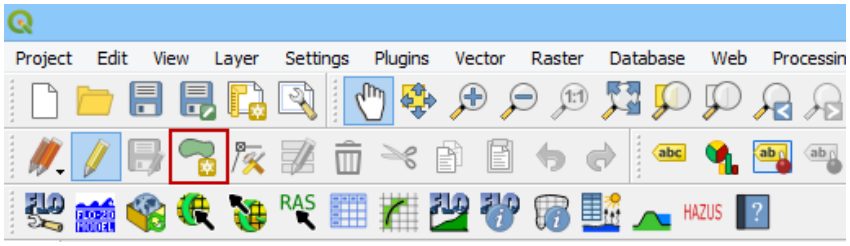


Create a Grid

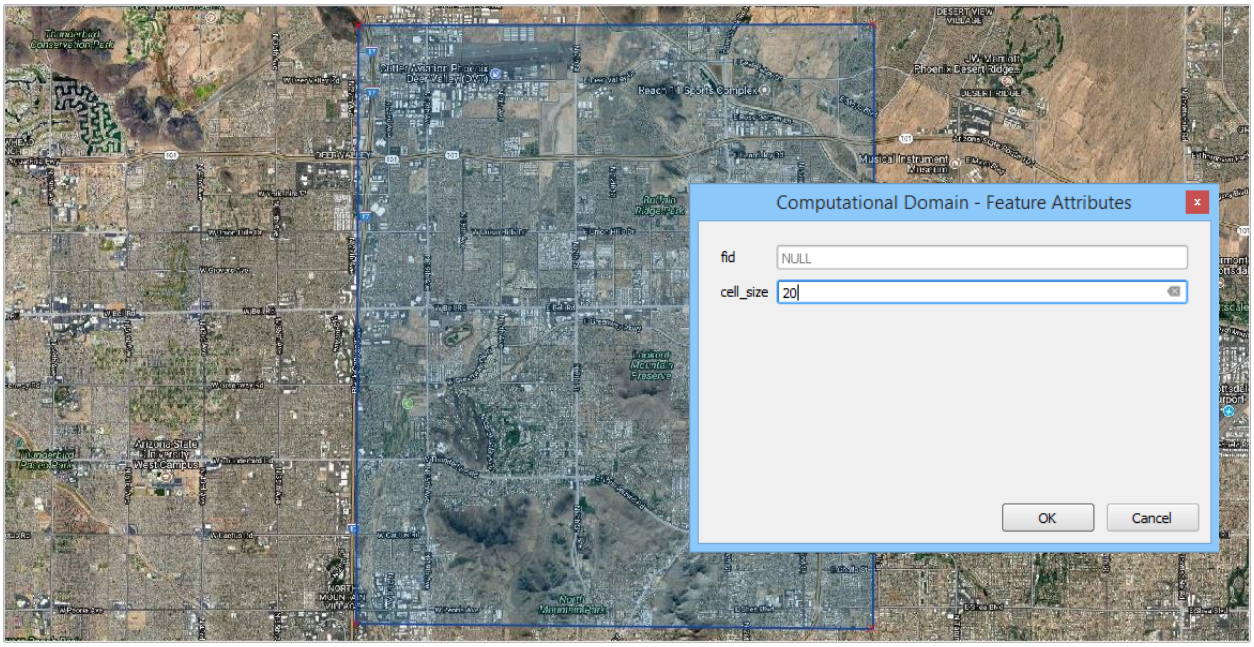
The first step in creating a FLO-2D grid system is to create the *Computational Domain*. Select the *Computational Domain* in the *Layers Panel>User Layers*.



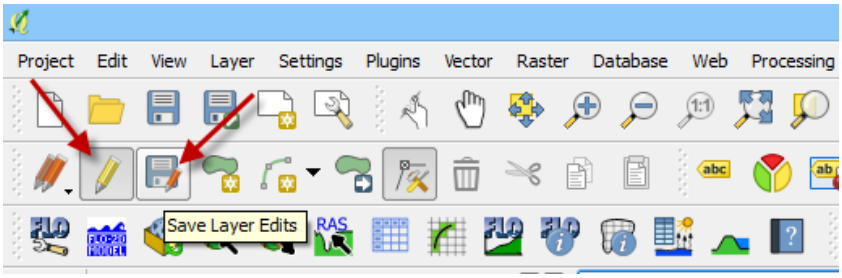
Select the *Toggle Editing* icon from the QGIS Toolbar to activate the editor and then click the *Add Feature* button to create a polygon. It is also possible to copy a polygon from a different layer into this layer.



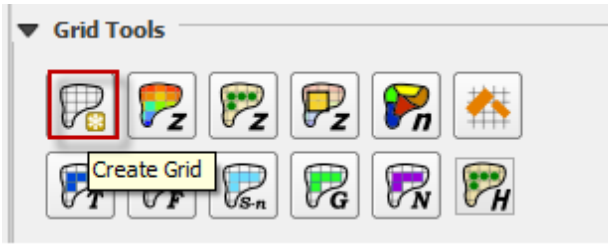
Digitize the polygon in the map canvas and right click to close the polygon. Set the grid element size and click *OK* to complete the polygon.



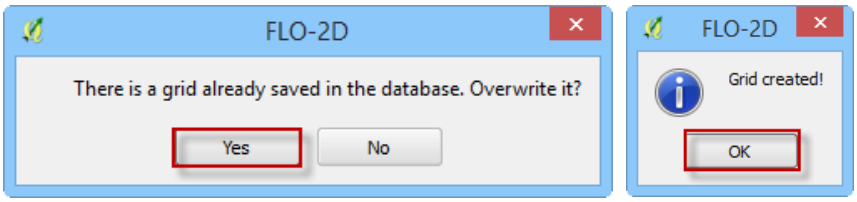
Save the layer and turn off the editor by clicking the Editor tool to toggle it off.



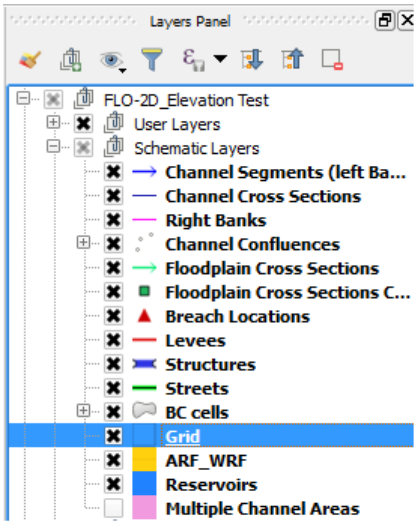
From the Grid Tools widget, select Create Grid.

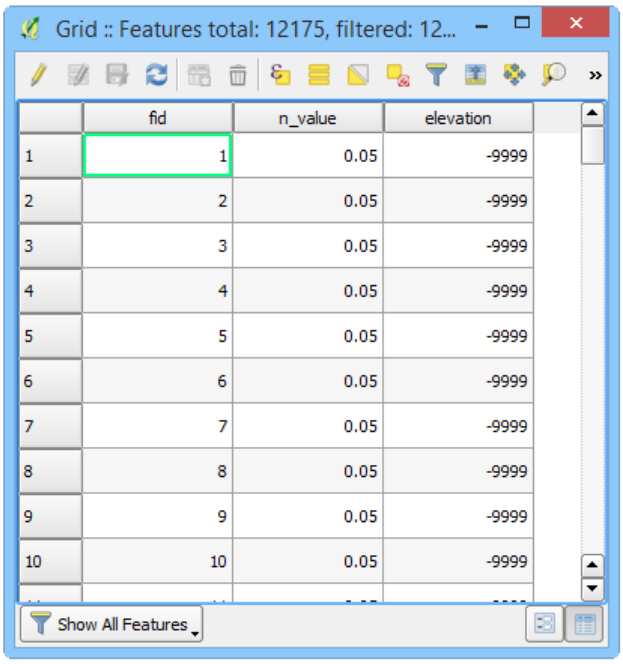


If this is a new project, the grid system will be created automatically. If this is a current project, the user will be asked to overwrite the current grid system. Click *Yes* to continue and *No* to cancel. Once the grid system is generated, the "Grid created!" message will appear. Click *OK* to close.



If the grid system is not as expected, edit the *Computational Domain* layer and repeat the *Create Grid* process. Each time the grid system is replaced, the elevation and roughness data are also reset and must be recalculated. Each time the grid system is replaced, it may be necessary to re-assign the *User Layers* to the *Schematic Layers*. The grid system data is saved to the *Grid Schematic Layer* as shown below.





Sample Elevation Data from Raster

To interpolate the elevation to a grid layer from a raster layer, use the *Sampling Grid Elevation from Raster Layer* icon.

Overview

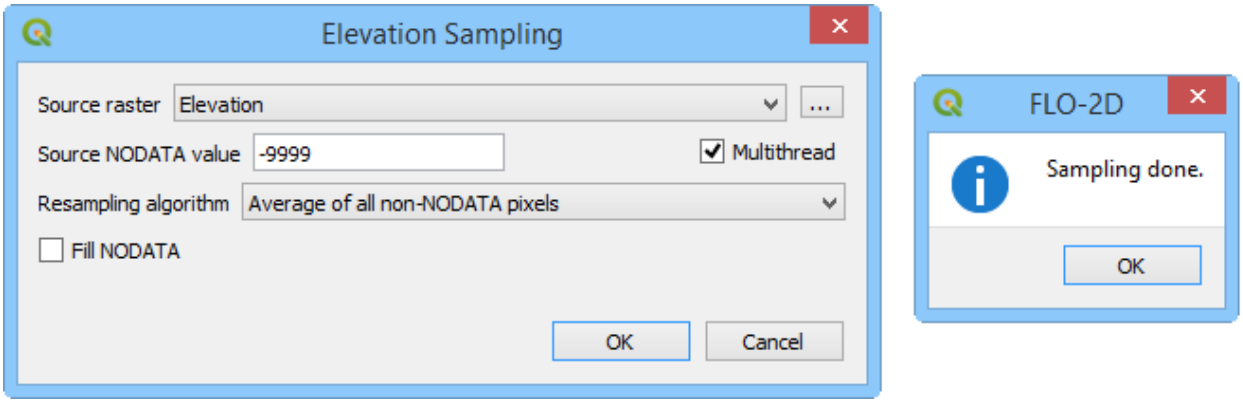
A basic understanding of a raster file is necessary to use this routine. A raster file is a georeferenced image file with band data assigned to pixels aligned in column and row. In this case, the bands are elevation. Important properties:

1. The raster must have the same coordinate reference system (CRS) as the project. If the CRS is missing or is set by the user, save the raster with the correct CRS.
2. The best resolution of the grid element elevation is achieved when the elevation raster pixel size is smaller than the grid element size.

3. The origin and layer extent may skew the raster warp and align. The best practice is to use an elevation raster that has the same origin and extent.

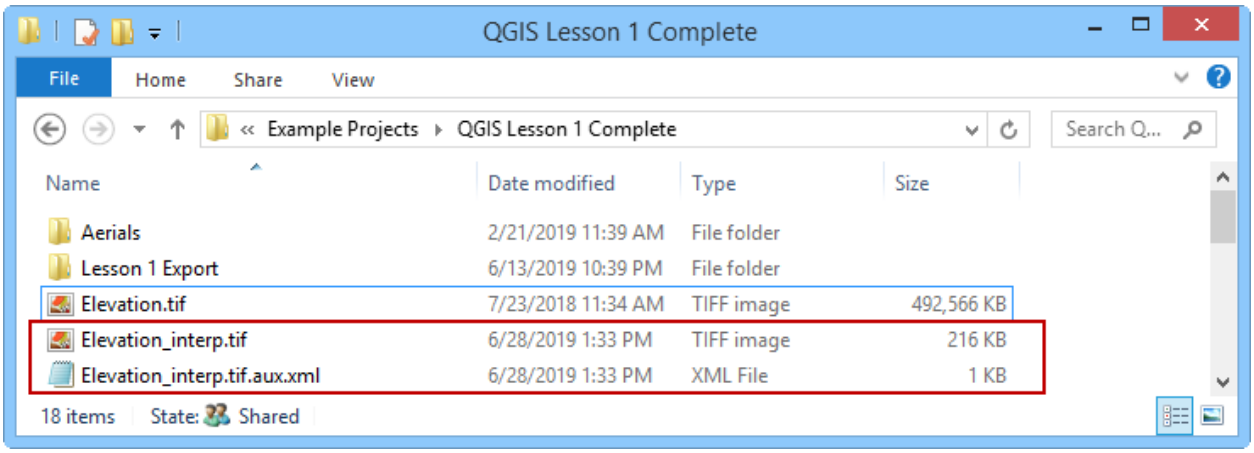
Sample Elevation

Click the *Sampling Grid Elevation Button* and enter the required data in the dialog fields. As shown below, when the elevation sample is complete, the *Sampling Done* dialog box will appear. Select the *Fill NoDATA* option to set the elevation of empty grid elements from neighbors.

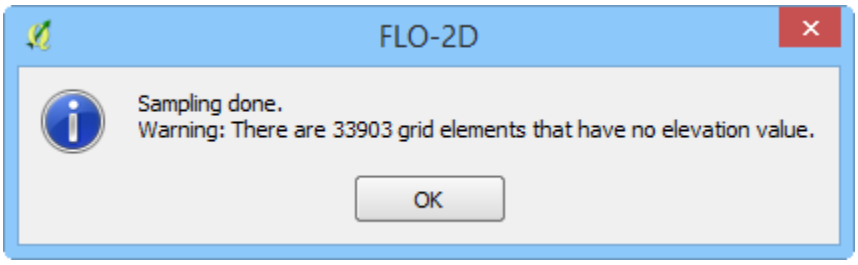


When the interpolation process is complete, the filename_interp.tif raster will be created in the project directory. This is the warped and realigned raster that was used to sample the elevation. It can be imported into the project at any time for review purposes.

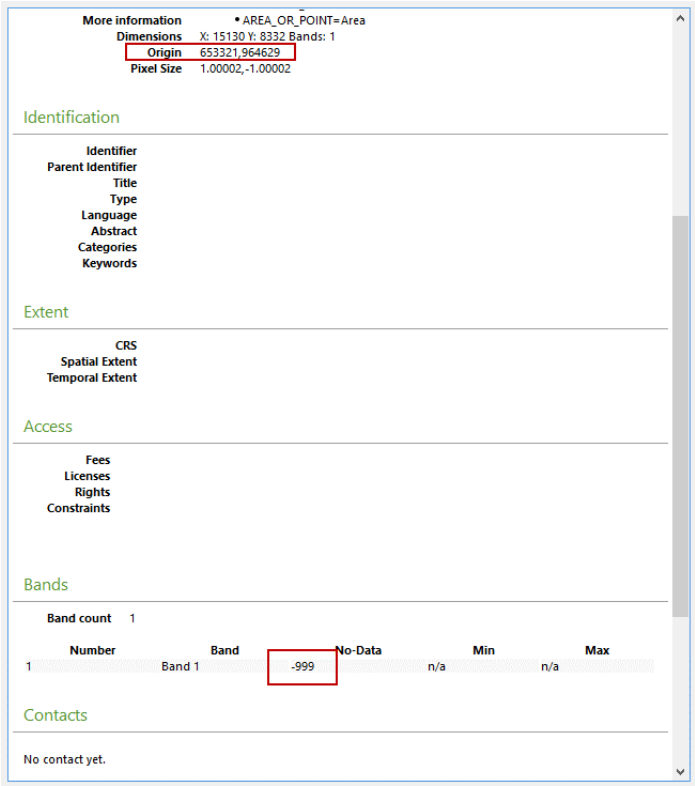
The NODATA value in the previous dialog box is applied to the interp.tif raster.



1. If the project or the data is on a server, this function may fail. Ensure all data is on the workstation.
2. If the raster has the wrong CRS or does not have complete coverage, the following message will be displayed. Save the raster with the correct CRS and make sure it has complete coverage.



3. If the "Sample Done" box is displayed but the grid table still has -9999 as the elevation, the raster CRS is not being read correctly. Save the raster as a new layer and assign the correct CRS.
4. If the grid layer elevation data does not seem accurate, review the origin vs layer extent. If the layer extent is offset from the origin, the raster realign will be inaccurate. Re-create the raster so that the origin and extent are not offset.

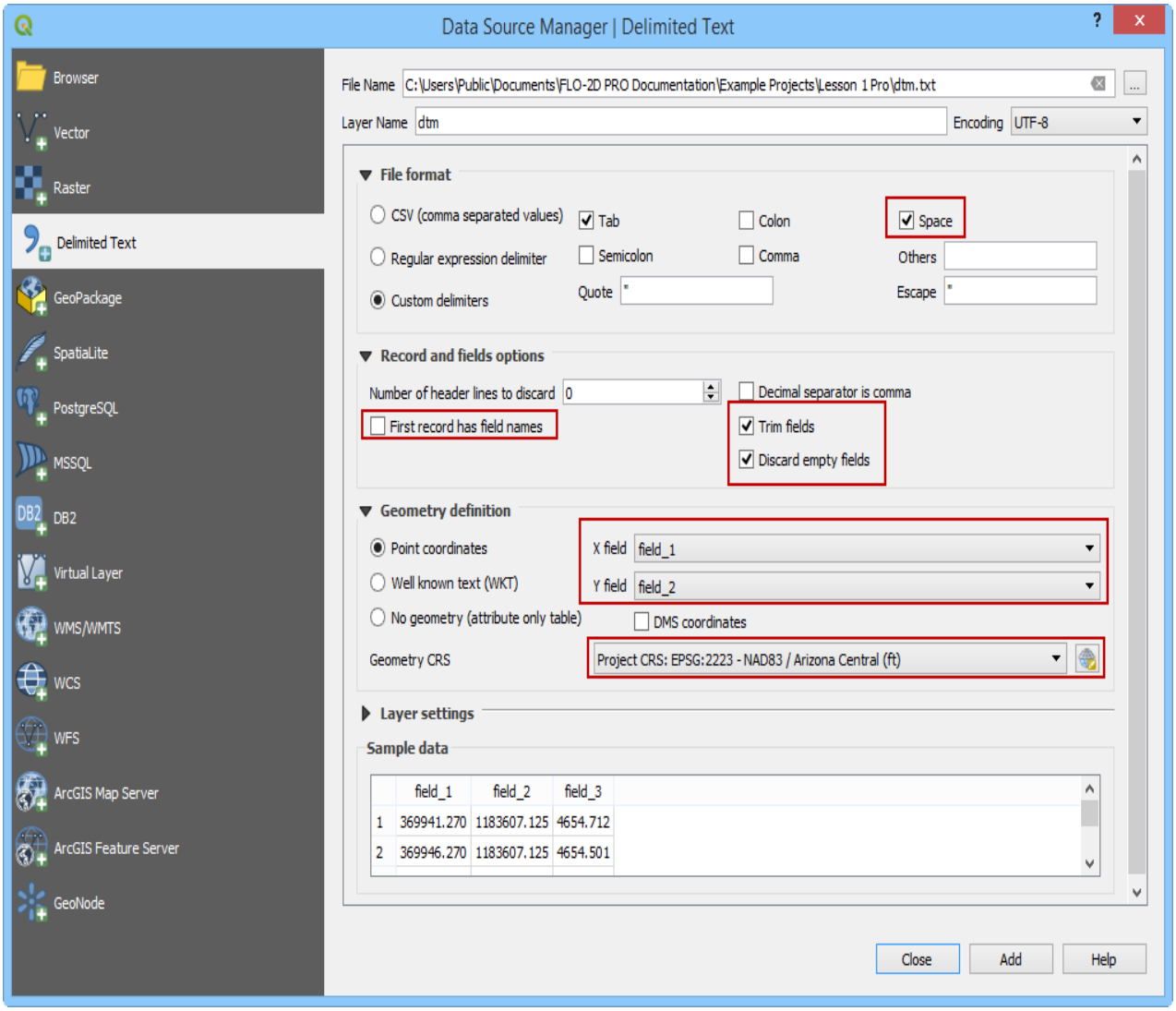


Sample Elevation Data from Points

Load Data

Click on *Layer>Add Layers >Add Delimited Text Layer* or click on the *Open Data Source Manager* icon and navigate to the *Delimited Text* tab. Add the delimited text data, following the figure below.

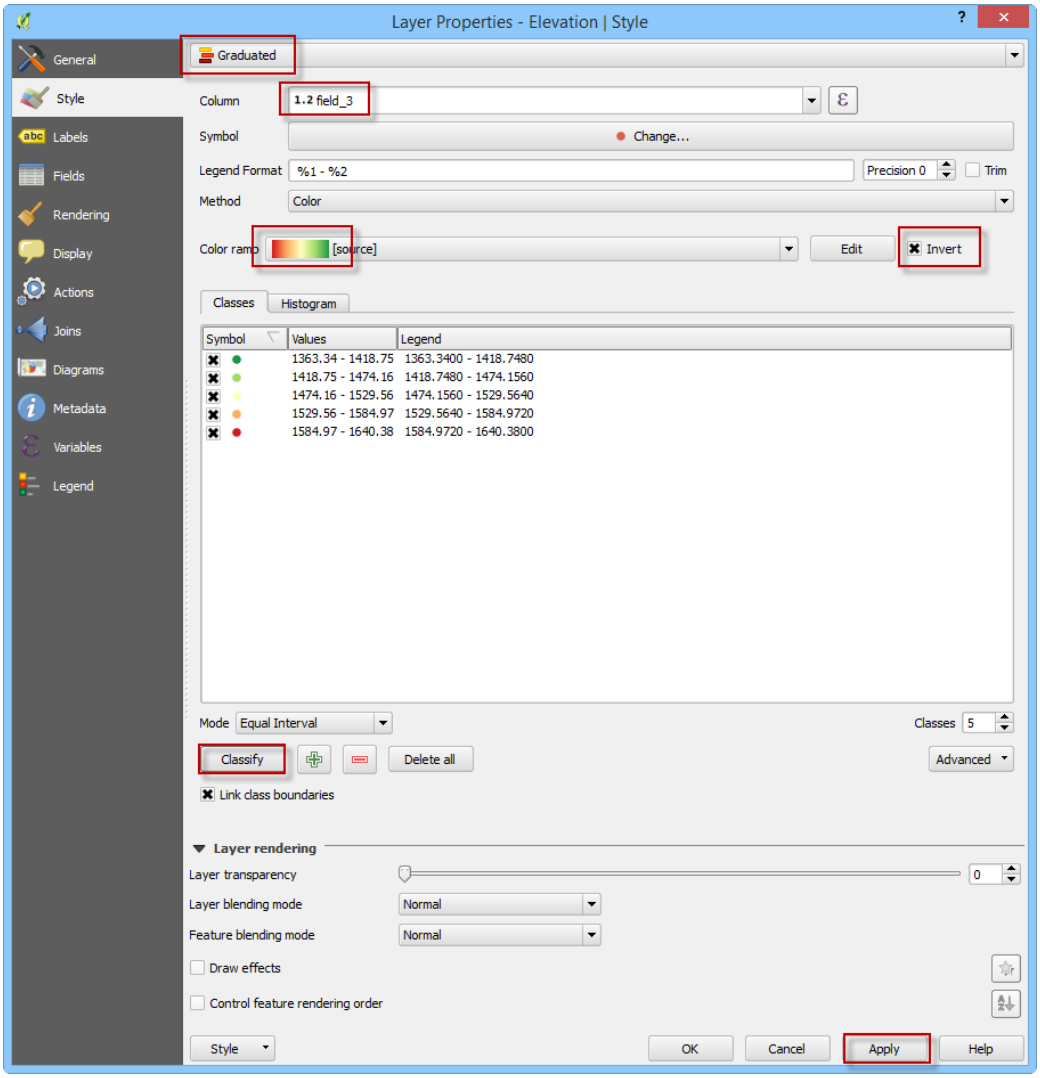
There are many options to help sort the data. Select the options that reflect the desired dataset and click *OK*.



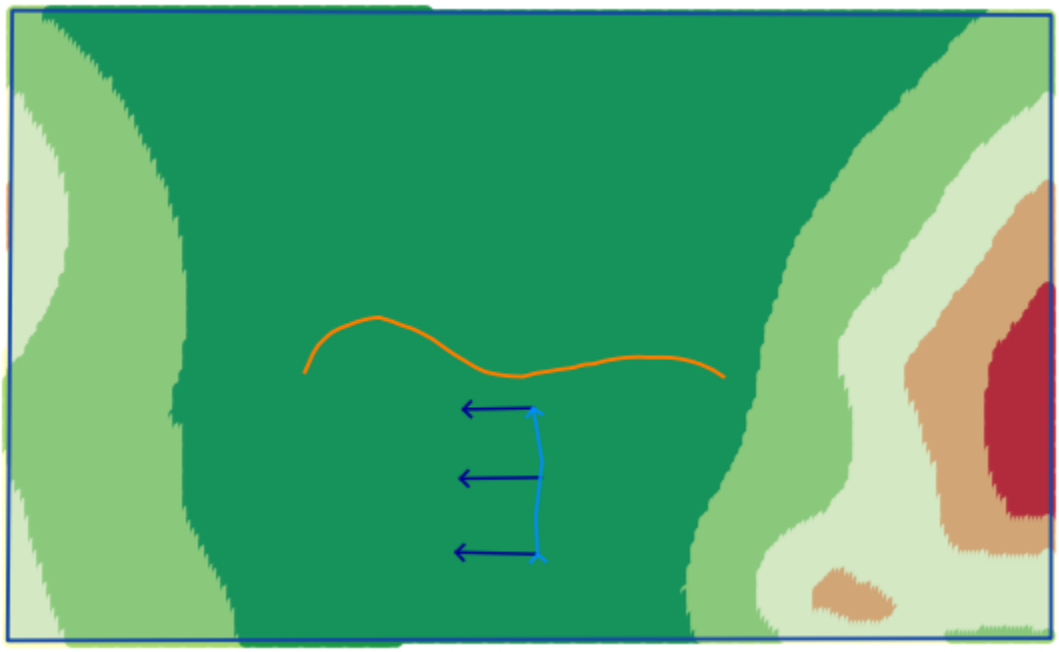
Adjust the Style properties of the elevation data to assist the quality control measures for reviewing the data. For example, elevation data that has a large range can wash out the detail in local areas of the project area.

Double click the polygon layer to open the *Properties* window and select the style tab to perform the following.

- Assign graduated colors;
- Select field to represent colors;
- Select color ramp;
- Classify the data (classifying the data adjustments will assist in locating erroneous data).



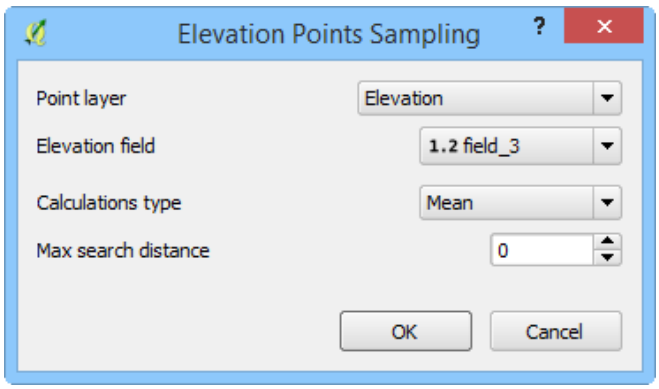
The point data style is a graduated color scheme set to the elevation scale (see figure below).



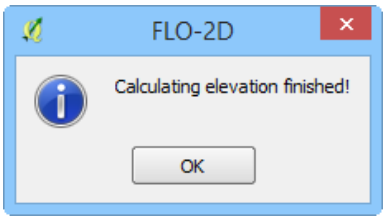
Sample Data

Click the *Assign Elevation to Grid from Points Layer* icon to interpolate the elevation data to the grid.

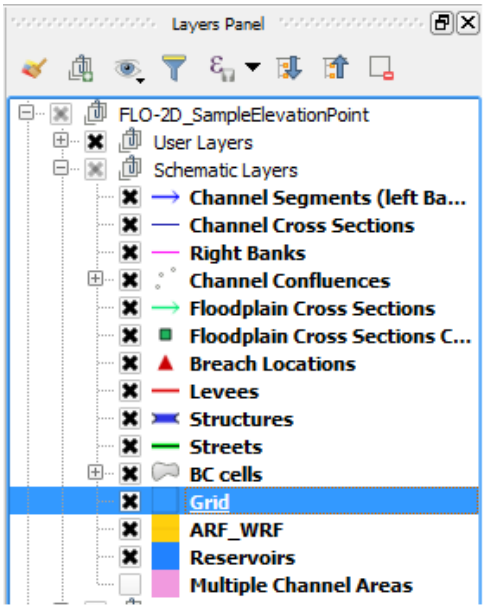
The sampling dialog box appears to select the point source, elevation field, and calculation type. Choose a max search distance to extend the search for empty grid elements. The distance is in the native map units. This field can be assigned a zero value to default to the minimum search distance.

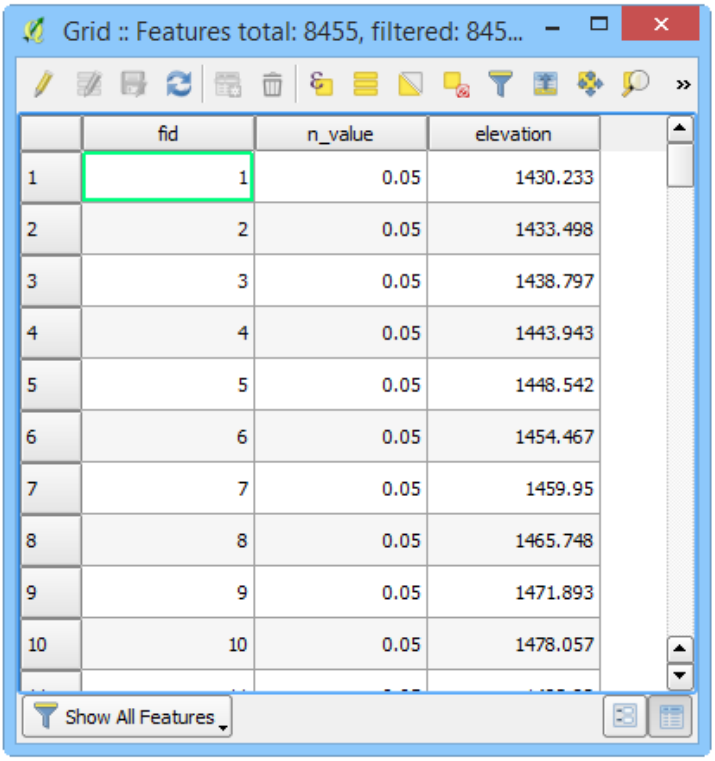


Once the calculation is complete, the following dialog is displayed. Click *OK* to continue.



The elevation data is saved to the *Grid* Layer in the *Schematic Layers* group.





	fid	n_value	elevation
1	1	0.05	1430.233
2	2	0.05	1433.498
3	3	0.05	1438.797
4	4	0.05	1443.943
5	5	0.05	1448.542
6	6	0.05	1454.467
7	7	0.05	1459.95
8	8	0.05	1465.748
9	9	0.05	1471.893
10	10	0.05	1478.057

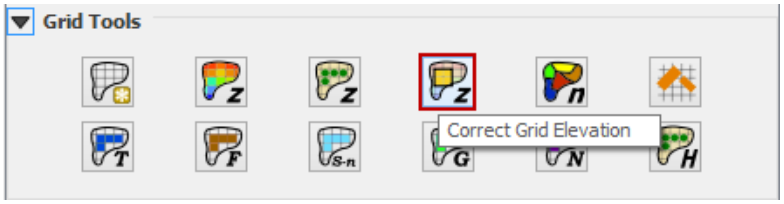
Troubleshooting

1. If the elevation data is not visible, check the CRS. It may be necessary to transform the data into the correct CRS.
2. If the elevation layer does not show up in the *Sample Elevation Dialog* box, make sure it is a point layer and that it is checked on in the *Layers* List.
3. If a Python error appears during the sampling, it may indicate that there is no attribute table. Save and reload the project into QGIS and try again.

Elevation Correction from Polygons, Polylines and Points

Overview

This tool is used to correct elevation data for polygons, points or polylines.



The correction datasets are set up in the *User Layers* group.

- Elevation Polygons
- Elevation Points
- Elevation Polylines
- Rasters

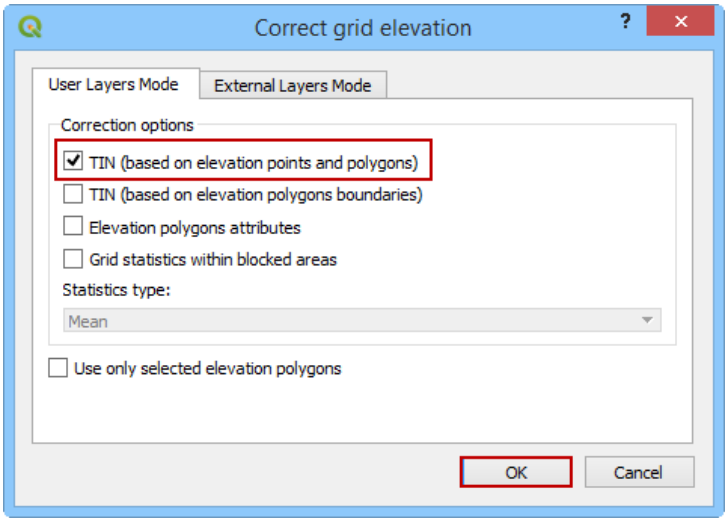
Sample Data

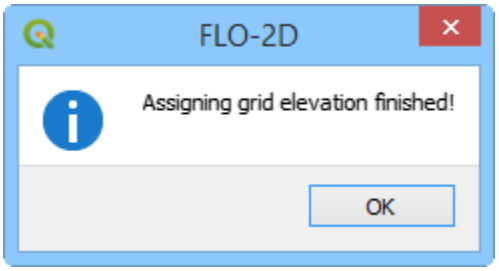
Users Layers Mode

There are multiple options in this tool. The following discussion will review each option in the *Users Layers Mode* tab.

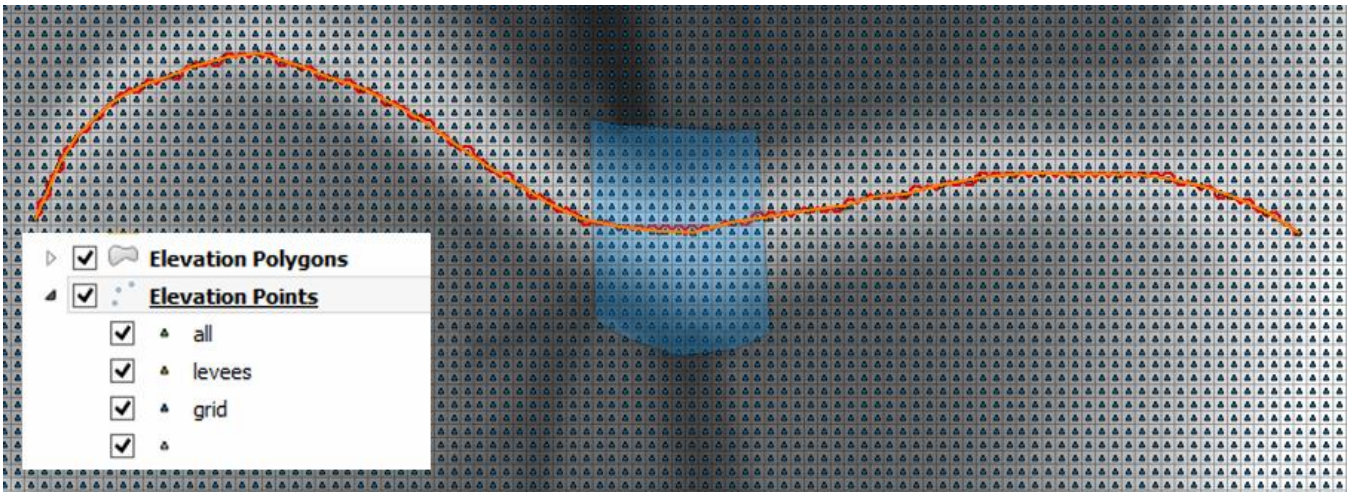
Tin from Points and Polygon

The first option is to edit the elevation on the grid using elevation points that are contained within a polygon boundary.



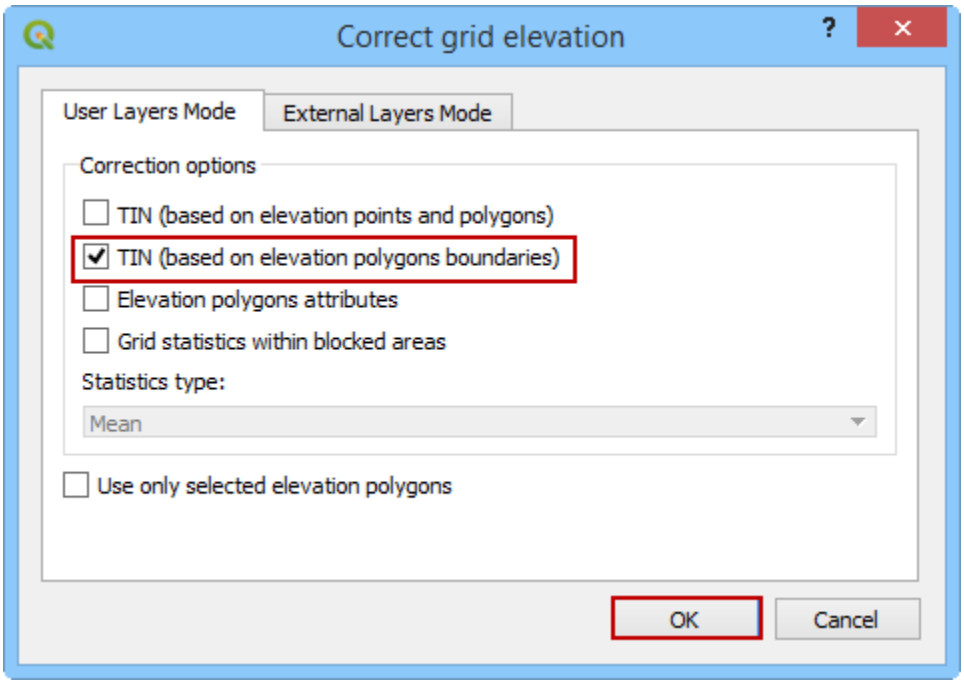


The tool creates a TIN that is confined to the elevation polygon layer. The TIN elevation is read from the *Elevation Points* layer. The elevation is assigned to the grid from the TIN as a correction to the grid elevation.

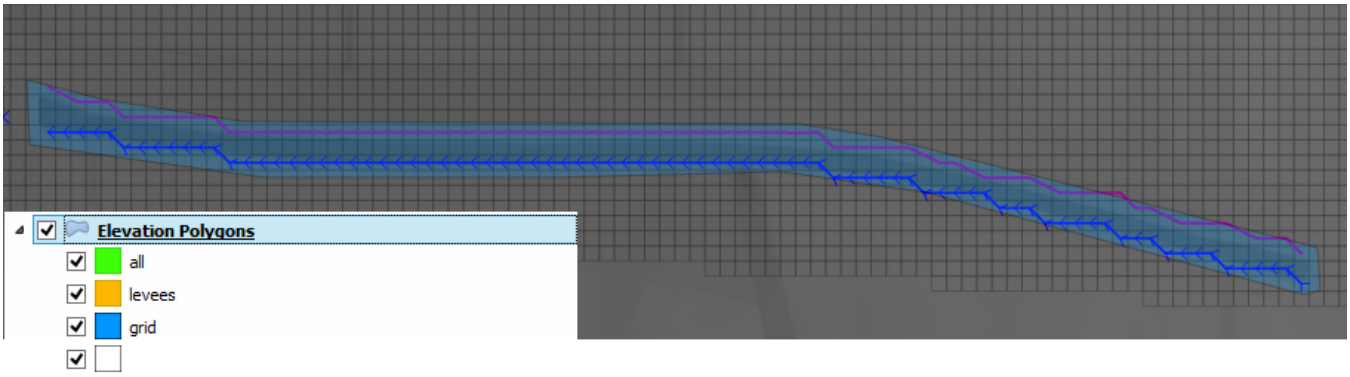


Tin from Points and Polygon

The second option is to build a TIN from the grid element elevation surrounding a polygon. This will assess the elevation where the polygon intersects the grid and interpolate that elevation to the TIN.

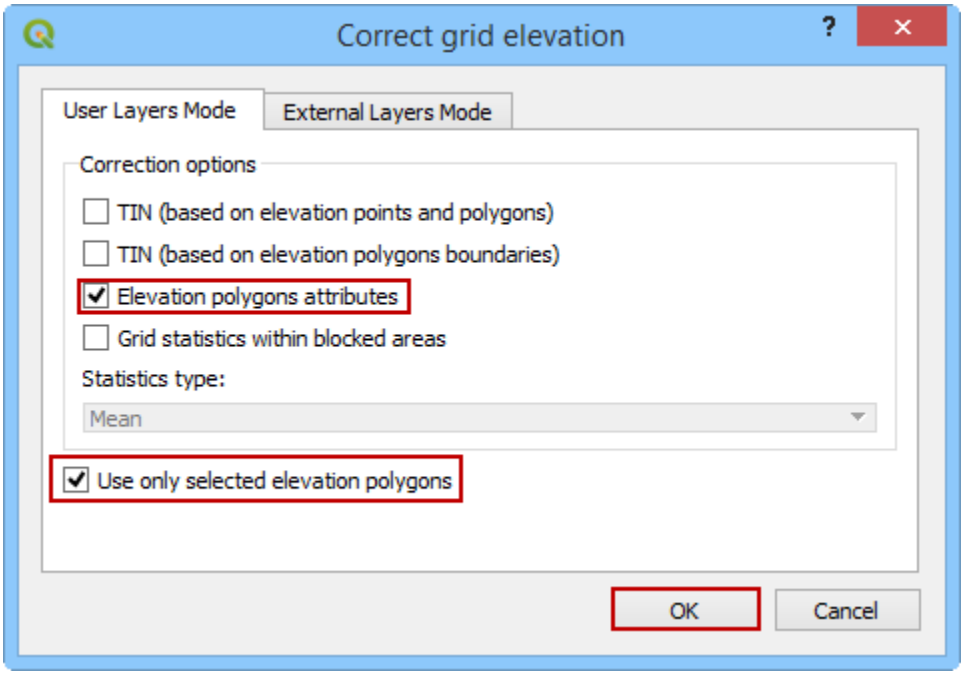


This option is used when a Cut or Fill correction is required. For example, to fill a channel with elevation data along the bank, cover the channel with an elevation polygon and apply the correction.

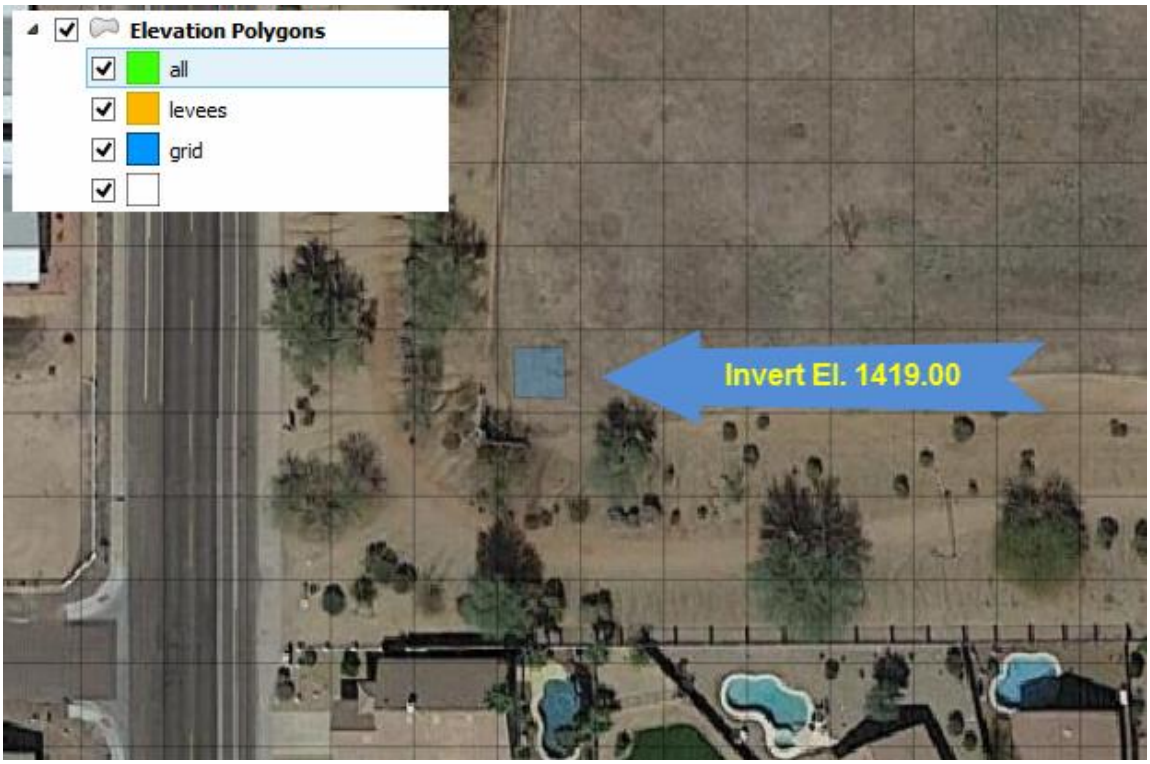


Elevation Polygon Attributes

This option is used when a single known elevation correction is required.

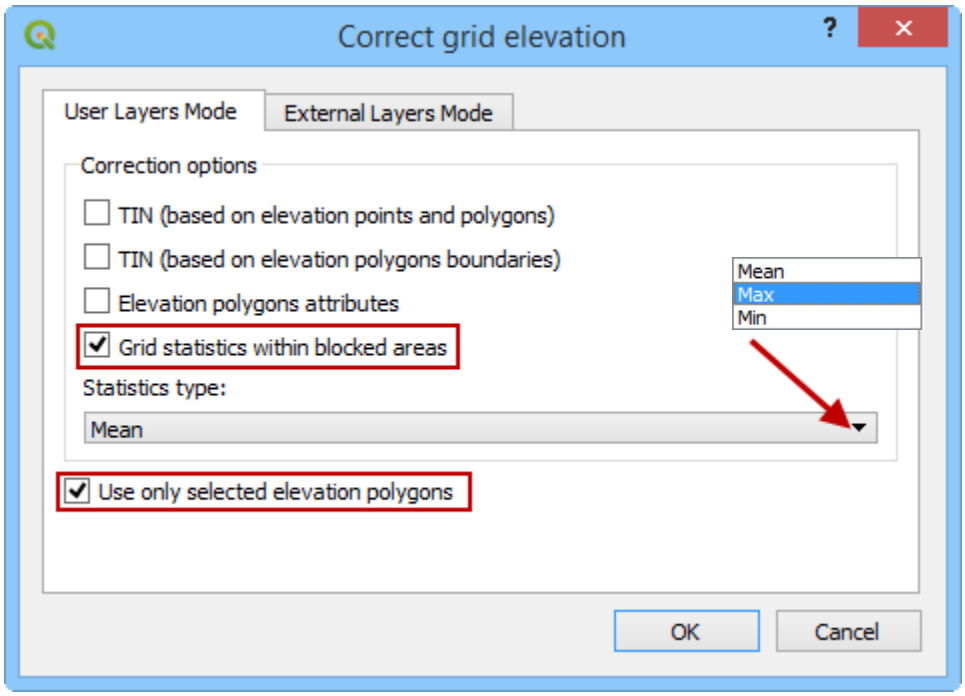


In this example, the invert elevation at the headwall inlet is incorrect. The polygon has a new elevation assigned that will be applied directly to the grid layer. In this case the correction is applied to the selected polygon only.



Grid Statistics within Blocked Areas

In this case, the correction is applied by analyzing the statistics of the elevations within the polygon. The mean, max or min elevation of the combined cells within the polygon are applied as a general condition to all of the cells within the polygon.



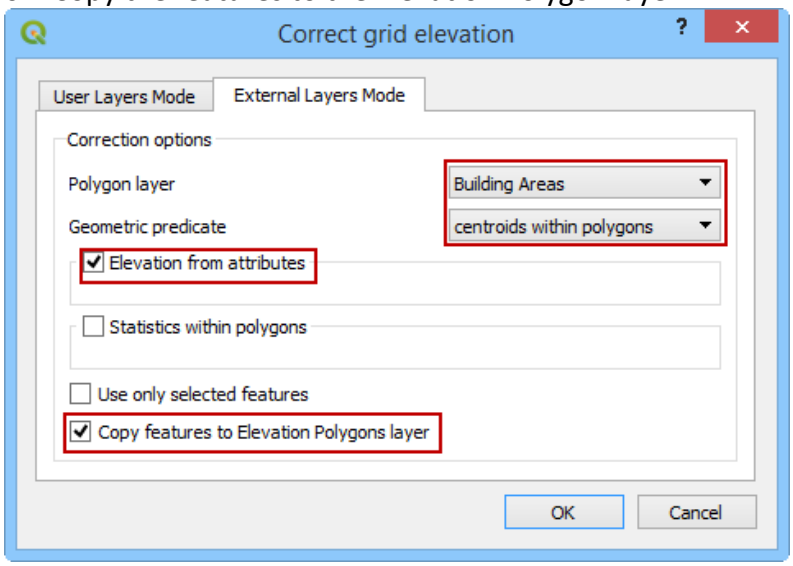
External Layer Mode

This section will review each option in the *External Layers Mode* tab:

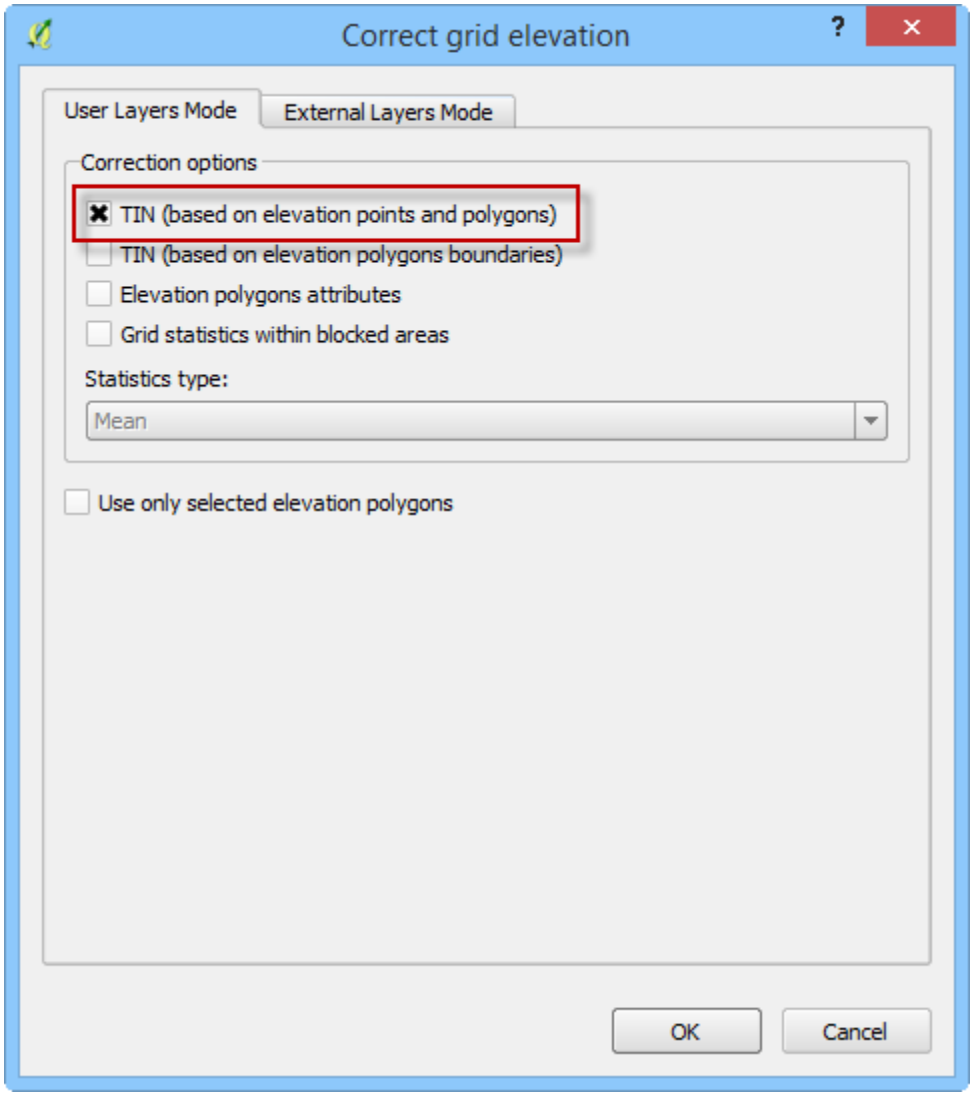
Correction Options

There are several grid element correction options available in this tool.

1. Select any polygon layer.
2. Define the geometric predicate. Grid centroid or grid element.
3. Take the elevation from an attribute table.
4. Take the mean statistics from the elements within the polygons.
5. Use only selected features or all features.
6. Copy the features to the Elevation Polygon layer.

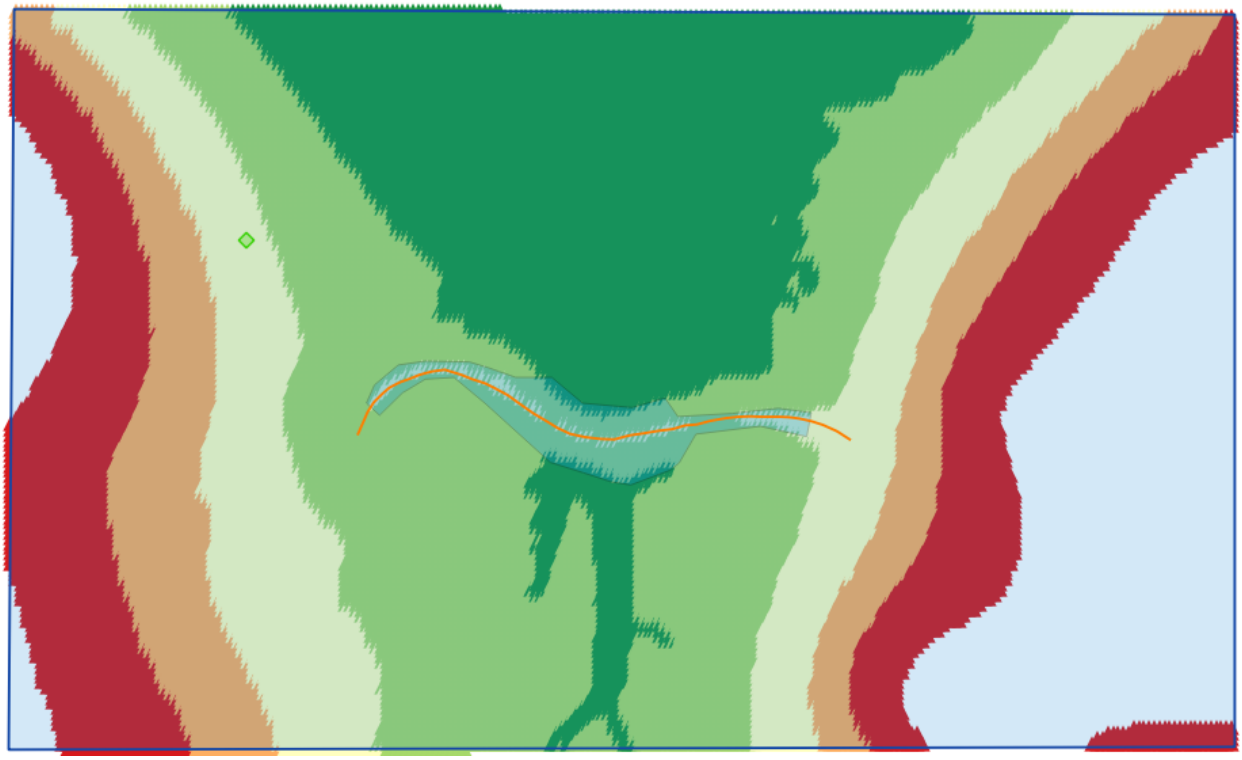


Levee from Elevation Polygon

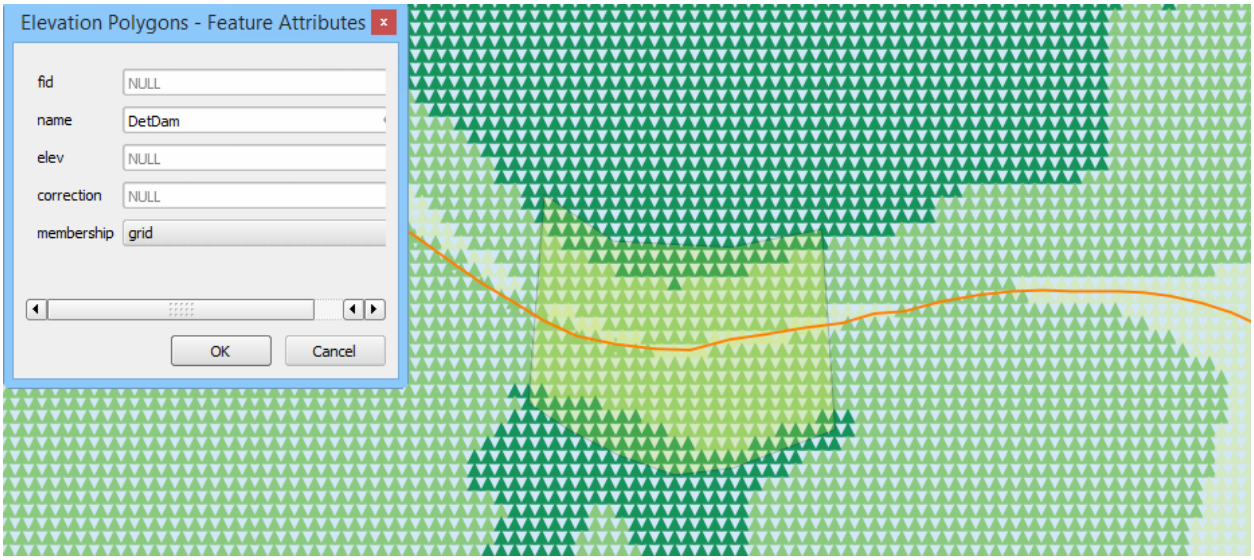


This option functions as a cut and fill tool. It will remove features surrounded by a polygon and replace that elevation by interpolating elevation points within the polygon. This following example will show a cut method.

First it is necessary to load or digitize the data that will be used to adjust the elevation data. In this project there is a detention dam. The dam topography is shown by the orange line (figure below). If dam breach is being simulated, it is necessary to take the dam topography out of the digital elevation model so that levee will fail from the crest elevation to the floodplain surface using the FLO-2D levee component.



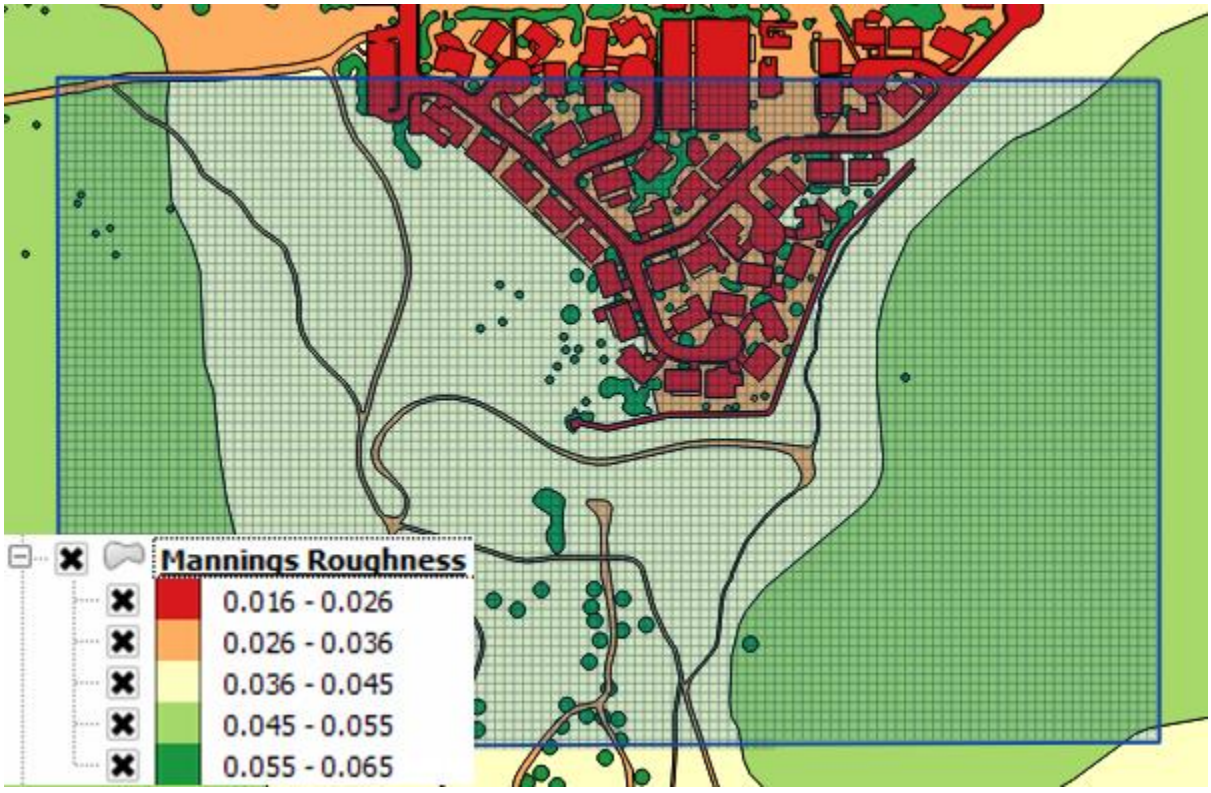
Select the *User Layers>Elevation Polygon* layer and click edit. A Polygon is digitized to across the dam as shown below. Click *OK* to fill the attribute of the polygon and close it. Point data can also be loaded into the project that represents pre-design or pre-construction conditions.



Sample Roughness Data

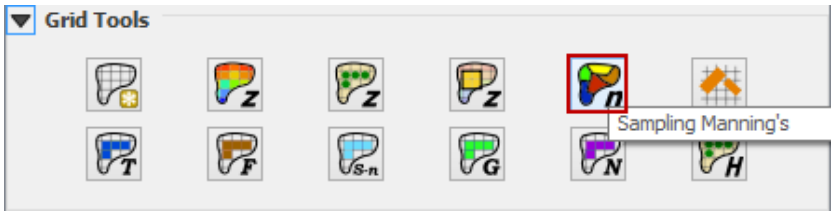
Overview

In this task, the spatially variable manning's roughness is calculated from a polygon shapefile. The polygons represent roughness associated with different LandUse categories such as building, street, grass, desert brush and many others. The Plugin will sample the roughness polygon from the centroid of the grid layer to assign the individual grid element roughness. This is not an interpolation average calculation. It is a point sample assignment.

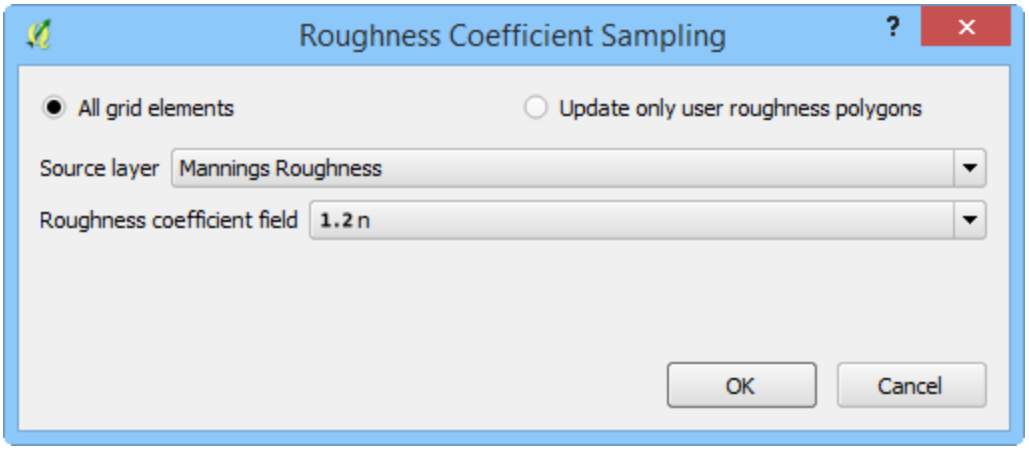


Sample Data

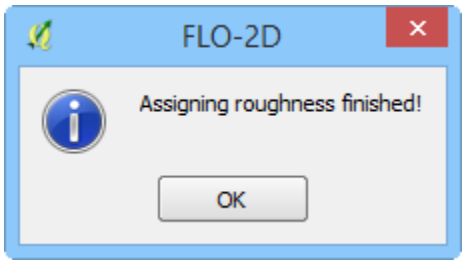
Click the *Sample Manning's icon* from polygon layers.



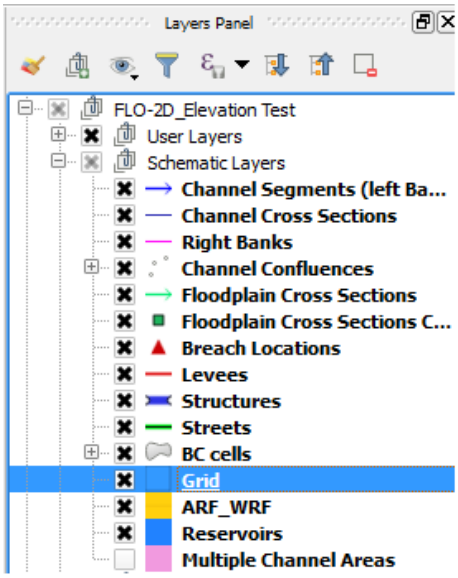
This layer requires a polygon shapefile with roughness data or digitized data assigned to the *Roughness User Layer*.

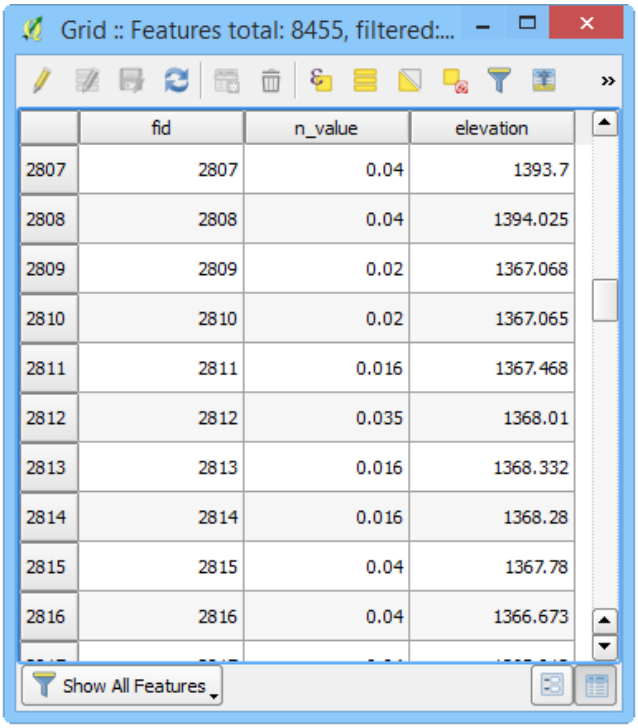


Once the sample is complete, the following window will appear. Click *OK* to close the window.



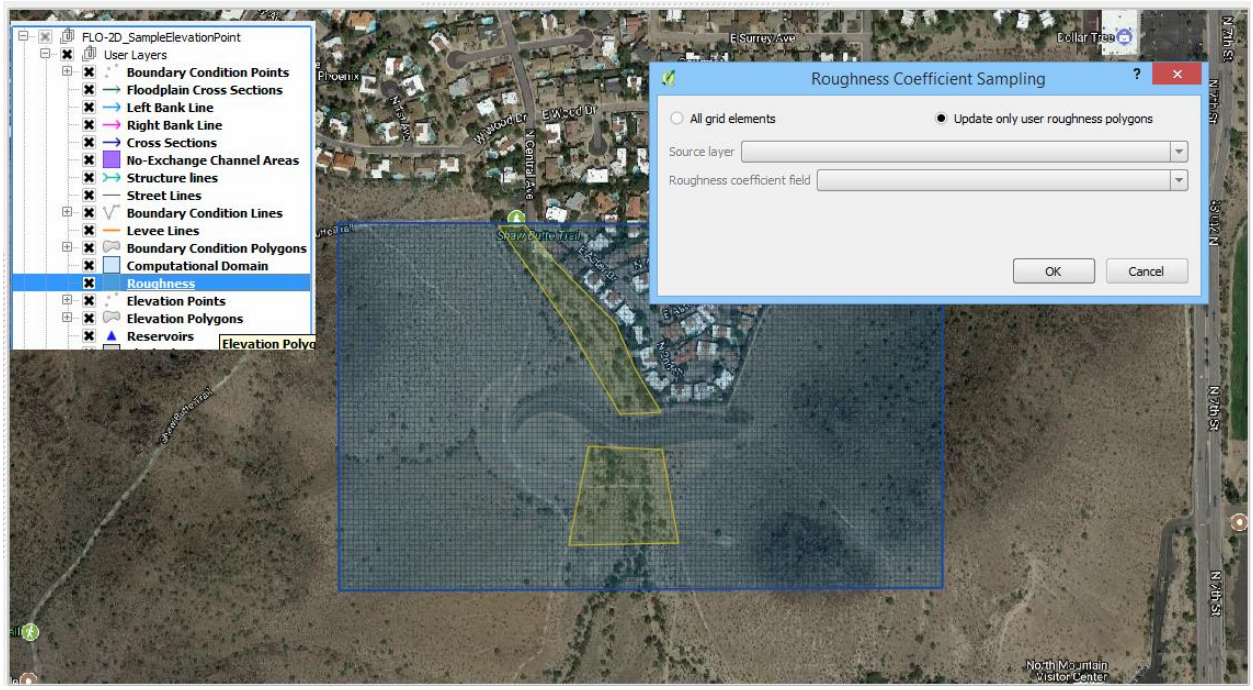
The roughness values are assigned to the *Grid* layer in *the Schematized Layers* group.





	fid	n_value	elevation
2807	2807	0.04	1393.7
2808	2808	0.04	1394.025
2809	2809	0.02	1367.068
2810	2810	0.02	1367.065
2811	2811	0.016	1367.468
2812	2812	0.035	1368.01
2813	2813	0.016	1368.332
2814	2814	0.016	1368.28
2815	2815	0.04	1367.78
2816	2816	0.04	1366.673

If it is necessary to update or change a small selection of elements, use the *Roughness* layer in the *User Layers* group.



Troubleshooting

1. The Roughness layer must be a polygon layer. It is usually a shapefile.
2. The layer CRS must match the project CRS.
3. If a Python error appears during the sampling process, it may indicate that attribute table is missing. Save and reload the project into QGIS and try again.

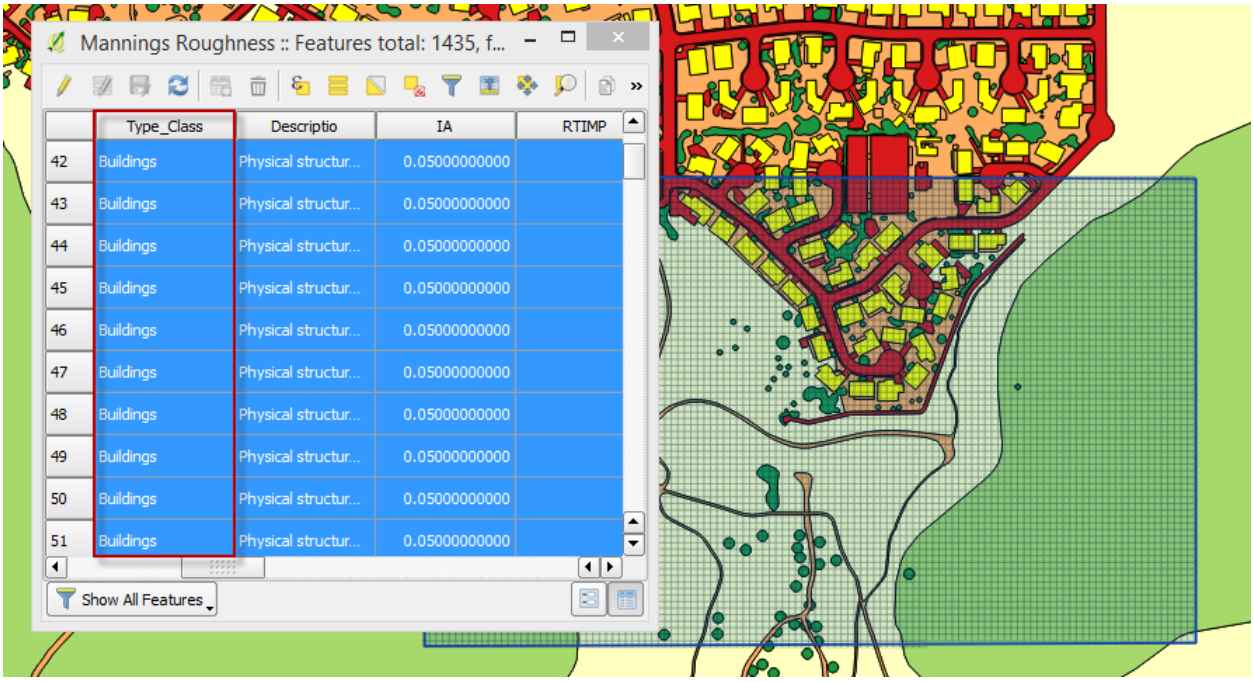
Area and Width Reduction Factor – Buildings

Overview

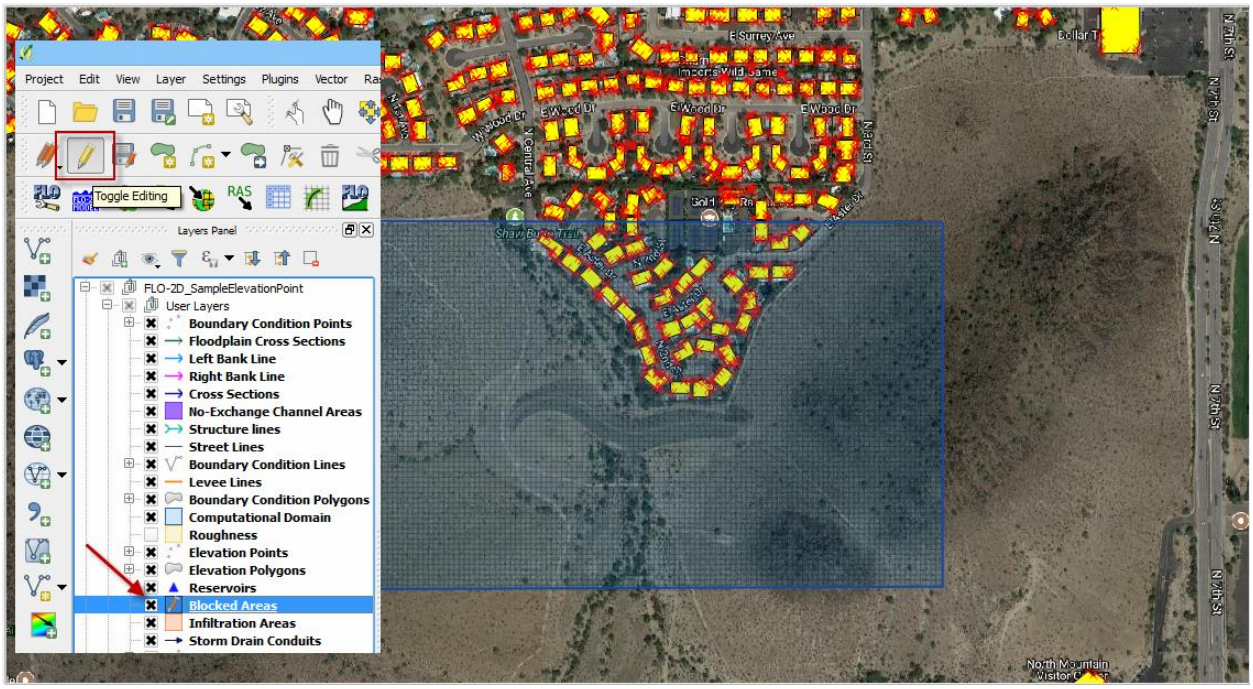
Buildings are only calculated from the *Blocked Areas* layer in the *User Layers* group. The blocked areas are polygons that represent buildings or other features that displace and redirect the flow as it moves over an area. The blocked areas are converted to FLO-2D Area Reductions Factors (ARF) and Width Reduction Factors (WRF) in the Schematic Layer.

Digitize or Copy Data

Use the editor and *Create Polygon* tool to digitize or outline buildings with polygons. If the buildings are in another layer, copy the polygons from the external layer and paste it into the *Blocked Areas* layer. Select *Buildings* from the external layer. If there are multiple types of polygons in the layer, it is possible to select the building by attribute type. Copy the selected polygons using Ctrl 'C'.



Highlight the *Blocked Areas* layer and click the *Toggle Editing* icon. Paste the selected polygons into the *Blocked Areas* layer.



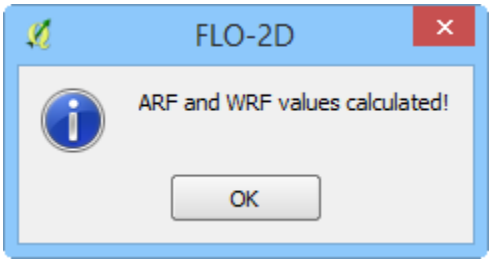
Click the *Toggle Editing* icon again to save and close the editor. The blocked layers attribute table can be edited to add:

- Building Collapse;
- ARF additional reduction coefficient;
- WRF additional reduction coefficient.

Use a table join or spatial join to fill the attribute data if it is available in an alternate layer. Commands and tutorials for these tools are available on the QGIS Tutorials website: [www.qgistutorials.com.](http://www.qgistutorials.com)

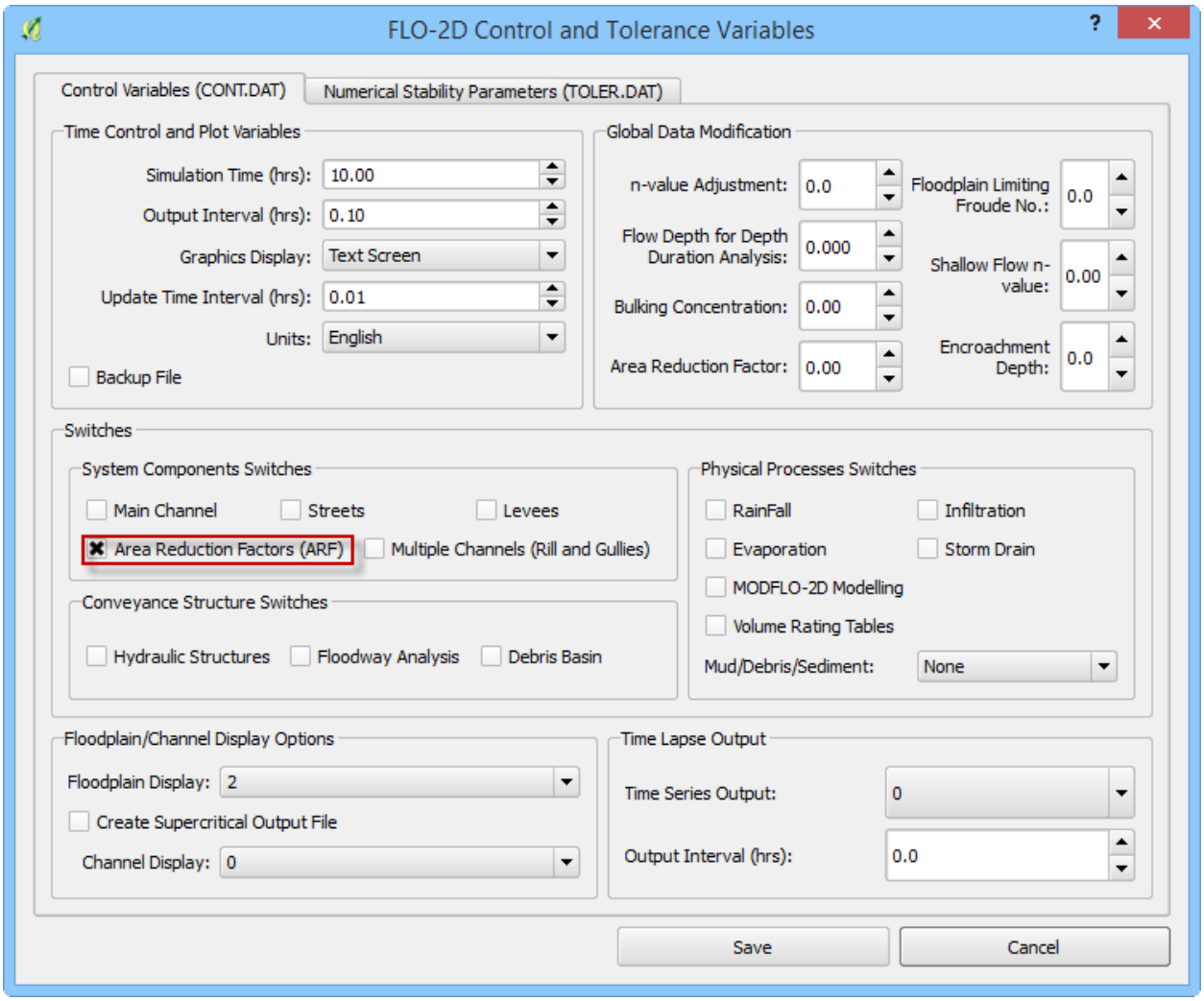
Calculate ARF and WRF layers

Click the *Evaluate Reduction Factors (ARF and WRF)* icon and wait for the procedure to finish. The following message will appear and click *OK* to close it.



The ARF and WRF features are visible in the *Schematic Layers* group.

Click on the *Set Control Parameters* icon, and then on the *Control Variables (CONT.DAT)* tab Check on *Area Reduction Factors (ARF)* and then click *Save*.



Troubleshooting

1. Missing building polygons from the *Blocked Areas* layer can be created.
2. If the *Grid* layer is empty, create a grid and try again.
3. If a Python error appears during the sampling procedure, the attribute table may be missing. Save and reload the project and try again.

Spatially Variable TOL Parameter – Low Impact Development (LID)

Overview

The spatially variable TOL parameter is specified depth below which no discharge is shared between grid elements. This FLO-2D parameter represents grid element depression storage and can be used to simulate Low Impact Development (LID) storage such as the volume in cisterns or infiltrated volume through permeable pavers. In the following example, the neighborhood collects rainwater using built-in cisterns that are attached to the gutters of the buildings. The cisterns have a fixed volume and there is one per building. The volume of each cistern is 50 gallons or 6.68 ft^3. Divide this volume by the surface area of a grid element to determine the TOL depth value assigned to each cell.

Digitize or Copy Data

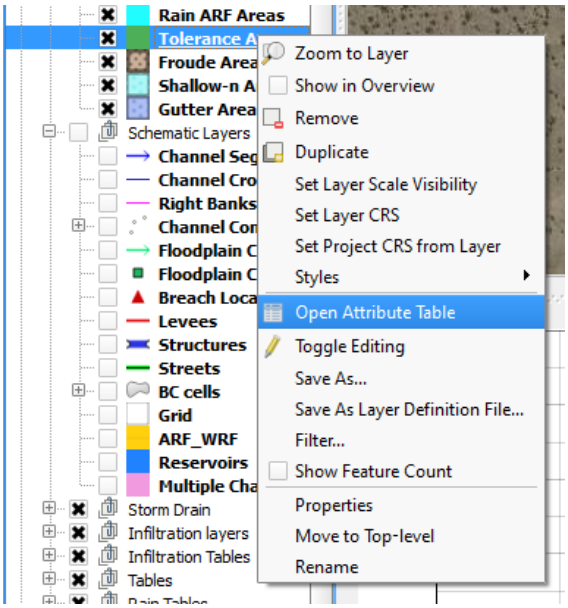
Click the *Tolerance Areas* layer and use the editor tool to digitize the outline of the building roof area. This is the area that collects water.

The roof polygons can be imported to a separate layer and pasted into the *Tolerance Areas* layer. In this instance, the polygons were copied from the *Blocked Areas* layer.

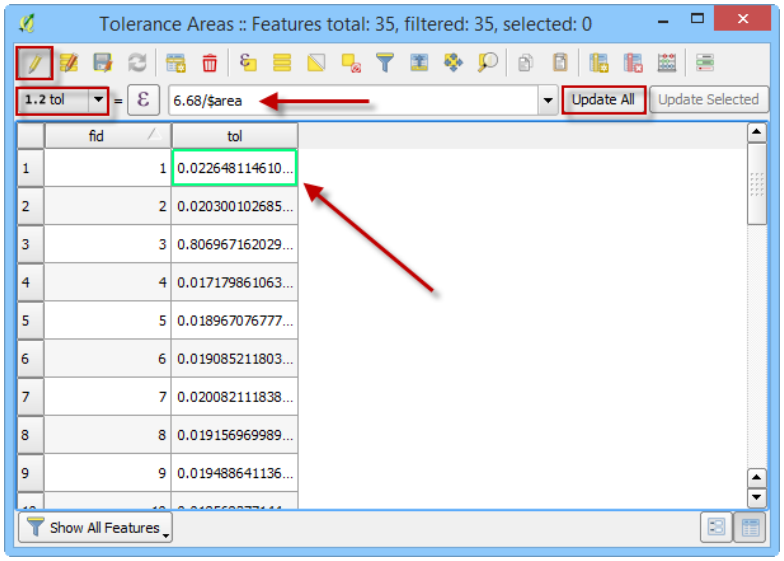
In this example the LID TOL value is unknown and must be calculated from the cistern volume and the collection area. Each house has a theoretical 50-gallon rain collection cistern. The volume is converted to 6.68 cubic feet. The tolerance value is a depth in feet, so it can be applied to the roof area of each house. For the sake of simplicity, the assumption is that the roof area and the house polygon area are the same.

Define the Tolerance Attribute

Open the attribute table of the *Tolerance Areas* layer.

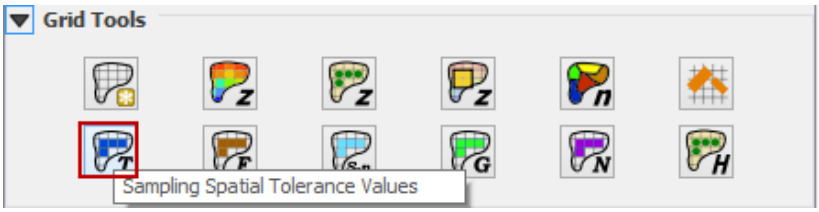


On the attribute table window, click on the *Toggle Editing* icon, change the field to TOL, add the command to set the TOL depth (6.68 / $area) and click *Update All*. Save and close the editor tool and close the attribute table. This process converted the volume of the cistern to a depth over the area of each polygon.

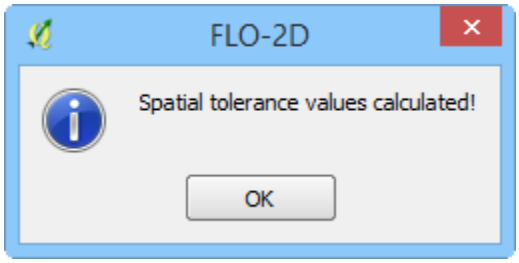


Sample Data

Click the *Sample Spatial Tolerance Values* icon.



Once the process is complete, Click *OK* to complete TOL the assignment.



This process has converted the volume from the polygon to the grid elements that intersect the polygon.

Troubleshooting

1. Create the tolerance polygons if they are missing from the *Tolerance Areas* layer.
2. If the *Grid* layer is empty, create a grid and try again.
3. If a Python error appears during the sampling process, the attribute table may be missing. Save and reload the project into QGIS and try again.
4. If the Attribute Fields: collapse, calc_arf, and calc_wrf are not filled by the user, the calculation cannot be performed.

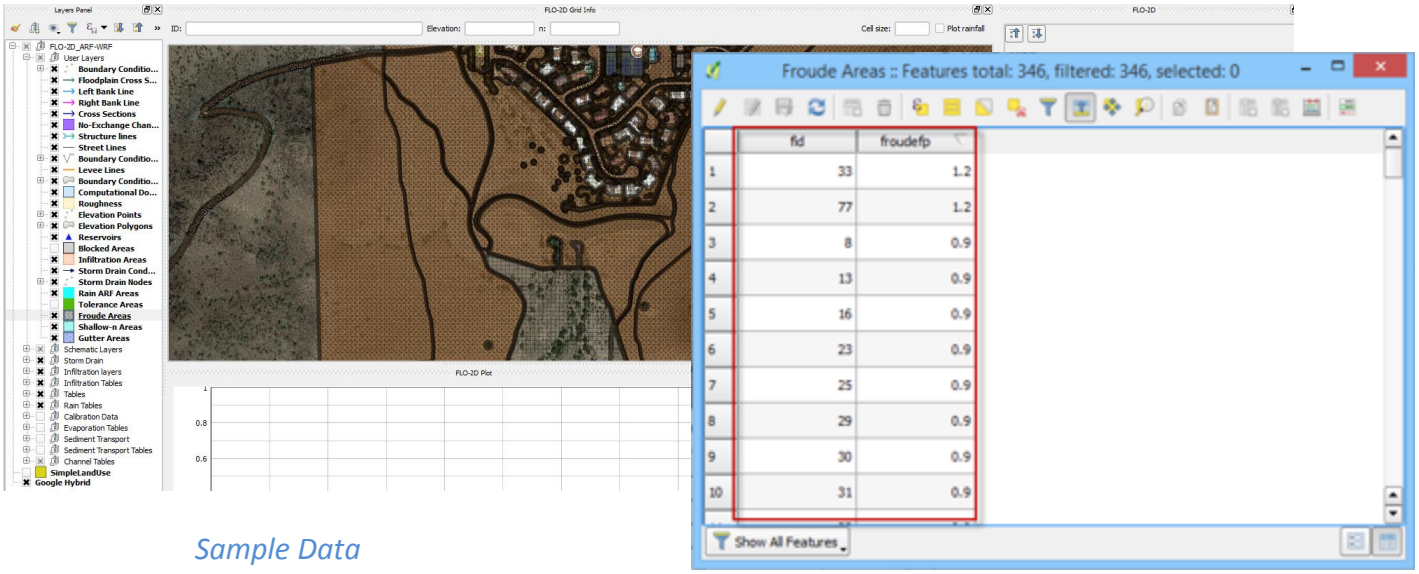
Spatial Limiting Froude Number

Overview

The Spatial Limiting Froude layer is used to set a limiting Froude number for individual FLO-2D grid elements. For example, overland flow on a gentle slope will become subcritical so a limiting Froude value of 0.9 can be assigned. If the limiting Froude is exceeded, the cell roughness n-value is incrementally increased and maintains numerical stability that might occur for severe supercritical flow. For further discussion on the limiting Froude numbers refer to the FLO-2D Data Input Manual.

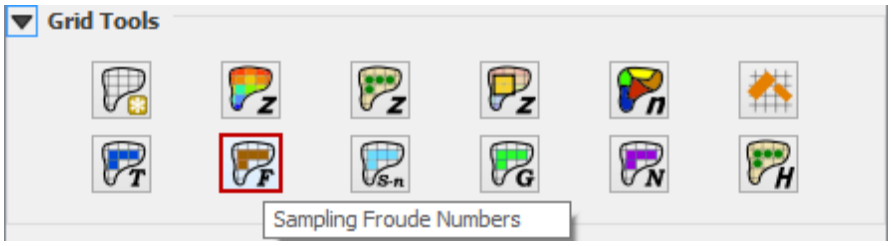
Digitize or Copy Data

Select the *Froude Areas* layer and click *Toggle Editing*. Create or copy the polygons that represent the Froude areas to the layer and save them.

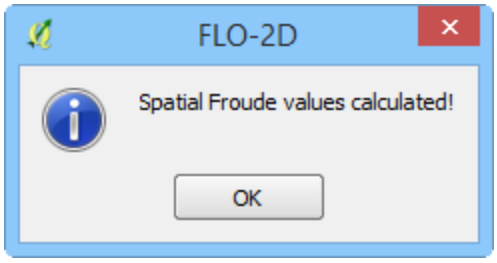


Sample Data

Click the *Sampling Froude Numbers* icon.



Once the process is complete *OK* to close it.



Troubleshooting

1. Create the limiting Froude polygons if they are missing from the *Froude Areas* layer.
2. If the *Grid* layer is empty, create a grid system and try again.
3. If a Python appears during the sampling process, the attribute table may be missing. Save and reload the project into QGIS and try again.

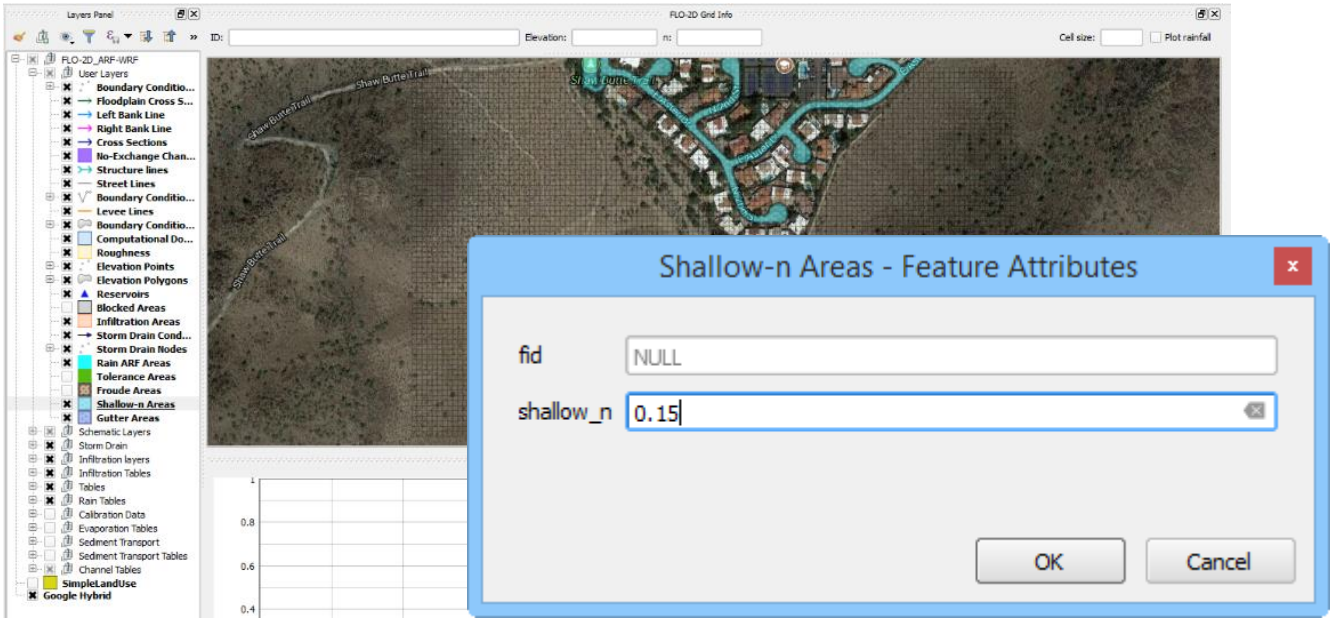
Spatial Shallow n

Overview

A spatially variable shallow n-value enables the flow roughness to be depth integrated enable high n-values at shallow flow depths. The value can also be globally assigned in the *FLO-2D Control and Tolerance Variables* window. Polygons can be applied to select individual or groups of grid elements to supersede the global n-value as in the case for streets and parking lots.

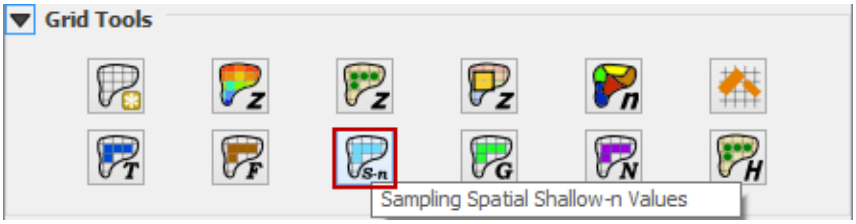
Digitize or Copy Data

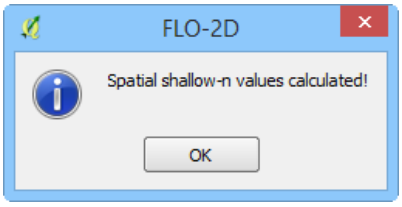
Select the *Shallow-n Areas* layer and click edit. Create or copy the polygons that represent the shallow roughness areas to the layer and save them.



Sample Data

Click the *Sampling Spatial Shallow-n Values* icon and then *OK* once the process is complete.





Troubleshooting

1. Create the shallow-n polygons if they are missing from the *Shallow-n Areas* layer.
2. If the Grid layer is empty, create a grid and try again.
3. If a Python error appears during the sampling process, the attribute table may be missing. Save and reload the project into QGIS and try again.

Gutters

Overview

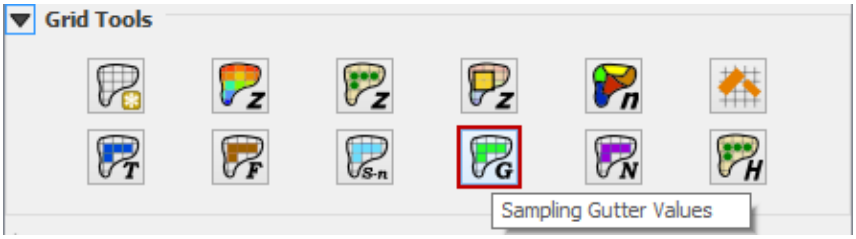
In the FLO-2D model street gutters convey shallow flow using a typical curb and gutter cross section that has a triangular shape created by the cross slope associated with the street crown. For the FLO-2D gutter routing, the triangular flow area is assumed to have a 2 percent cross slope. The gutter flow is exchanged with other upstream and downstream gutter elements, the sidewalk, and other street elements (not having a gutter). Floodplain flow is exchanged with the gutter elements through the sidewalk interface.

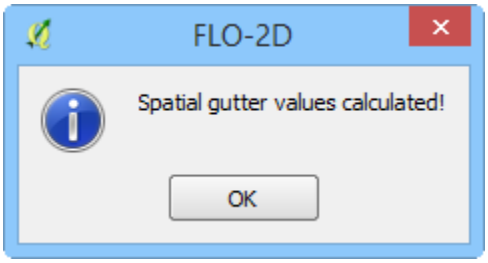
Digitize or Copy Data

Select the *Gutter Areas* layer and click *Toggle Editing*. Create polygons that represent the gutter alignment. Enter the data from the attributes table. Save the Gutter layer and close the editor.

Sample Data

Click the *Sampling Gutter Values* icon in the *Grid Tools* and click *OK* when the process is complete.





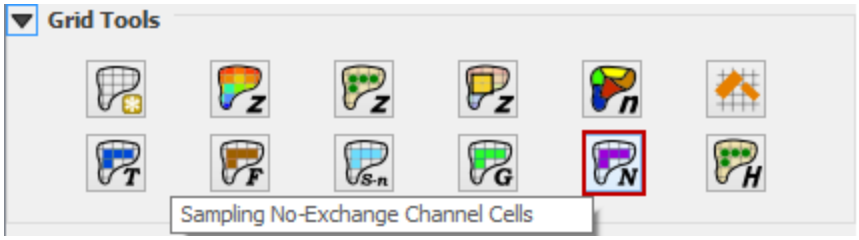
Troubleshooting

1. Create the gutter polygons if they are missing from the *Gutters Areas* layer.
2. If the *Grid* layer is empty, create a grid and try again.
3. If a Python error appears during the sampling process, the attribute table may be missing. Save and reload the project into QGIS and try again.

No Exchange Channel Cells

Overview

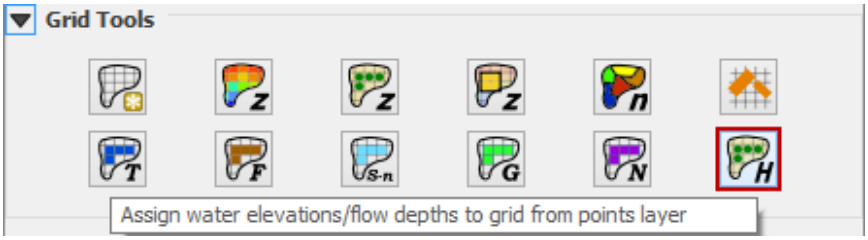
The no exchange elements can be set up for any channel left bank element. This will prevent the channel at that location from sharing discharge with the flood-plain.



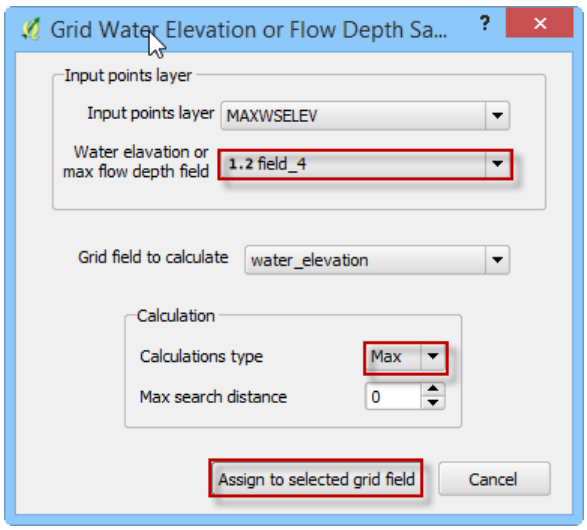
Assign Results from Points Layer

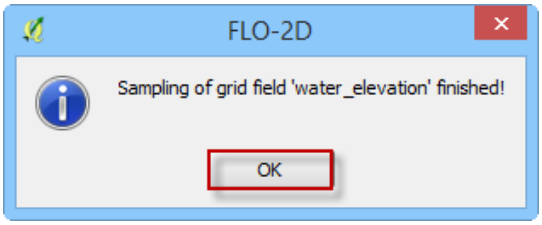
Overview

This tool is used to assign water surface and depth to the grid layer.



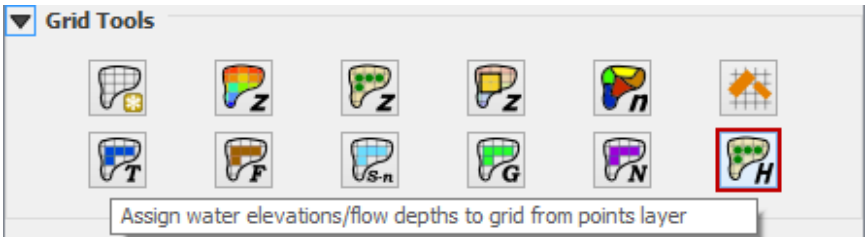
Edit the dialog box as shown below and click the *Assign to selected grid field* button.



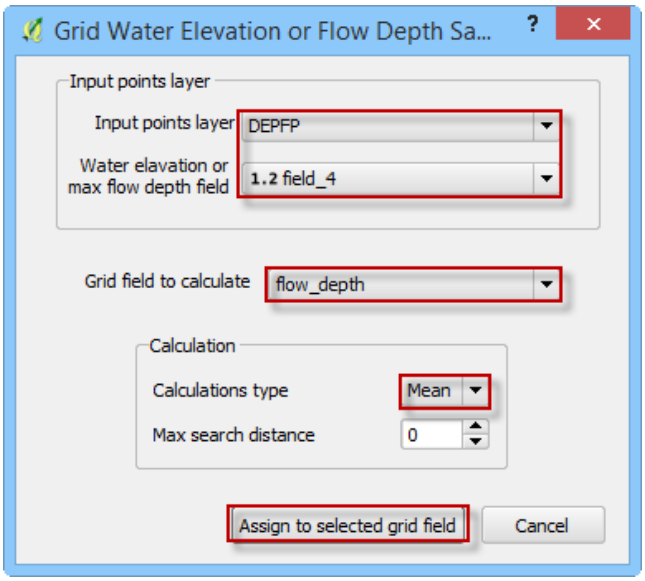


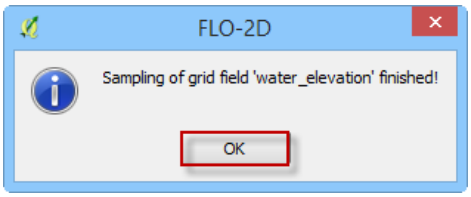
Repeat the process for the *Depth* layer.

Select the Grid Tools *Assign water elevations/flow depths to grid from points layer* icon.



Edit the dialog box as shown below and click the *Assign to selected grid field* button.





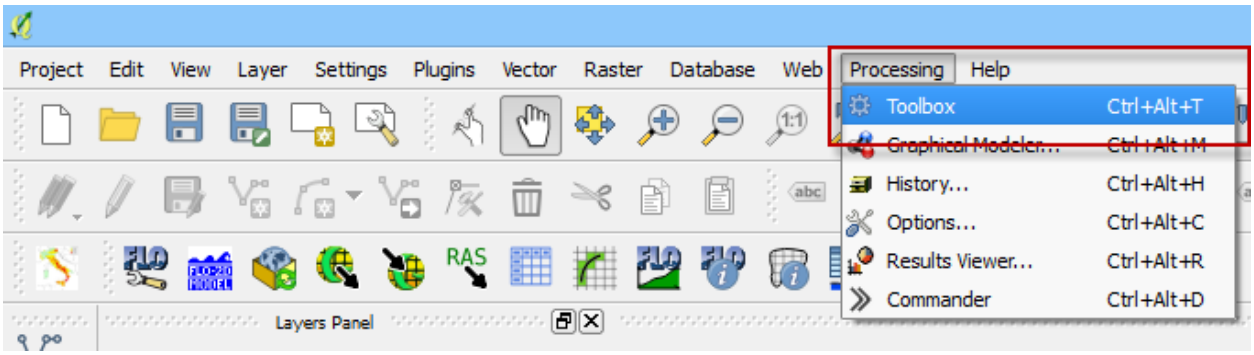
Profile Tool

The profile tool is primarily used to review the channel profile but can also be used to review table data for specific features.

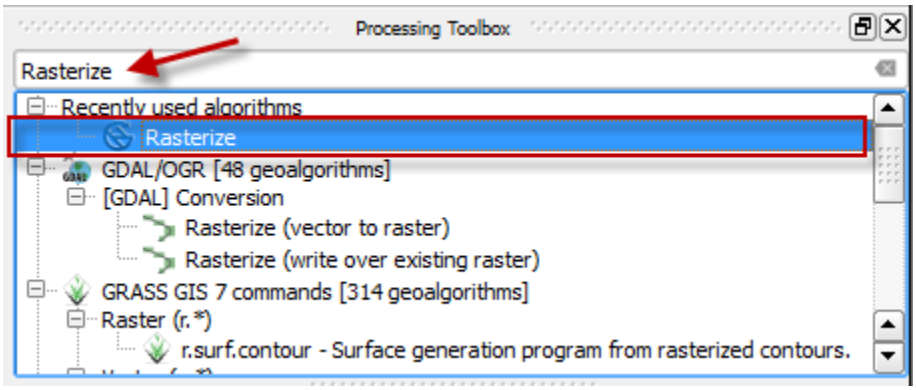
Raster Profile

Create a raster

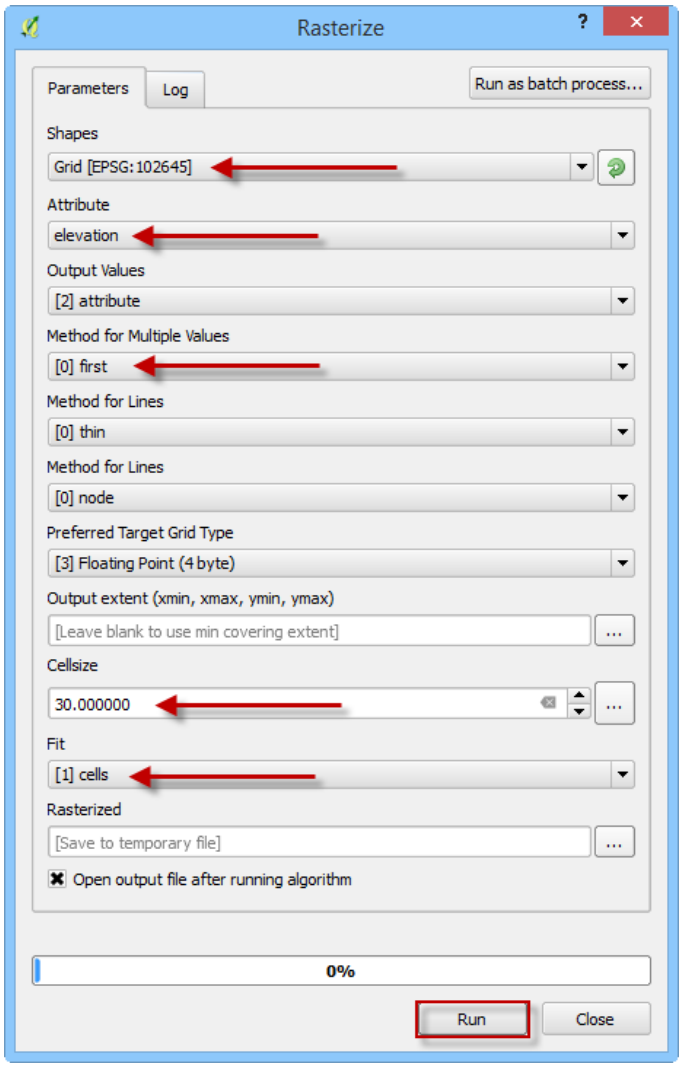
Set up the raster in the *Profile Tool* widget. In this example, the grid elevation raster is used. To create a grid elevation raster, use the Saga-Rasterize tool. On the Main QGIS Menu, click *Processing>Toolbox*.



Enter the search term *Rasterize* in the Processing Toolbox search feature. Double click the Saga Rasterized tool.



Change the dialog box as shown below and click *Run*. Cell size is the grid element size. Fit = cells will ensure that the raster pixels are centered on the grid elements. Saga-Rasterize is a good raster generator for FLO-2D data because it builds a raster with hidden null data. QGIS has many raster generators and they all work slightly differently.



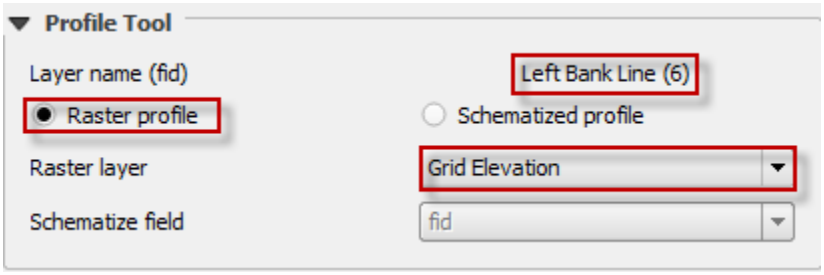
The raster that is created is a temporary layer called *Rasterize*. This layer should be saved to a permanent layer using the QGIS *Save As* feature. This feature is found by right clicking the layer.

Plot a profile of a polyline on the raster.

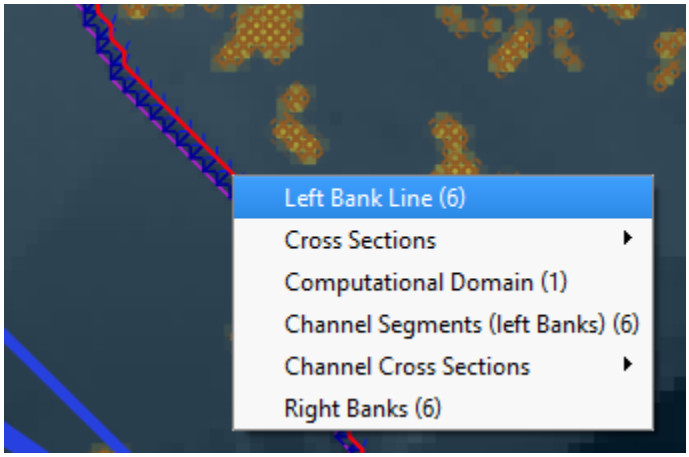
The profile tool works for Levee, Channel and Street polylines that are digitized in the user layers only. This example will test it on a channel polyline.

1. Click the *Raster profile* radio button and set the raster.

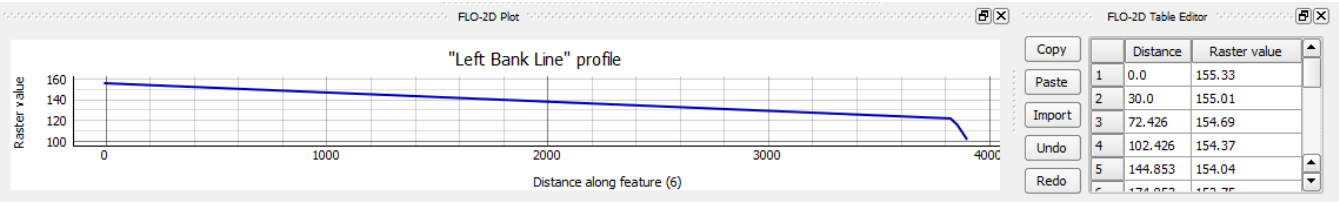
   This example uses *Grid Elevation* raster.



2. Click *the FLO-2D Info Tool* and click on a channel segment.



3. Select the segment and the elevation profile along the segment line is plotted in the FLO-2D Plot area and the Raster data along the channel line is printed to the FLO-2D Table. Any data that can be rasterized can be plotted in this manner. The table can be copied to an Excel sheet and saved.

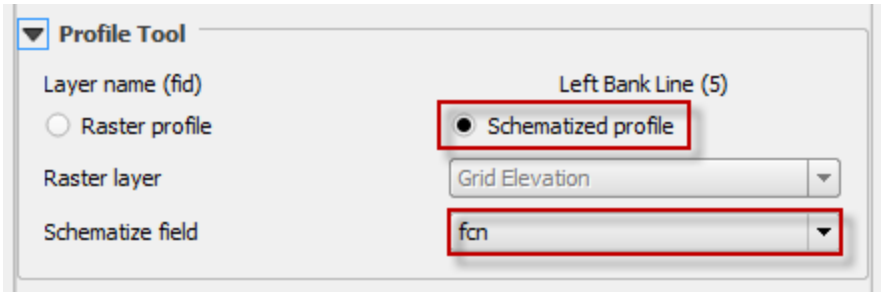


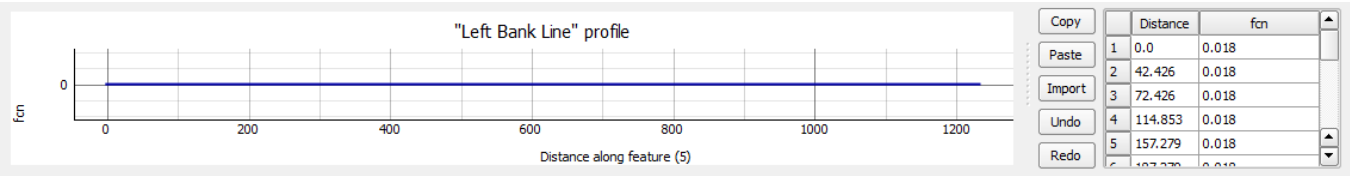
Plot a Profile of the Schematic Data

The Profile Tool will plot the data stored in the attribute table of the left bank layer. It also works for levees and streets.

1. Select the FLO-2D Info Tool and click the feature to be plotted.

2. Select *Schematized profile* and the field.





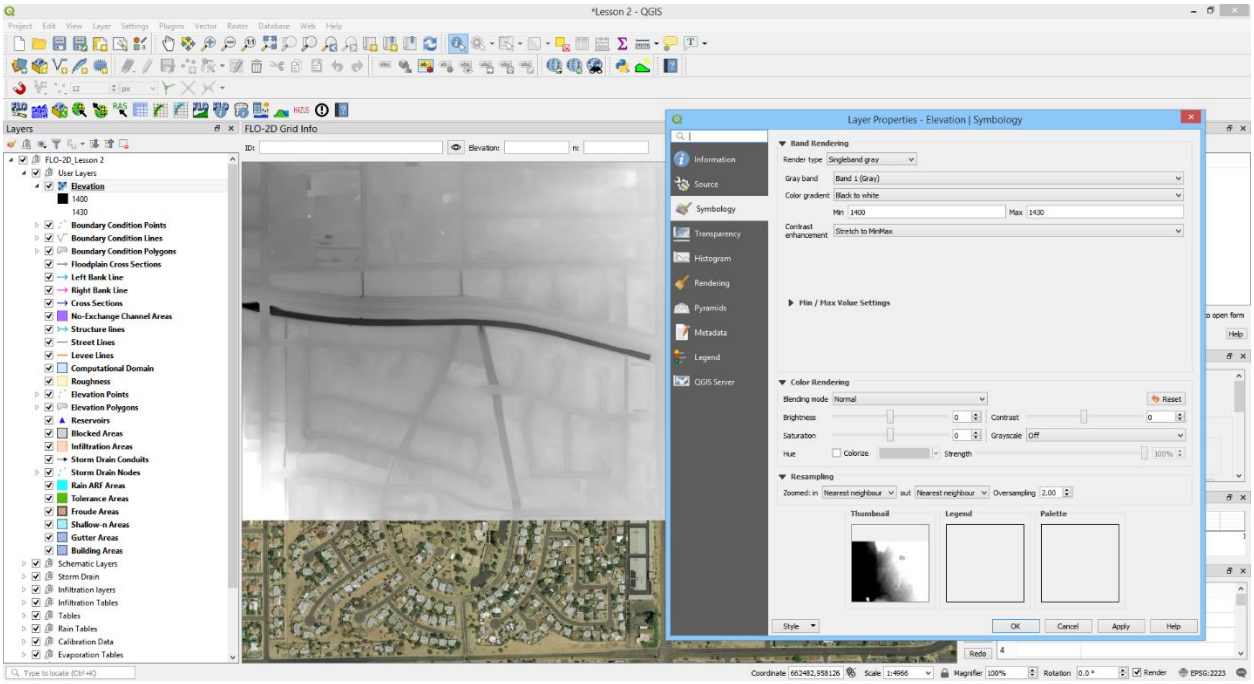
User Cross Sections Editor

This section will cover many of the tools available to develop channels for FLO-2D. This is not a step by step procedure. It is a compilation and detailed outline of the tools available and how to use them. For step by step instructions, see the workshop lessons.

Channels

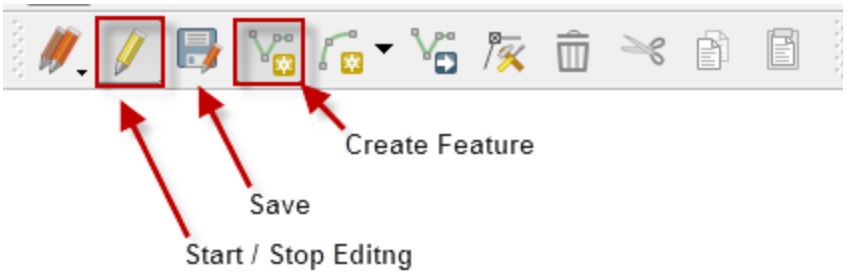
Identify the Channel

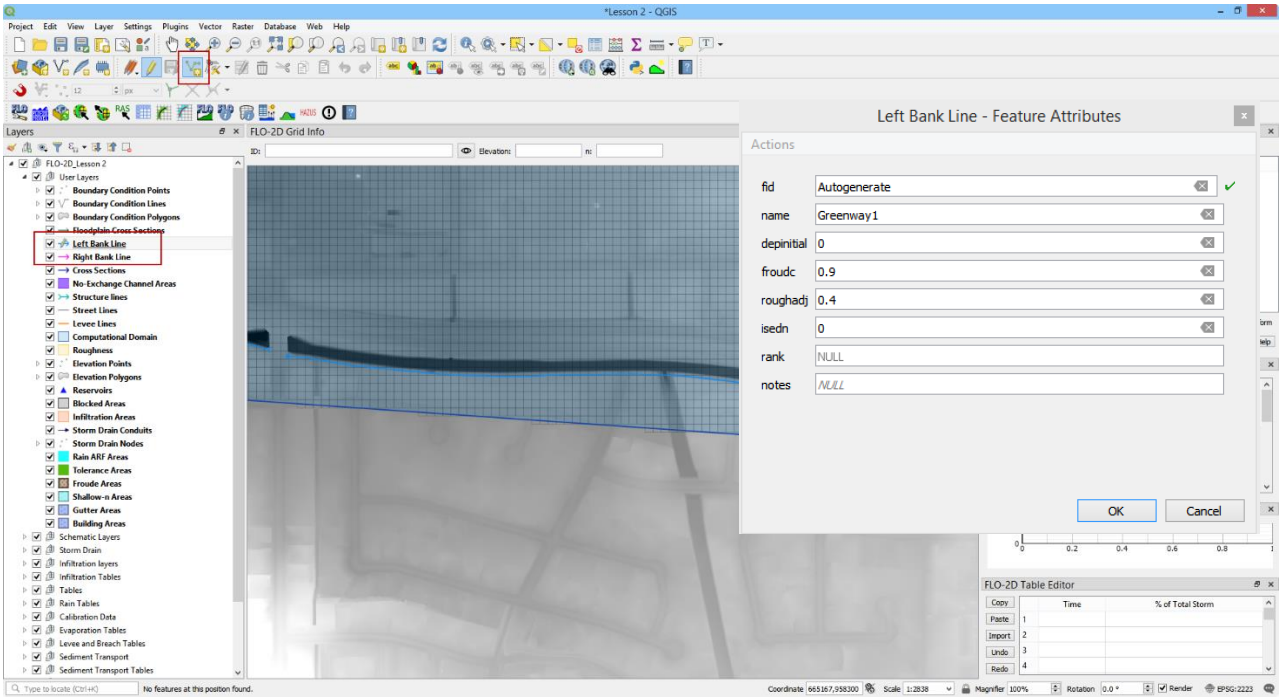
The elevation raster style can be edited so that the channel is well defined. Adjust the min / max view setting to enhance the channel depth.



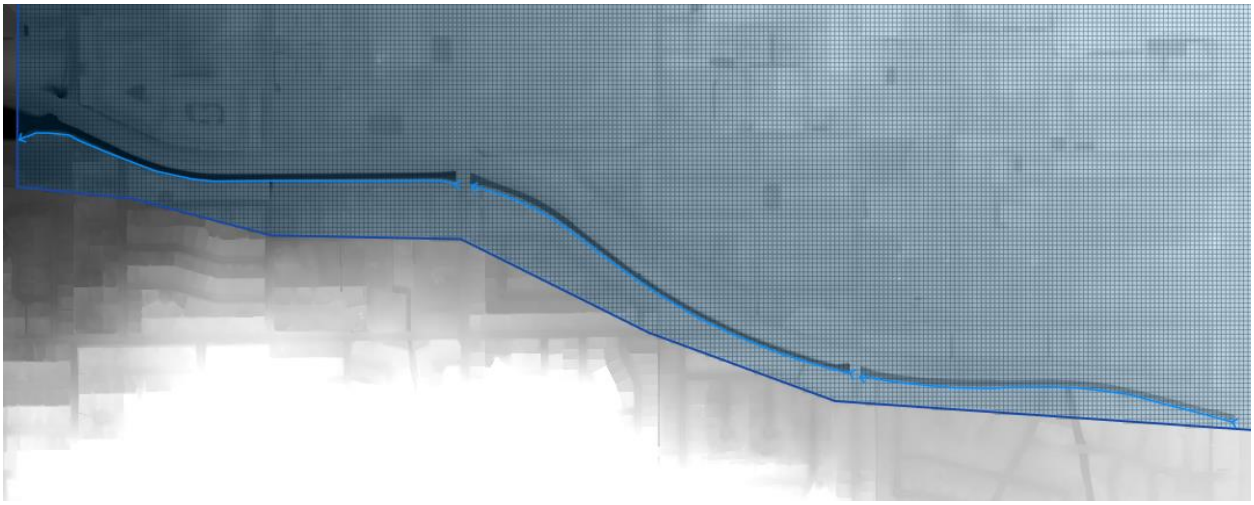
Left Bank

Select the *Left Bank Line* layer and click on both the start editing button and the add feature button. Create a polyline to represent the left bank looking downstream and click to save the layer.



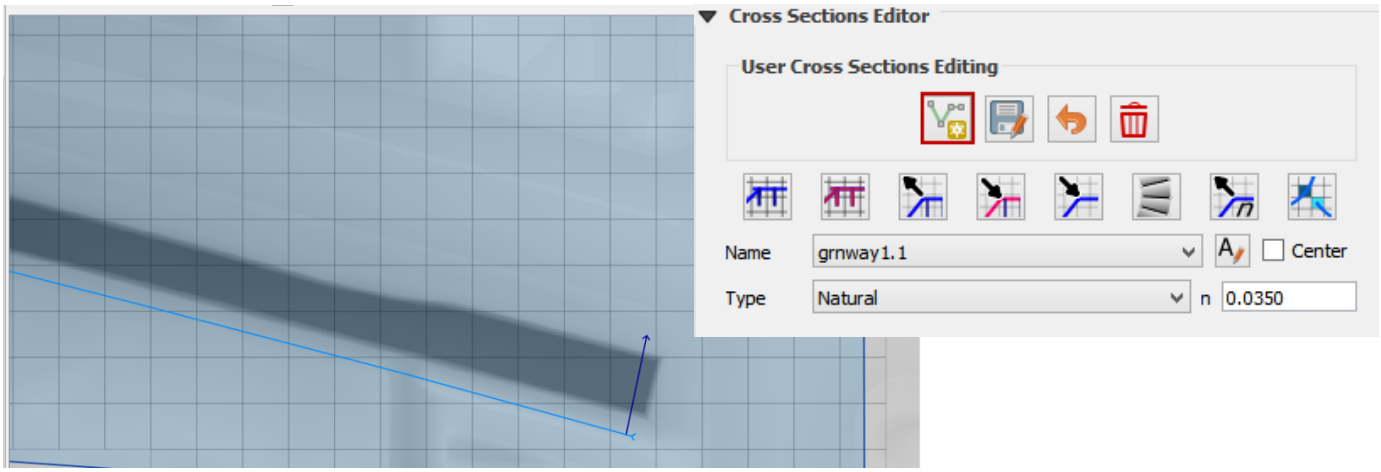


Repeat the process for the all the various channel segments and save.



Natural Cross Sections

To create natural cross sections, the cross sections should be numbered consecutively from upstream to downstream for each channel segment. Click the add Cross sections lines button and draw a polyline to represent each cross section. Click save to complete the process and load the data.

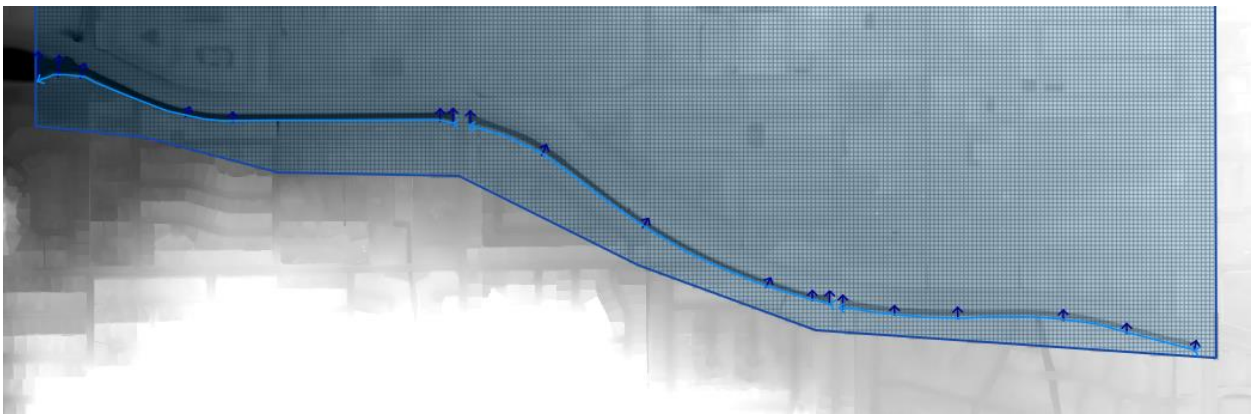


The cross sections shown in the following image have been digitized.

Note: The cross section slightly crosses the left bank. Draw the polyline from the left bank to the right bank. For the first cross section, make the first vertex in the first left bank grid element.

"Place holder data" is added to the cross section table when the save button is clicked.

Repeat the cross section digitizing process for each channel segment. The channel cross sections will appear as shown in the following figure when complete.

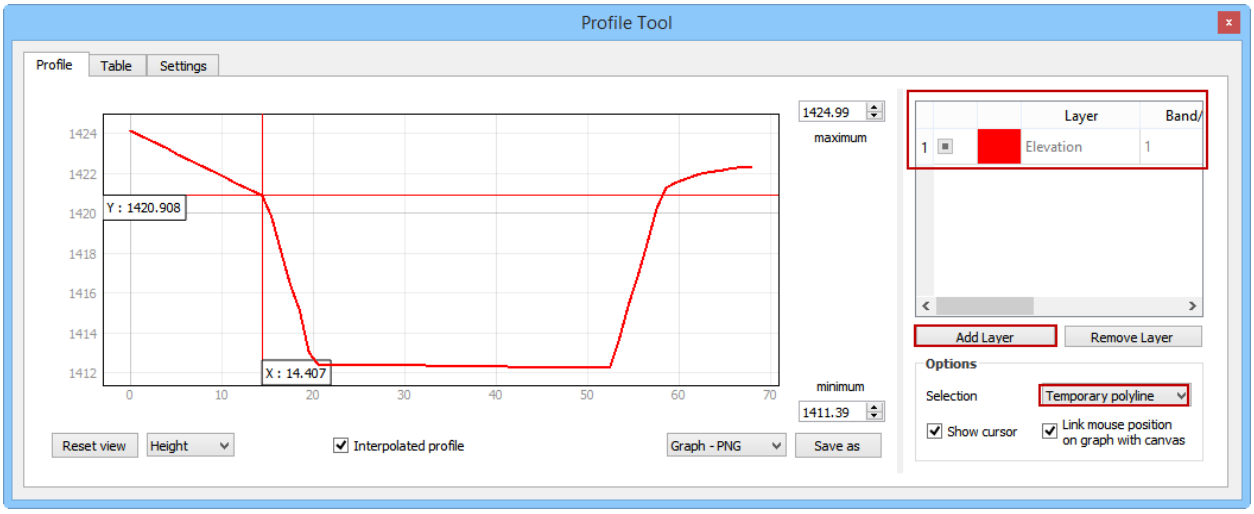


Station Elevation Data Development

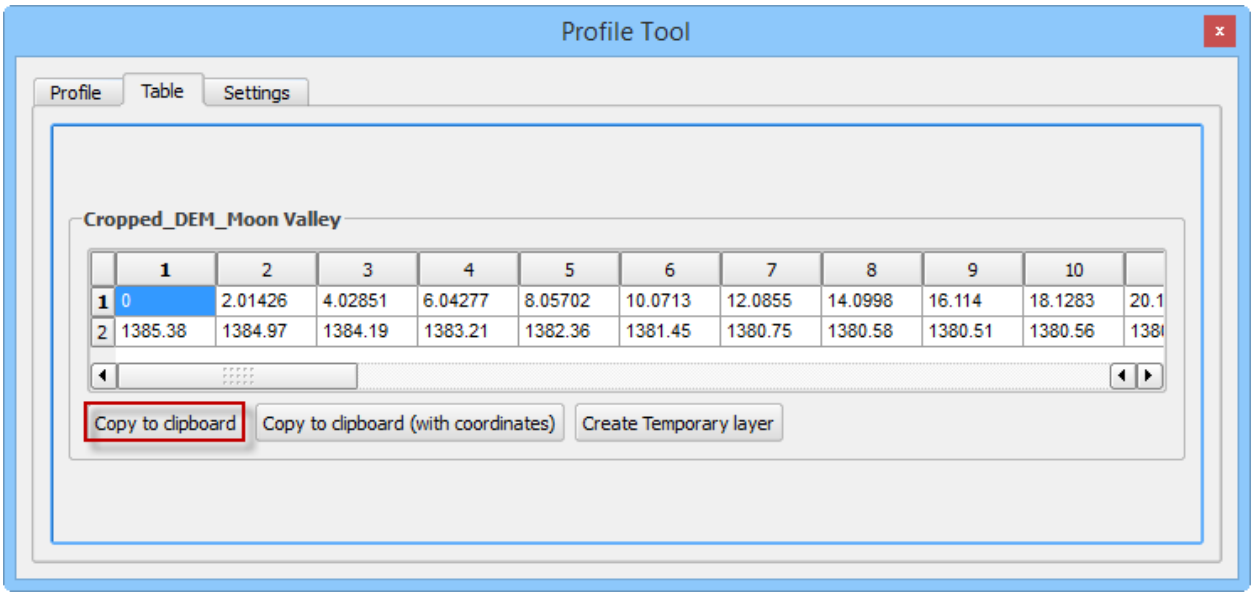
Several methods can be used to create the cross section data. This example will scan the map for station-elevation data using a plugin called Profile Tool.

1. Find and install the plugin Profile Tool.
2. Select the first cross section in the Cross Section Editor widget. This activates the cross section table and plot.

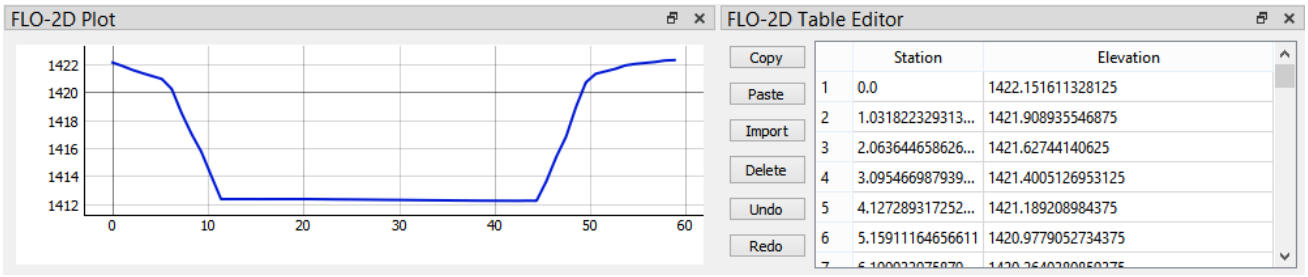
3. Click the Profile button to open the Profile Tool plugin,
4. Click the add layer button and select the Elevation Raster layer.
5. Draw a simple line over cross section 1.



6. The cross section station elevation data is listed in the Table tab shown below. Copy it to the clipboard.

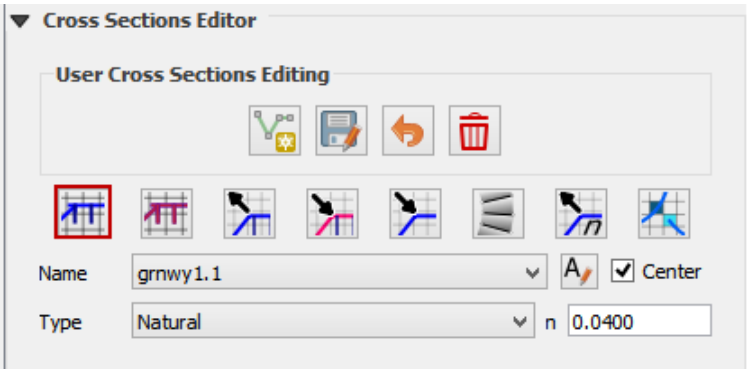


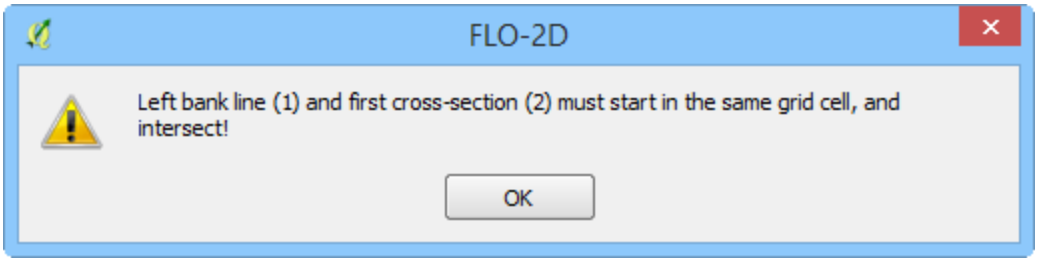
7. Place the cursor in the first cell of the FLO-2D Table Editor and click Paste. The cross section data is pasted to the table. Repeat the process for the remaining cross sections. The cross section is then loaded in the layer as shown below.



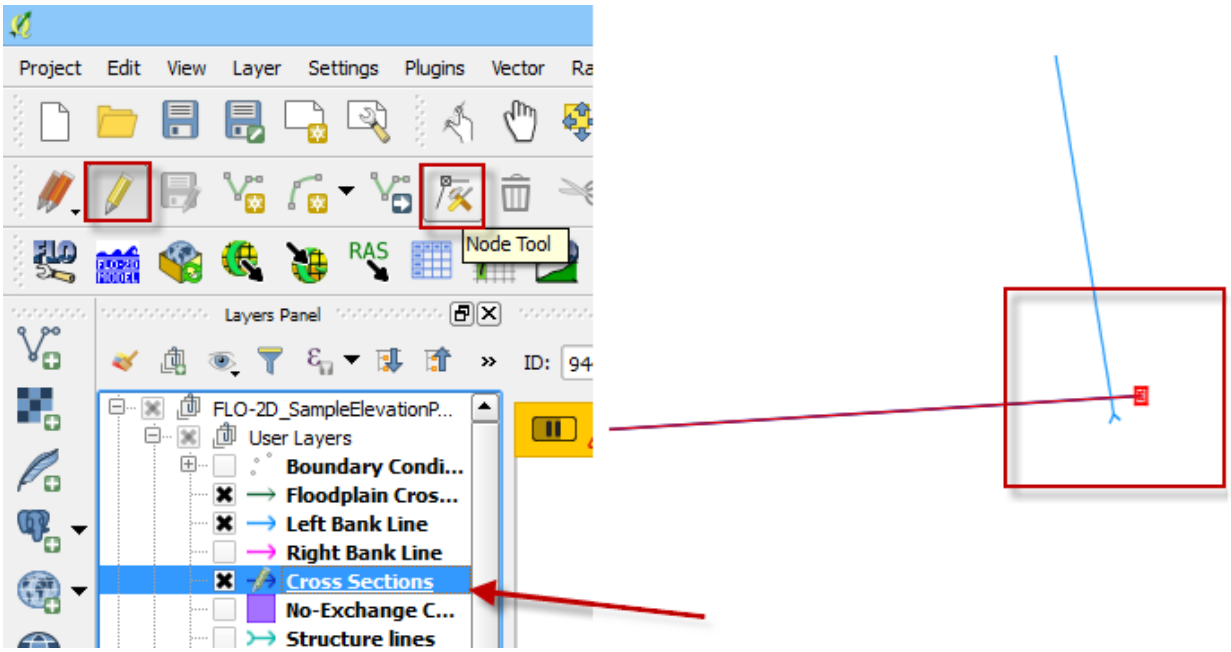
Schematize Channel

The next step is to Schematize the channels. Click the Schematize left banks and cross sections.



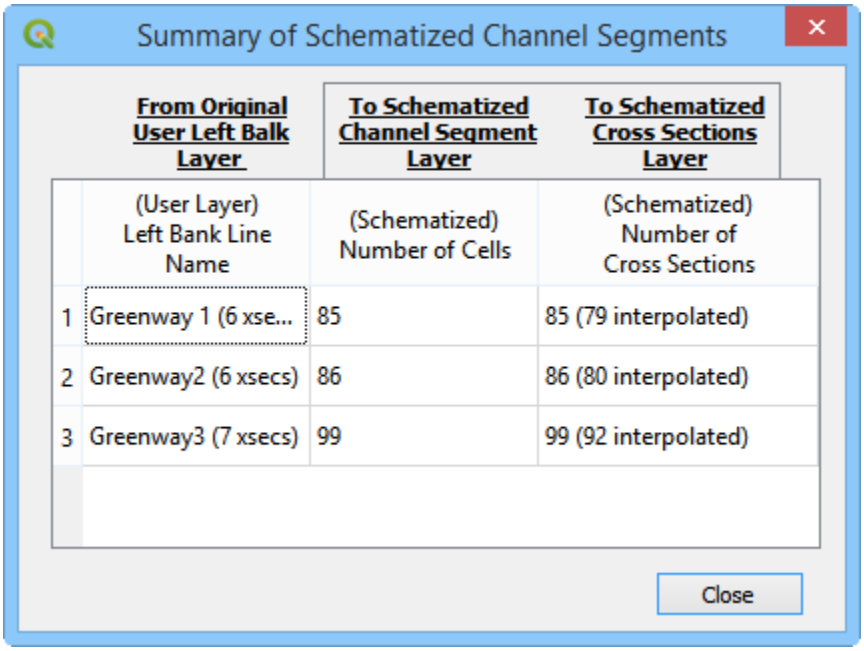


Make sure each cross section crosses the left bank and touches it. If one a cross section is not correct, select the cross section layer, click edit and select the node tool. Click the node and make the adjustment. Stop editing and save the edits. Rerun the Schematize tool.

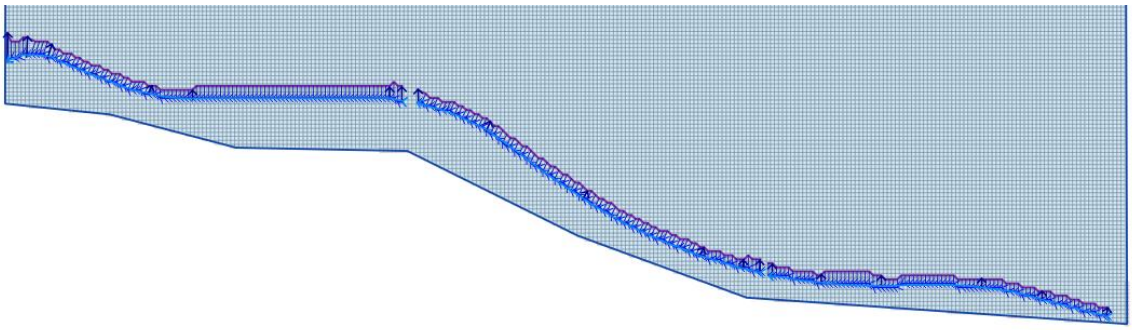


If the channel schematization process was successful, the following message will appear. Click close to load the channel info for the schematized layer.

The message shows that 5 cross sections in the first channel segment and 9 cross sections in the second channel segment were schematized. This matches the dataset.

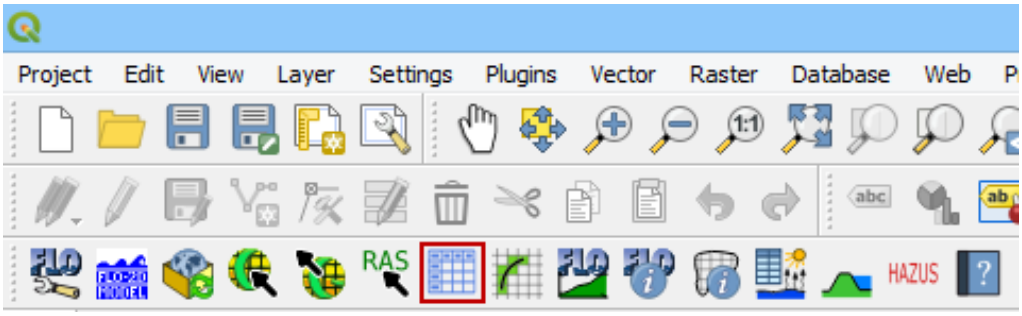


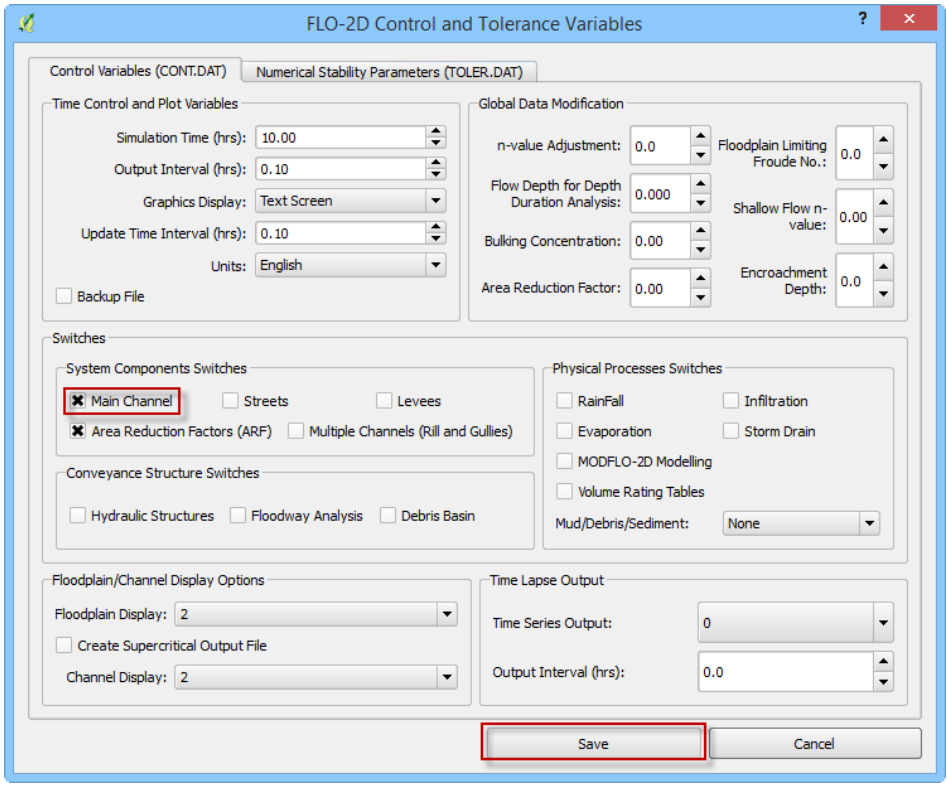
The schematized layers now have complete left bank, right bank, and cross section data. Adjust cross section and left bank alignment now. It is easier apply changes before interpolating the cross section data.

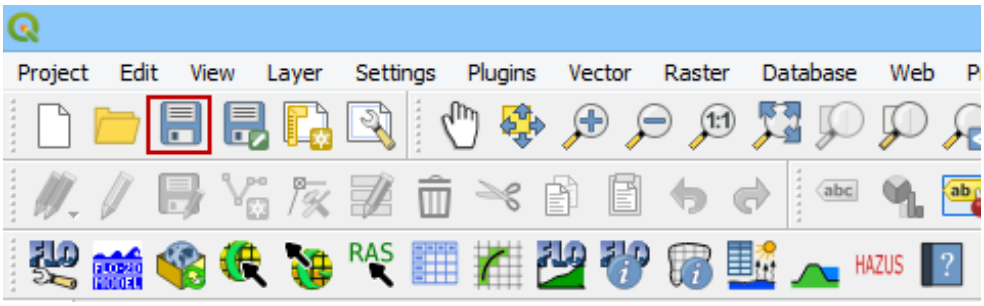


Save the Project

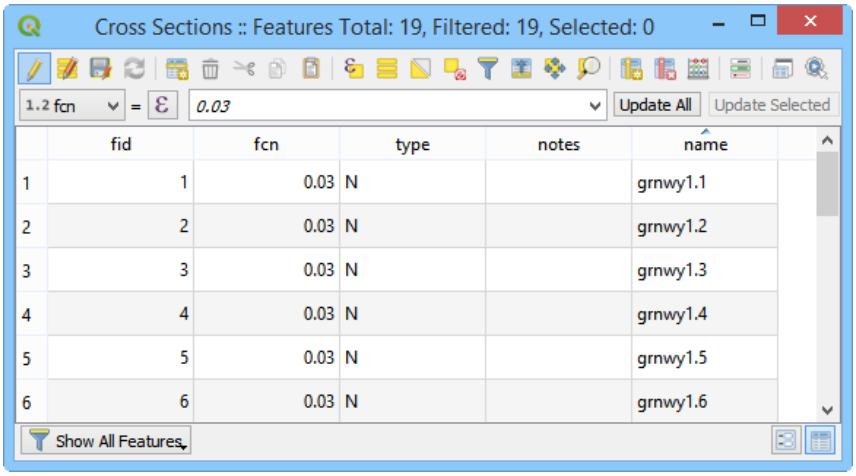
Check the channel switch in the control variables window. Use the Save Project button to save the QGIS project.


Project
Edit
View
Layer
Settings
Plugins
Vector
Raster
Database
Web
RAS
HAZUS


FLO-2D Control and Tolerance Variables
Control Variables (CONT.DAT)
Numerical Stability Parameters (TOLER.DAT)
Time Control and Plot Variables
Simulation Time (hrs): 10.00
Output Interval (hrs): 0.10
Graphics Display: Text Screen
Update Time Interval (hrs): 0.10
Units: English
Backup File
Global Data Modification
n-value Adjustment: 0.0
Floodplain Limiting Froude No.: 0.0
Flow Depth for Depth Duration Analysis: 0.000
Shallow Flow n-value: 0.00
Bulking Concentration: 0.00
Area Reduction Factor: 0.00
Encroachment Depth: 0.0
Switches
System Components Switches
Main Channel
Streets
Levees
Area Reduction Factors (ARF)
Multiple Channels (Rill and Gullies)
Conveyance Structure Switches
Hydraulic Structures
Floodway Analysis
Debris Basin
Physical Processes Switches
RainFall
Infiltration
Evaporation
Storm Drain
MODFLO-2D Modelling
Volume Rating Tables
Mud/Debris/Sediment: None
Floodplain/Channel Display Options
Floodplain Display: 2
Create Supercritical Output File
Channel Display: 2
Time Lapse Output
Time Series Output: 0
Output Interval (hrs): 0.0
Save
Cancel

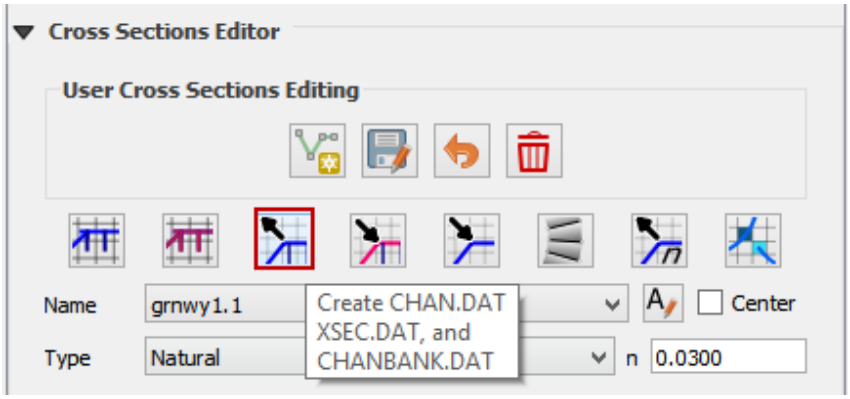

Project
Edit
View
Layer
Settings
Plugins
Vector
Raster
Database
Web
RAS
HAZUS

Inspect the cross section n-value field to ensure all n-values are present. If missing, fill the required n-value to the field.

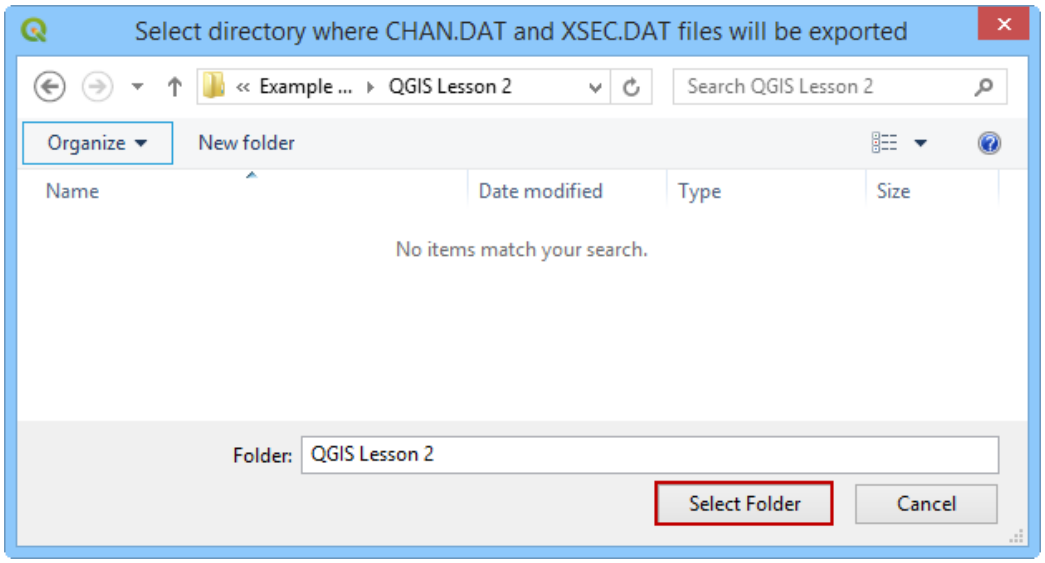


	fid	fcn	type	notes	name
1	1	0.03	N		grnwy1.1
2	2	0.03	N		grnwy1.2
3	3	0.03	N		grnwy1.3
4	4	0.03	N		grnwy1.4
5	5	0.03	N		grnwy1.5
6	6	0.03	N		grnwy1.6

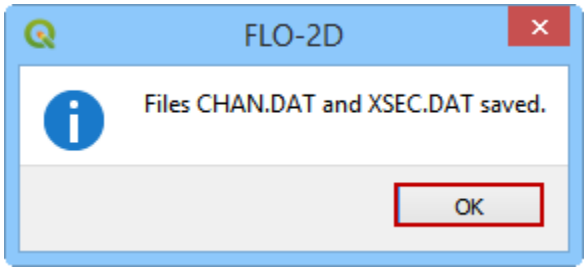
To interpolate the channel segments, export the channel data and run the interpolator.



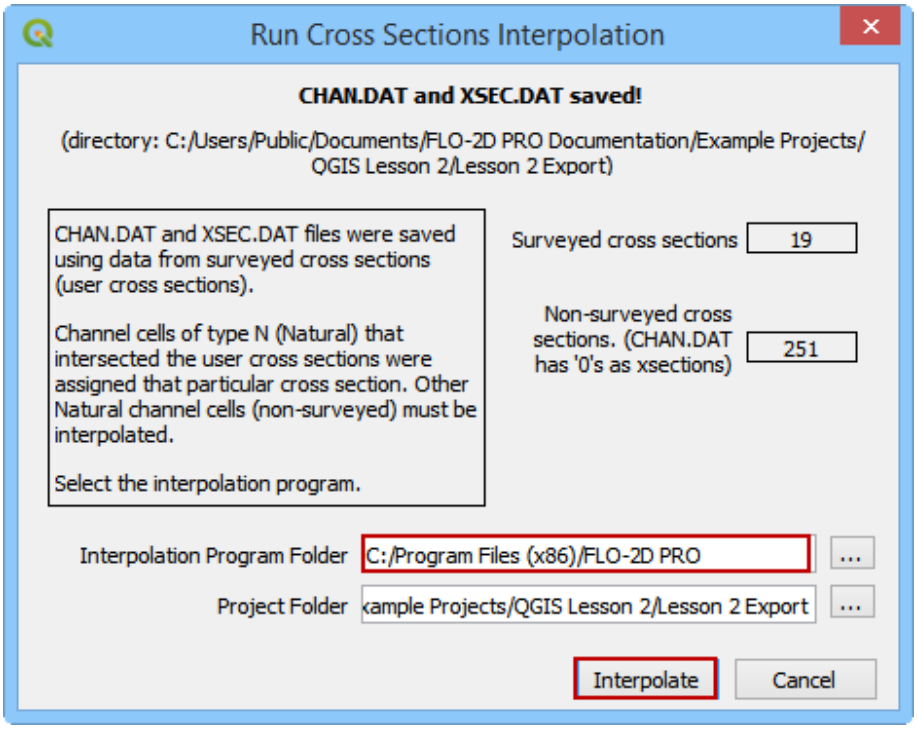
Select the folder where the *.DAT files will be saved.



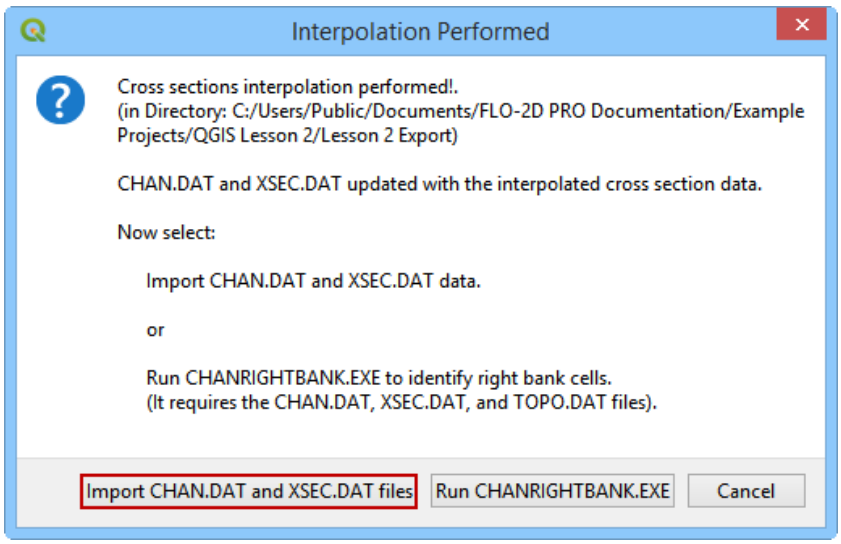
Once the data files are written, click ok to close the following dialog box.



Select the FLO-2D Pro Folder and click Interpolate.



If the interpolation is performed correctly, the following message will appear. Get the new data into the GeoPackage by clicking Import CHAN.DAT, AND XSEC.DAT.

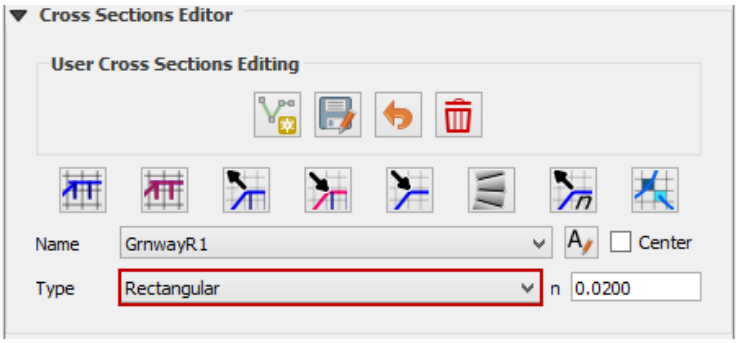


Prismatic Cross Sections

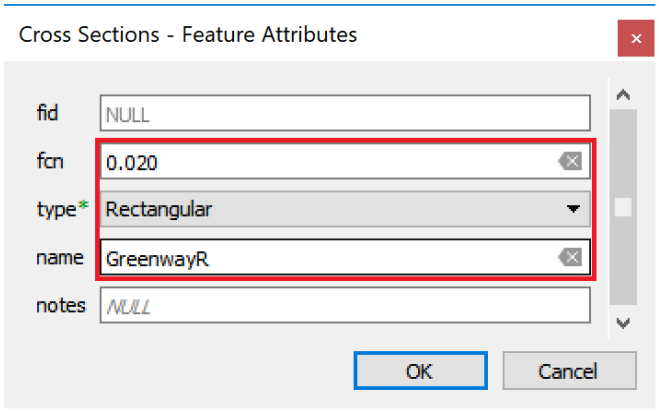
Prismatic channel data can be entered and interpolated using the cross section editor. Use this option for creating Rectangular and Trapezoidal channel segments. This example will use two segments of channel data. One for a rectangular channel and one for a trapezoidal channel.

Rectangular Cross Sections

1. Set up the Editor Widget. Type = Rectangular base n = 0.020

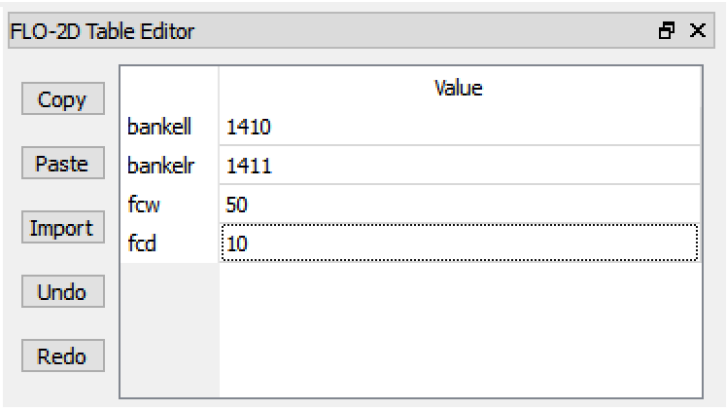


2. Click the create cross section button.
3. Draw the first cross section and enter the Feature Attributes.



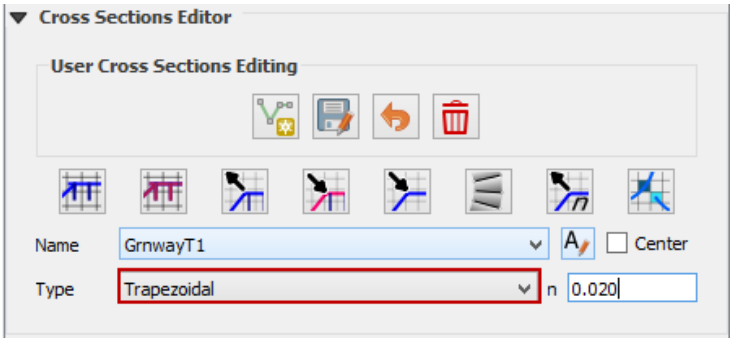
4. Click Save to load the cross section into the Table Editor.

5. Edit the cross section left and right bank elevation and geometry in the table. Repeat the process for each cross section.

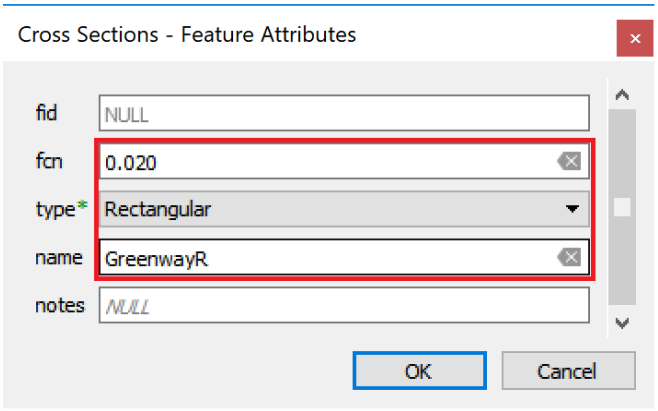


Trapezoidal Cross Sections

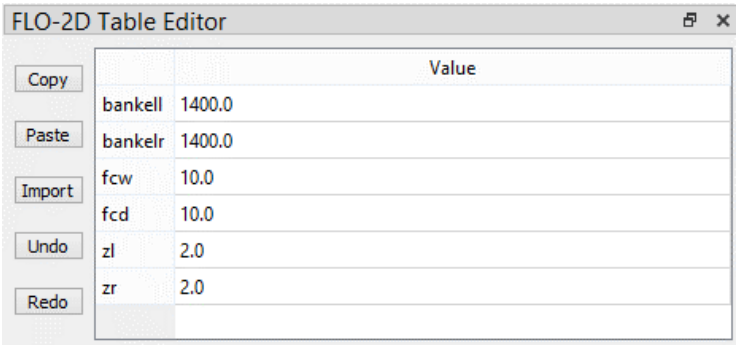
6. Set up the Editor Widget. Type = Trapezoidal base n = 0.020

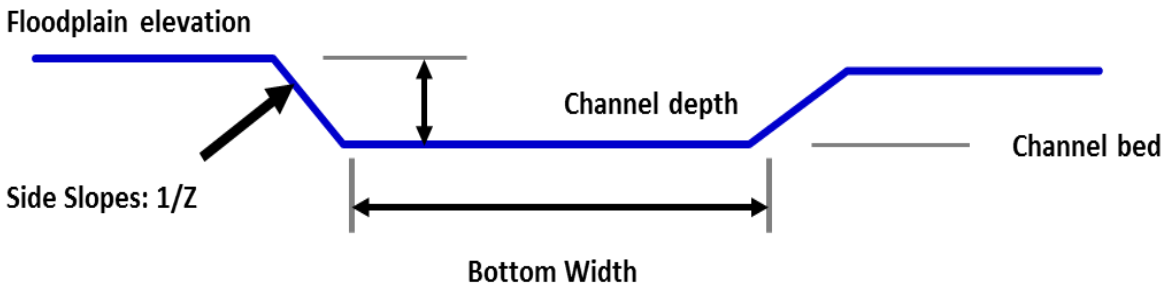


7. Click the create cross section button.
8. Draw the first cross section and enter the Feature Attributes.



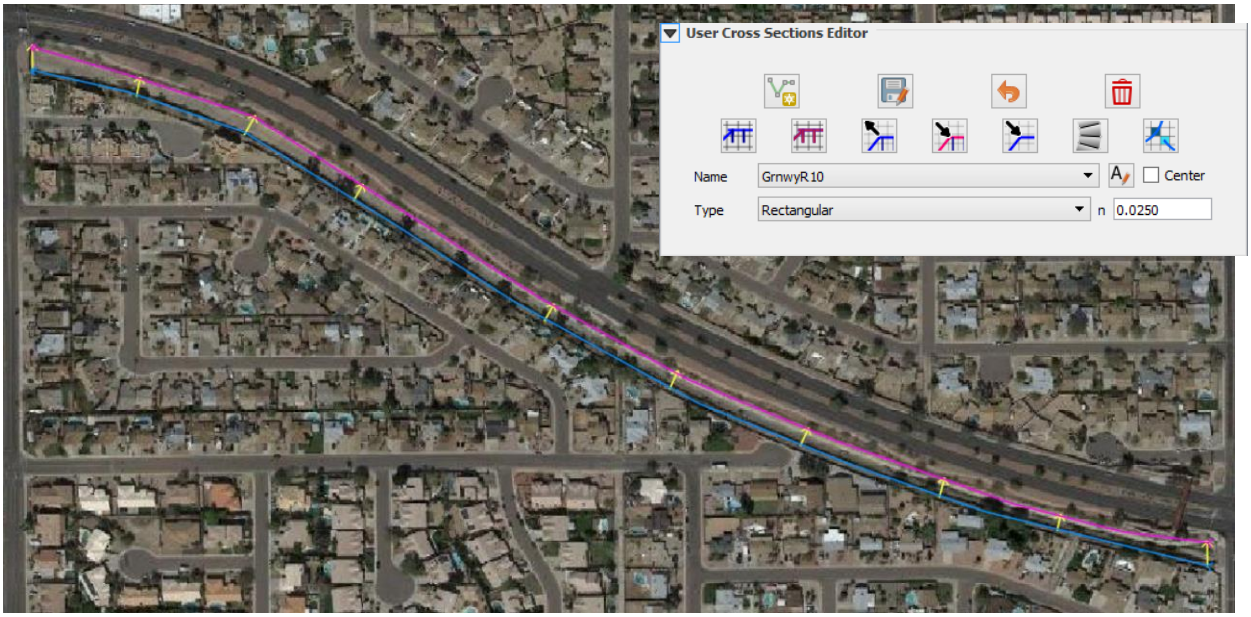
9. Click Save to load the cross section into the Table Editor.
10. Edit the cross section left and right bank elevation and geometry in the table. Repeat the process for each cross section.





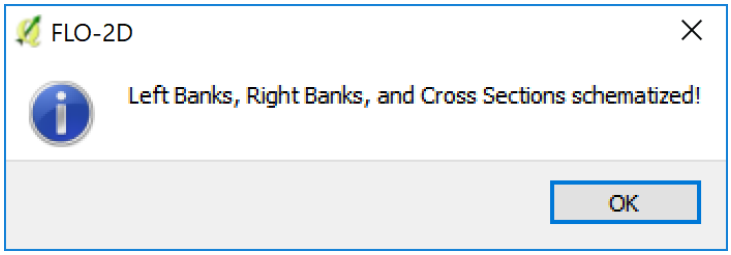
Schematize Rectangular and Trapezoidal Channel Segment

In this example, ten Rectangular, ten Trapezoidal and 10 natural cross sections are digitized.

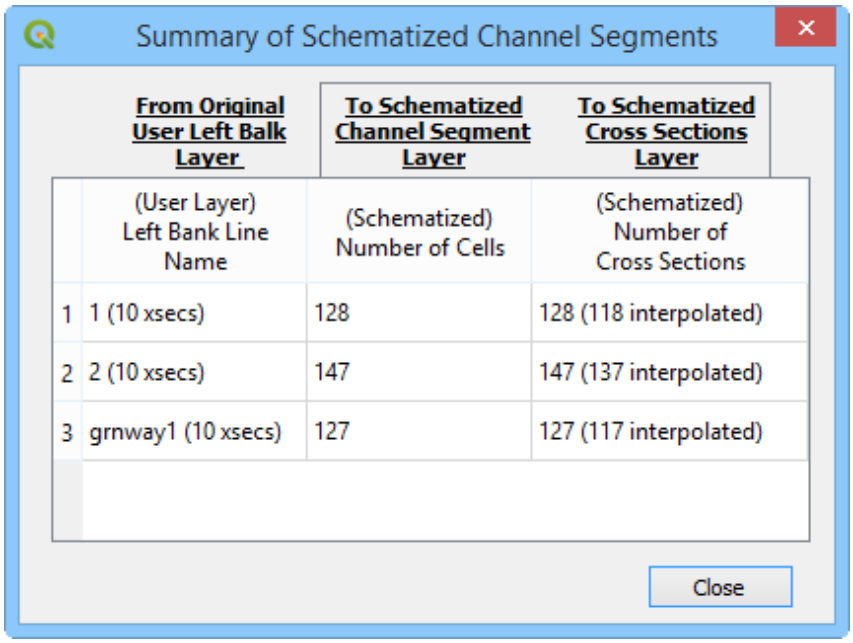


Click the Schematize button.

If the following message appears, the schematization worked properly.



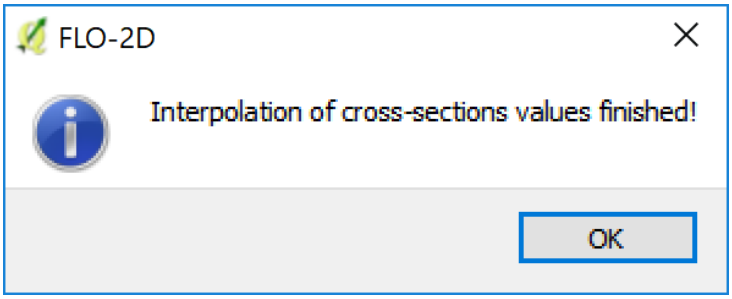
This dialog box shows the number of original cross sections and the number of schematized cross sections.



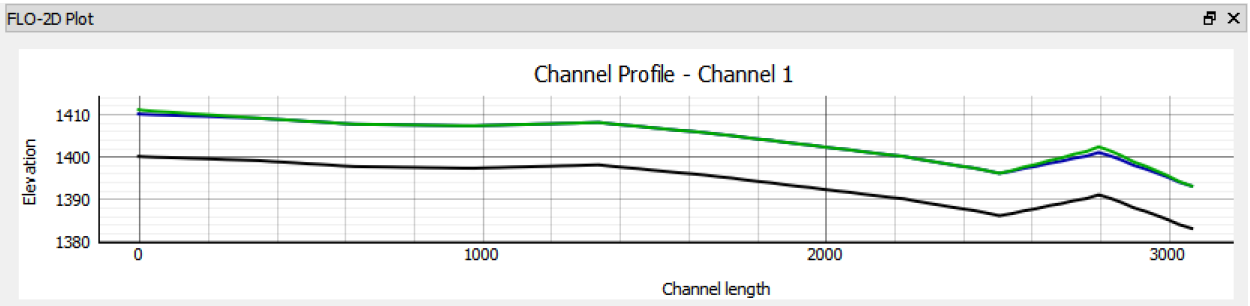
Interpolate Prismatic Channel Data

Click the Interpolate button to interpolate the left and right bank of the rectangular channel.

If the process finished correctly, the following box will appear. Click OK to close the box.

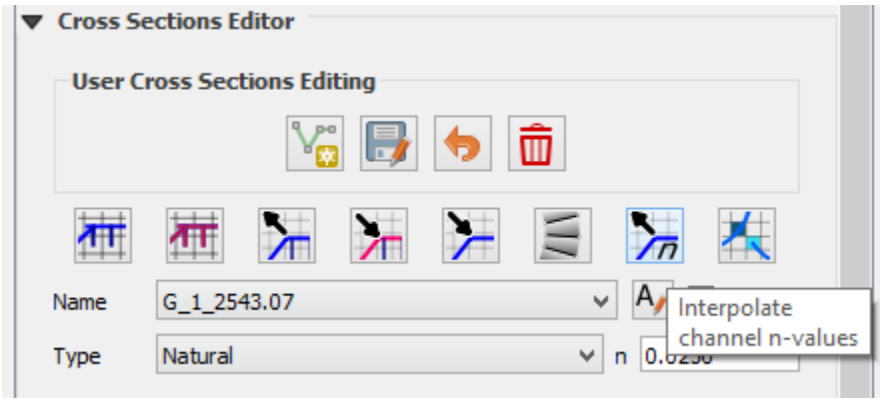


Click the channel profile tool and the left bank to check the profile of the channel.

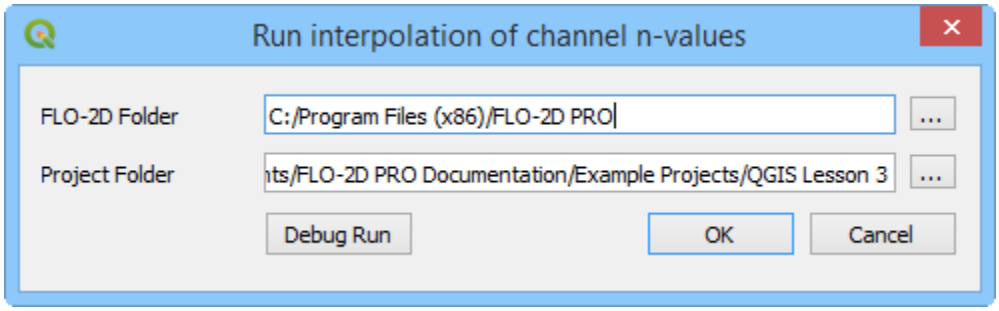


Channel n-Value Interpolator

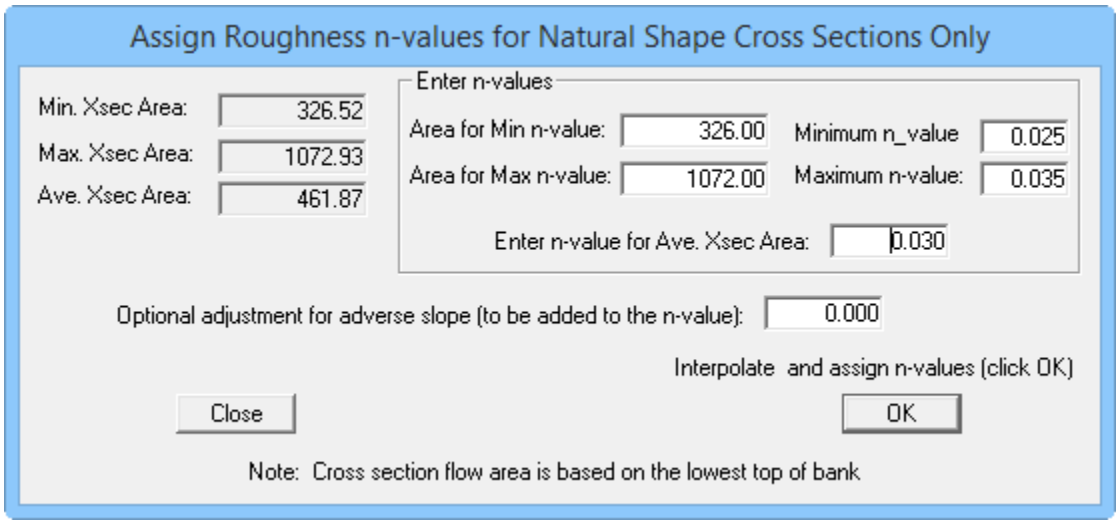
The channel n-Value interpolator tool is used to define the n-value of a channel cross section based on the cross section geometry.



The button calls the tool externally.



The tool assigns an n-value for the chan.dat file based on the picture below. The user can choose the n values for a minimum or maximum cross section area.



Boundary Condition Editor

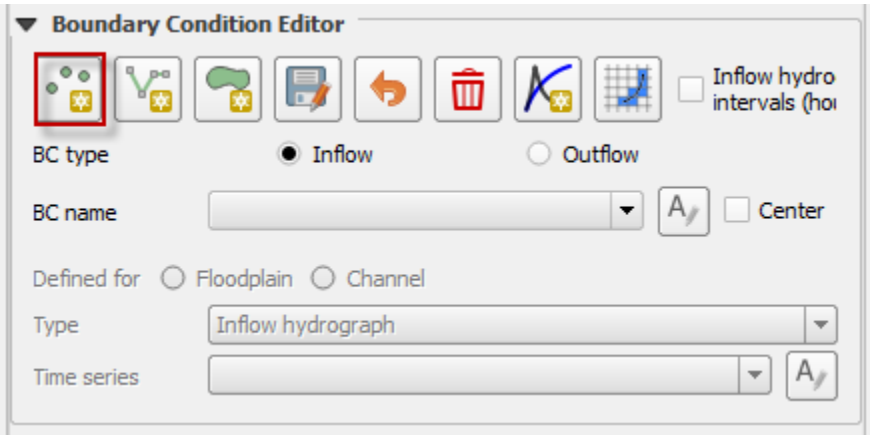
The boundary condition editor is used to define the inflow and outflow boundaries. The data is written to the INFLOW.DAT and OUTFLOW.DAT files.

Inflow Boundary Condition Editor

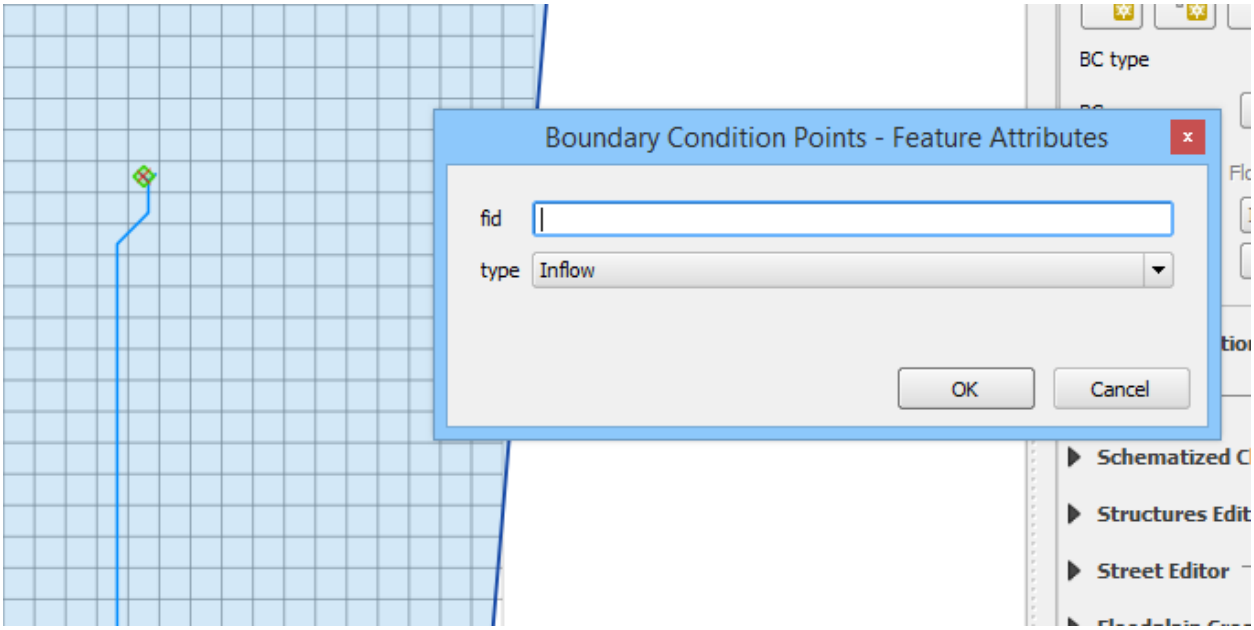
Any number of inflow hydrographs to the FLO-2D model can be assigned to channel, floodplain or even the 1-D street component. This represents a flood inflow to the flow domain from an off-site source.

Inflow Node

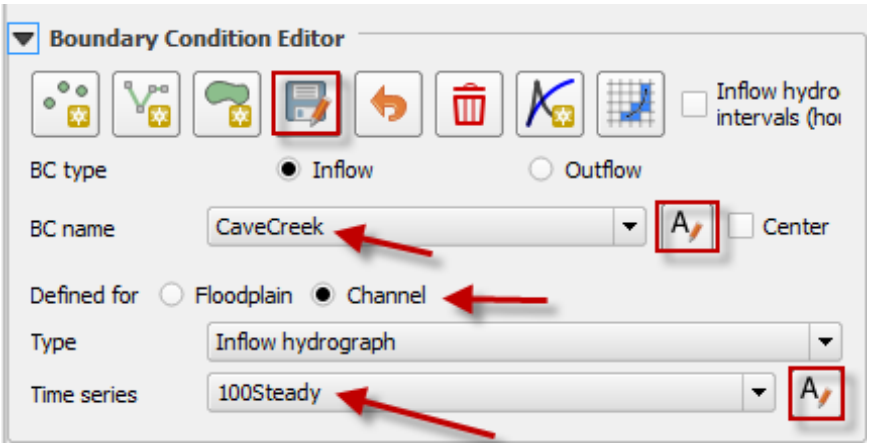
To create a point of inflow, click the Add Point BC button on the Boundary Condition Editor widget.



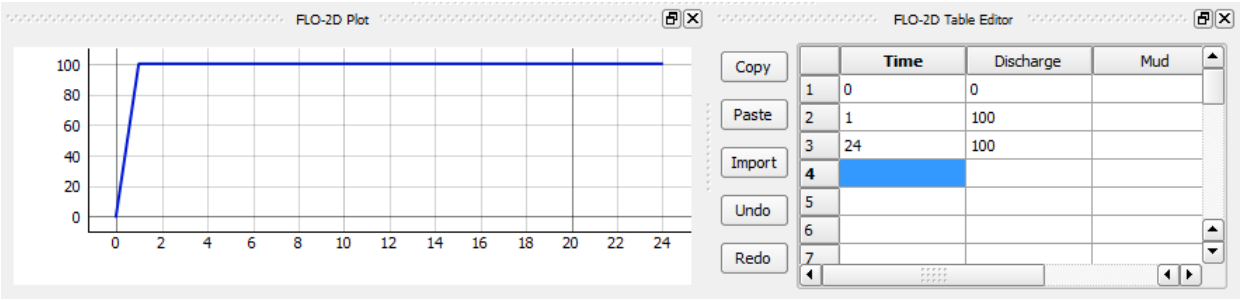
Identify the inflow node by left clicking the location of the inflow node on the map. In this example, the inflow node is a channel inflow node. It is not necessary to enter the fid. Click *OK* to create the feature.



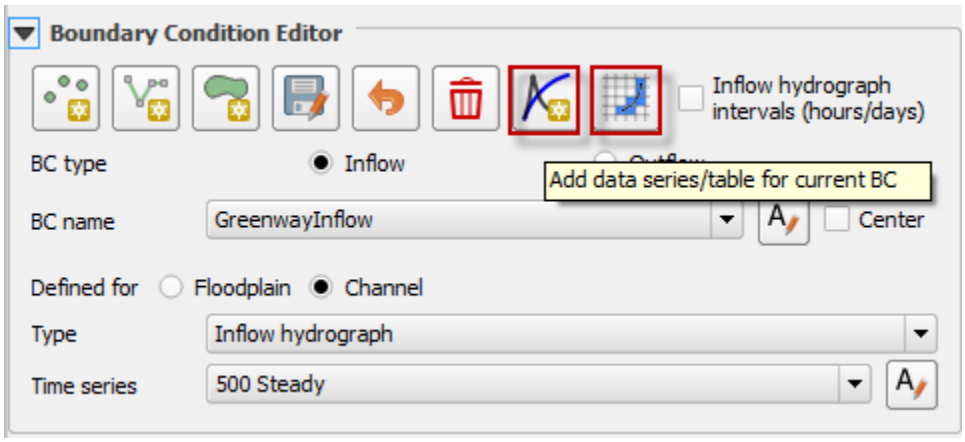
1. Click the Save Button to load the data into the editor.
2. Assign the defining detail conditions to the inflow node in the following image.
3. This example inflow node will have a steady hydrograph with 100 cfs assigned to Cave Creek inflow node.

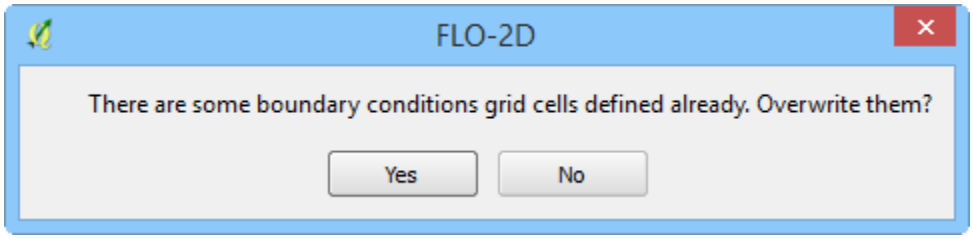


The time series inflow hydrograph is assigned in the table editor where time is in hours and discharge is cfs or cms. This is a clear water inflow hydrograph and no sediment concentration is assigned.



Repeat the process to add additional inflow hydrographs. Use the *Add* data series/table for current BC button to create a new hydrograph. Use the Schematize icon (shown below) to save the data to the Schematic Layers and click *Yes* to overwrite the layers.





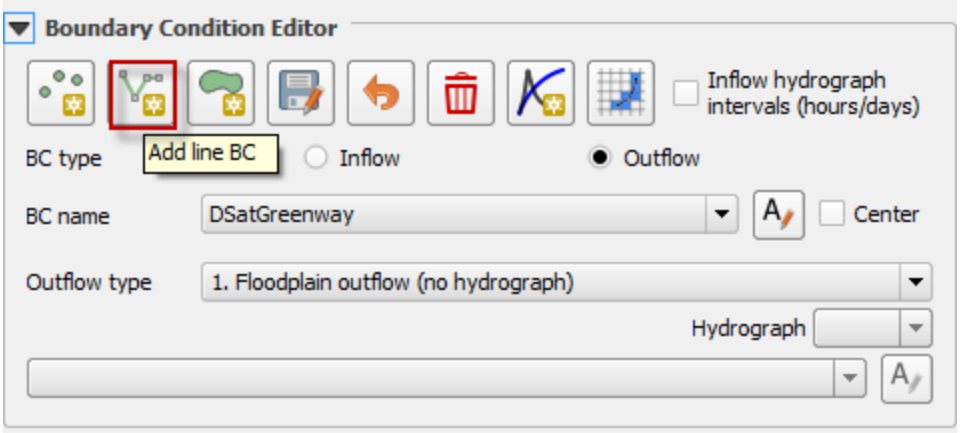
Outflow Boundary Conditions

Boundary outflow conditions include stage time relationships, stage discharge relationships or normal depth control. These conditions define how the channel or floodplain discharge enters or leaves the FLO-2D model domain.

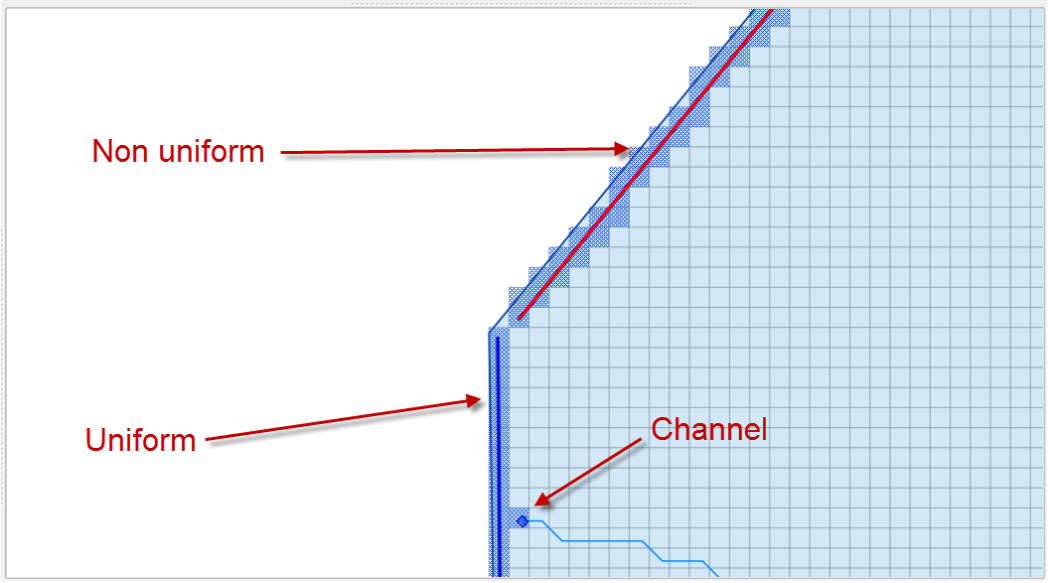
Outflow Floodplain

For a FLO-2D model normal depth outflow condition where the flow leaves the flow domain without effecting the upstream water surface elevation, select the outflow nodes along the downstream boundary or any location in the project domain.

1. Click the *Add line BC* button and draw a line to represent the outflow cells.
2. Click *Save* to load save the feature to the layer and activate the editor.
3. Name the boundary and set the boundary conditions (Outflow type).



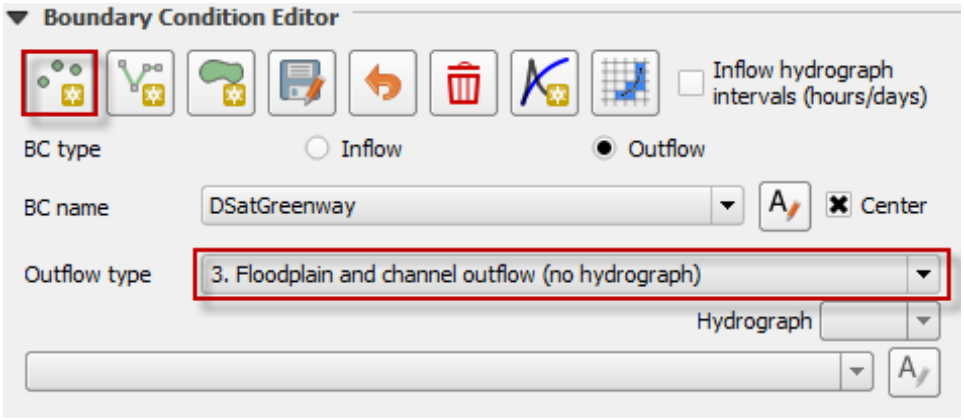
For multiple outflow nodes in a line, the outflow nodes should be continuous with no gaps (across diagonals). If the outflow nodes are stacked up (two or more elements wide), the model will generate an error message at runtime indicating that some outflow nodes have no access to upstream flow.

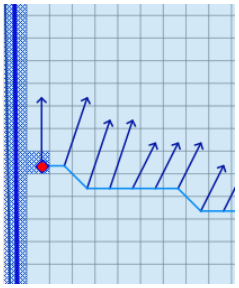


Outflow Channel

For channel outflow at a normal depth condition, select the channel outflow node on the downstream boundary or channel segment terminus.

1. Click add a point boundary and click the last channel node.
2. Click *Save*.
3. Name the feature.
4. Set the outflow condition for the channel. A combined floodplain and channel outflow condition (3) is the typical setting.





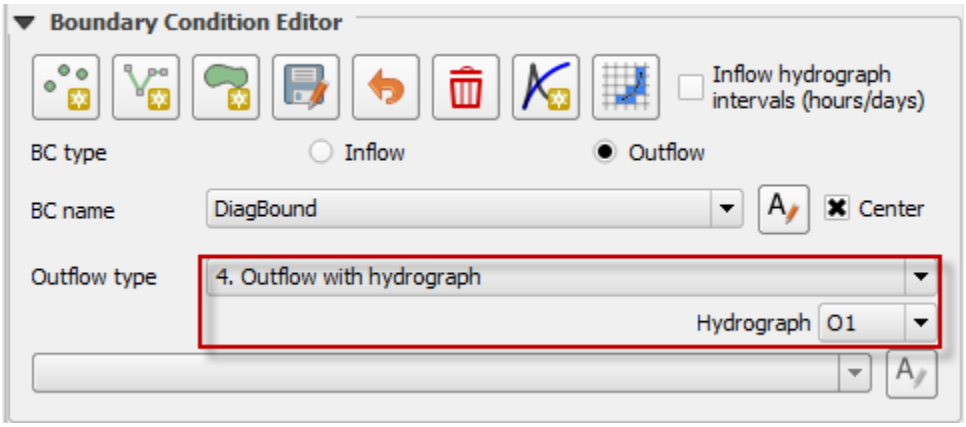
Outflow with Hydrograph

Use this option with any floodplain downstream boundary that will capture the outflow hydrograph and report it to file as inflow to a second downstream FLO-2D model. This outflow option will write a line ID of O1 thru O9 lines in the outflow.dat file. It requires the cadpts_ds1.dat thru cadpts_ds9.dat data files from the downstream models to calculate the inflow_ds1.dat thru inflow_ds9.dat files at runtime (see the FLO-2D Data Input Manual). Select the outflow nodes along the downstream boundary to assign the hydrograph boundary condition.

1. Click the *Add line BC* button and draw a line through the desired outflow nodes.

2. Click *Save* to load save the feature to the layer and activate the editor.

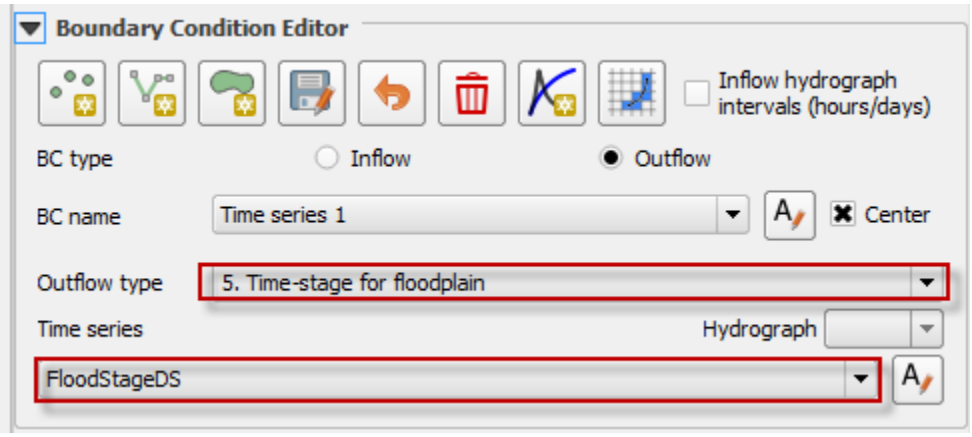
3. Name the boundary condition and set the boundary *Outflow type* conditions as *Outflow with Hydrograph*.



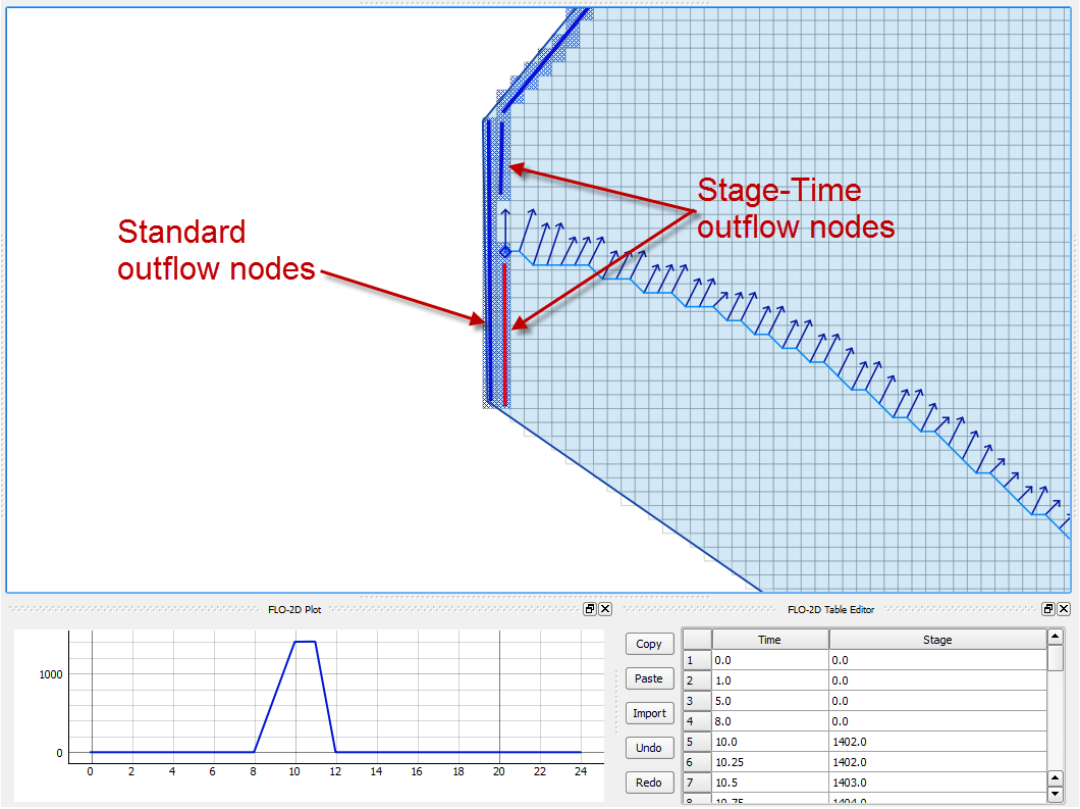
Outflow with Time – Stage Hydrograph for Floodplain

To represent variable time-stage boundary conditions such as tides, storm surge, tsunamis, or flooding from a large river, use this outflow option. The time–stage relationship can be synchronized with rainfall and upstream watershed flooding. Select the outflow nodes along the downstream boundary as a row of elements inside the boundary (by one or more rows of elements).

1. Click *Save* to load save the feature to the layer and activate the editor.
2. Name the boundary and set the boundary conditions.
3. Name and fill the Time Series table.



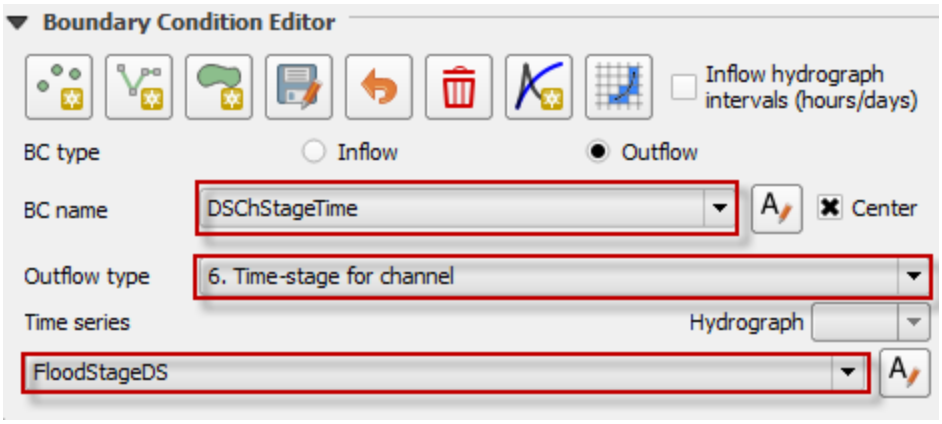
There are two types of outflow time-stage conditions, one where the inflow hydrograph moves upstream and there is no outflow downstream and a second possible condition where after the storm surge recedes, the downstream flow resumes and flows off the grid system. This second option requires a double row of outflow nodes. The row along the boundary is a conventional outflow node operating at a normal depth. The upstream row is a time-stage outflow node. When the upstream water surface elevation is greater than the stage in the time-stage outflow node, the time-stage condition is turned off and the downstream flows goes through the time-stage outflow elements to the standard outflow nodes for the normal depth flow condition.

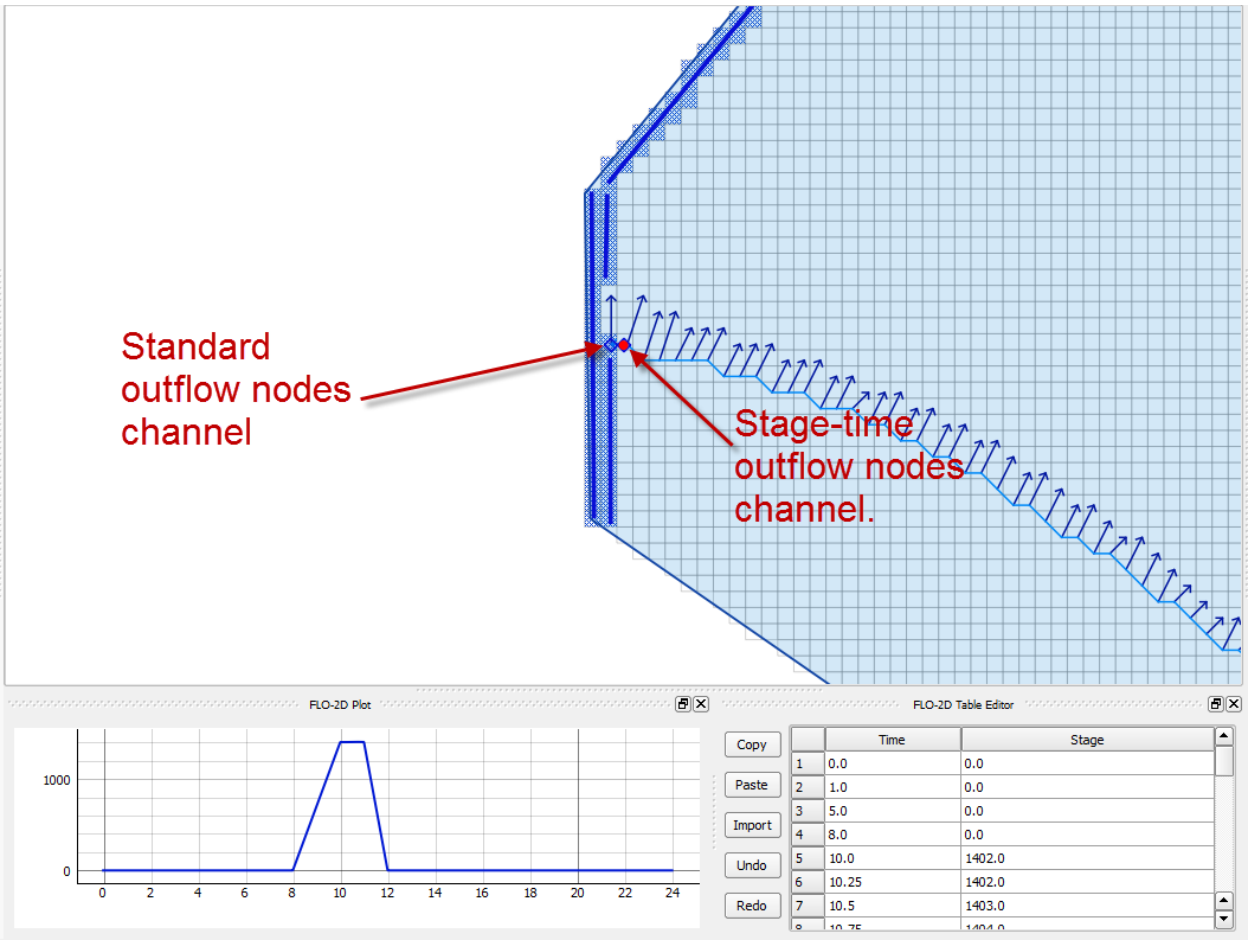


Outflow with Time – Stage Hydrograph for Channel

Similar to the time-stage condition for the floodplain select this option to represent ocean tide, storm surge, tsunamis, or flooding from a large river control in a channel terminus. The time – stage relationship can be synchronized to rainfall and watershed flooding.

1. Select the standard outflow node at the end of the channel.
2. Set the time-stage node one element upstream.
3. Click *Save* to load save the feature to the layer and activate the editor.
4. Name the boundary and set the boundary conditions.
5. Name and fill the Time Series table.

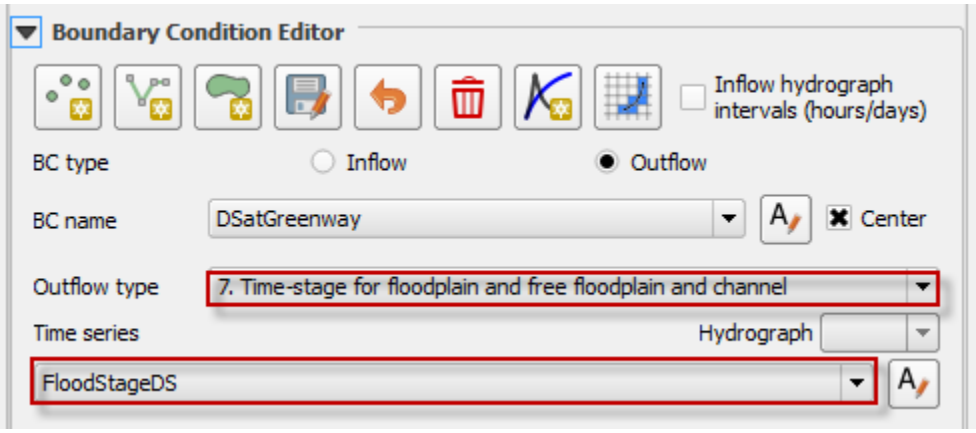

Boundary Condition Editor
Inflow hydrograph intervals (hours/days)
BC type
Inflow
Outflow
BC name
DSChStageTime
Center
Outflow type
6. Time-stage for channel
Time series
Hydrograph
FloodStageDS


Standard outflow nodes channel
Stage-time outflow nodes channel.
FLO-2D Plot
1000
0
0 2 4 6 8 10 12 14 16 18 20 22 24
FLO-2D Table Editor
Copy
Paste
Import
Undo
Redo
Time
Stage
1 0.0 0.0
2 1.0 0.0
3 5.0 0.0
4 8.0 0.0
5 10.0 1402.0
6 10.25 1402.0
7 10.5 1403.0

Time-Stage for Floodplain and Free Floodplain and Channel

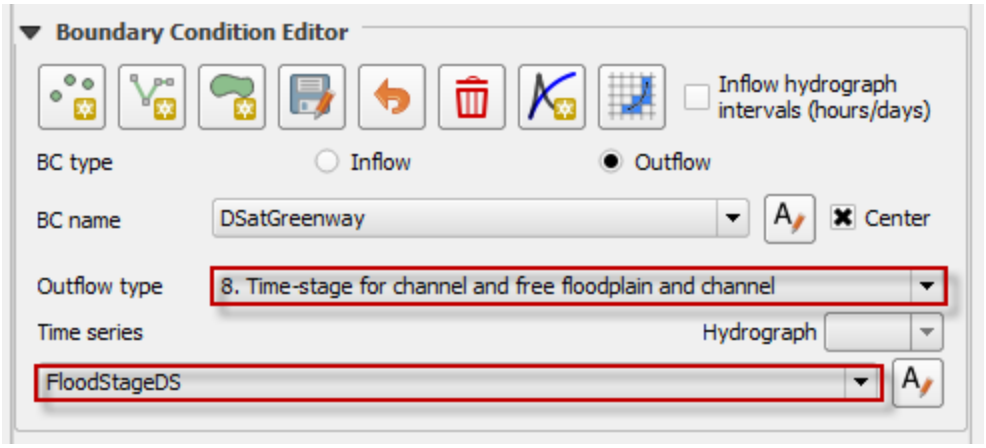
Use this option to set the stage of a downstream boundary. This node will allow water to collect on the boundary until it can exceed the stage along the boundary. It can be used for two purposes.

1. Anytime there is a control on the boundary that releases water at a known stage.
2. Set the elevation for matching the water surface elevation of an existing FEMA map.



Time-Stage for Channel and Free Floodplain and Channel

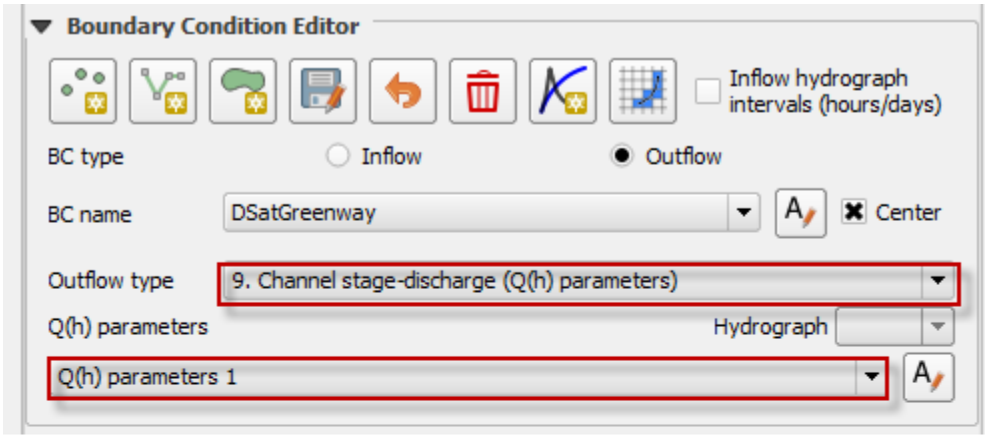
This option is the same as option 7 with the condition that the stage – time table is assigned to the channel instead of the floodplain.

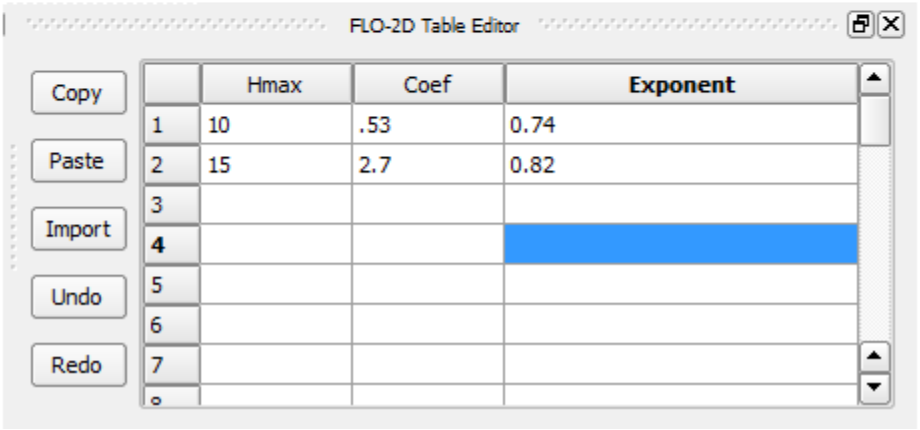


Channel Stage-Discharge Parameters

This outflow option defines the discharge from a channel based on the stage using rating curve. Several rating curves can be assigned for multiple limiting depths. This system is used when there is a control or a gage at the channel with a known stage-discharge relationship.

1. Select the stage-discharge node at the end of a channel segment.
2. Click *Save* to load save the feature to the layer and activate the editor.
3. Name the boundary and set the boundary conditions.
4. Name and fill the Q(h) parameters table.

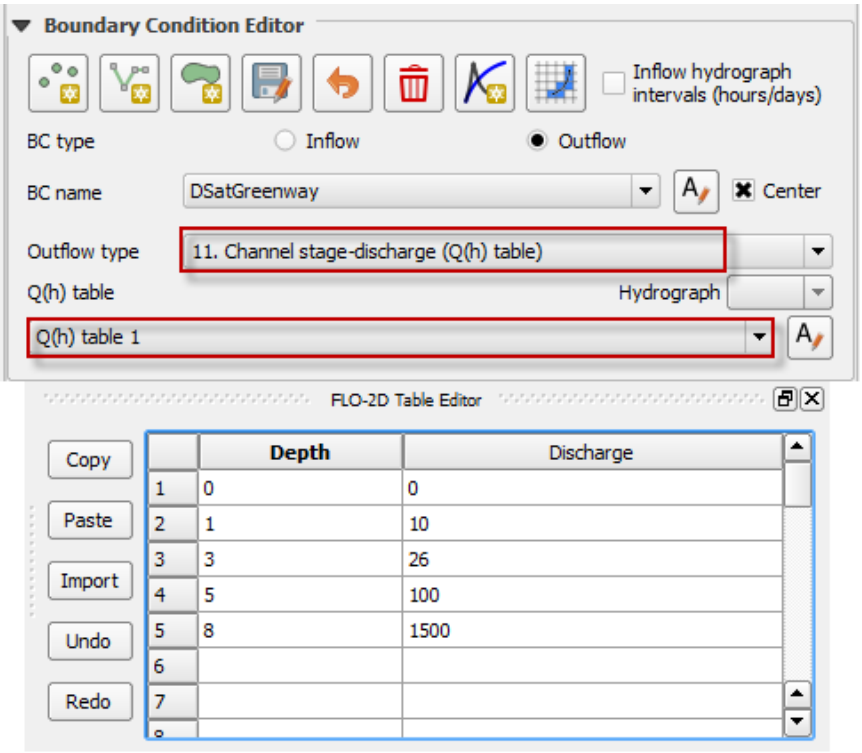




Channel Stage-Discharge (Q(h) table)

The final outflow option is used to define the downstream boundary with a stage-discharge table.

1. Select the stage-discharge node at the channel terminus.
2. Click *Save* to load save the feature to the layer and activate the editor.
3. Name the boundary and set the boundary conditions.
4. Name and fill the Q(h) table.



Troubleshooting

1. The most common problems with creating outflow.dat data is caused by creating conflicts by putting other components in the outflow grid elements.
2. The schematic layers and tables will reset each time the Schematize tool is used. This could cause overwriting of imported data. Convert the Boundary Conditions to User Layers for projects that are imported into QGIS before performing the schematization process.

3. If the data does not export correctly, check the tables. The tables can be edited directly or can be copied into an OUTFLOW.DAT file.
4. Saving and restarting might resolve some issues with the GeoPackage but check the layers attributes prior to restarting QGIS.

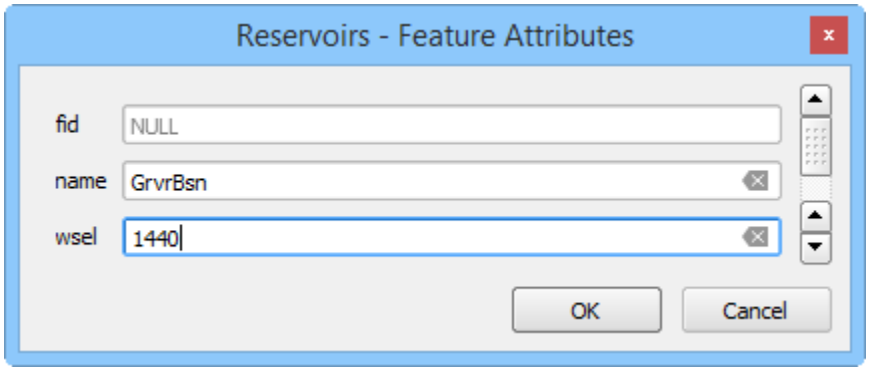
Reservoir Node

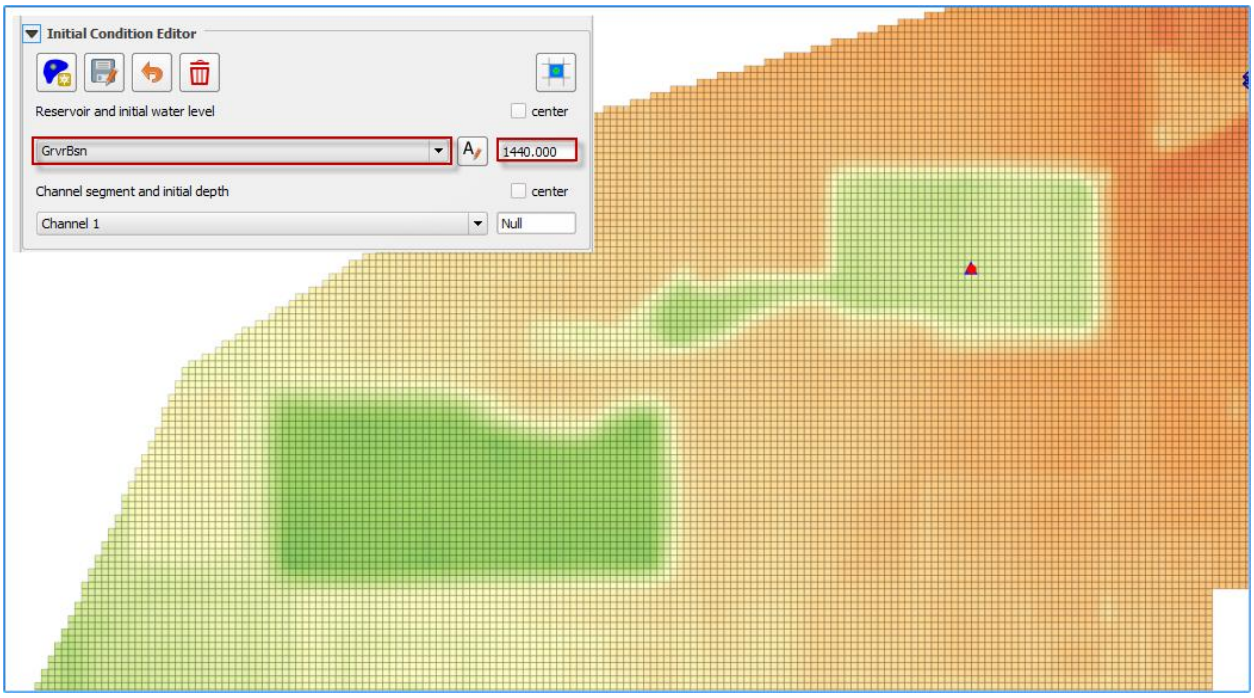
The reservoir node is used to assign the water surface elevation for a reservoir or pond by specifying a single reservoir node elevation. The FLO-2D model will identify every grid element as a reservoir element whose bed elevation is less than the prescribed water surface elevation at runtime as assign the starting reservoir flow depth. The reservoir data is written to the INFLOW.DAT file.

1. Click the *Reservoir* button.
2. Click a cell within the reservoir.

3. Set the name and elevation.
4. Click save to activate the form.

5. Click *Schematize* to save the data to the schematic layers.



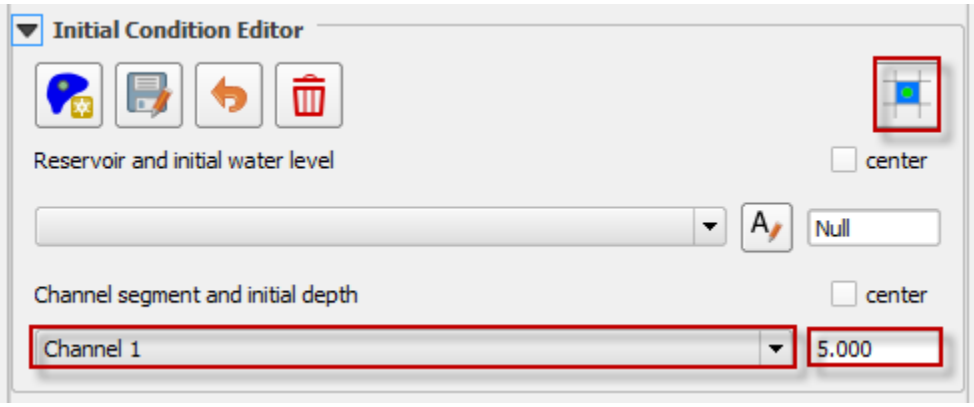


Troubleshooting

The most common issue encountered with this tool is low reservoir bank elevation surrounding the reservoir. If the confining reservoir bank element is lower than the reservoir elevation, the water will spill out of the cell. Check the reservoir containment by running the FLO-2D simulation for short duration of 0.01 hours. The reservoir will be filled and display any leaks in the maximum depth output files.

Channel Segment Initial Depth

Use this option to set an initial depth in any channel segment. The water depth will be assigned to every channel cross section within the segment at runtime. The initial condition will be written to the CHANNEL.DAT file.



1. Select the channel segment.
2. Set the initial depth.
3. Click *Schematize* to save the data to the schematic layers.

Troubleshooting

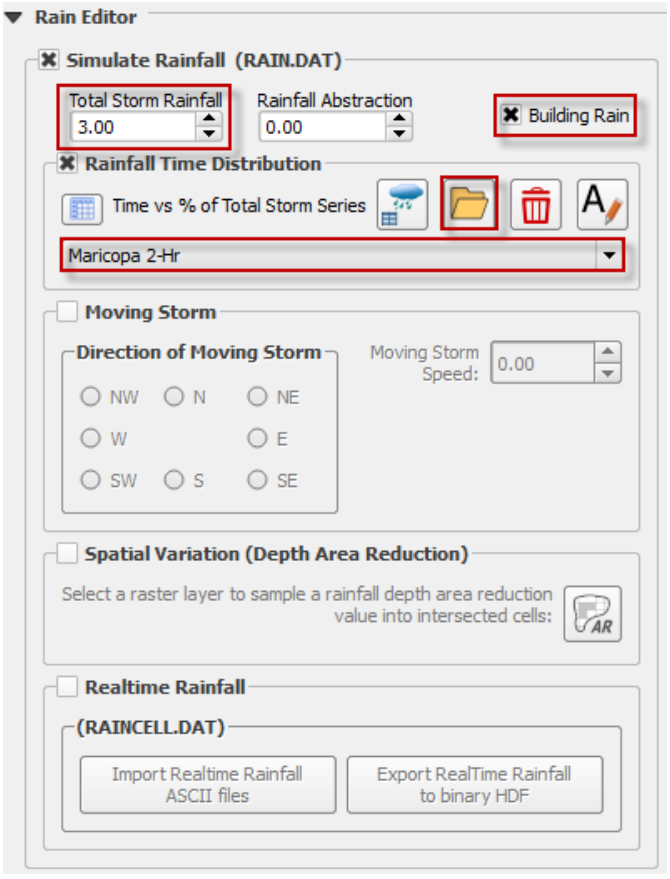
The simplicity of this tool makes it simple to adjust. It is not necessary to use the tool here because the variable can also be set in the Schematized Channels Editor.

Rainfall

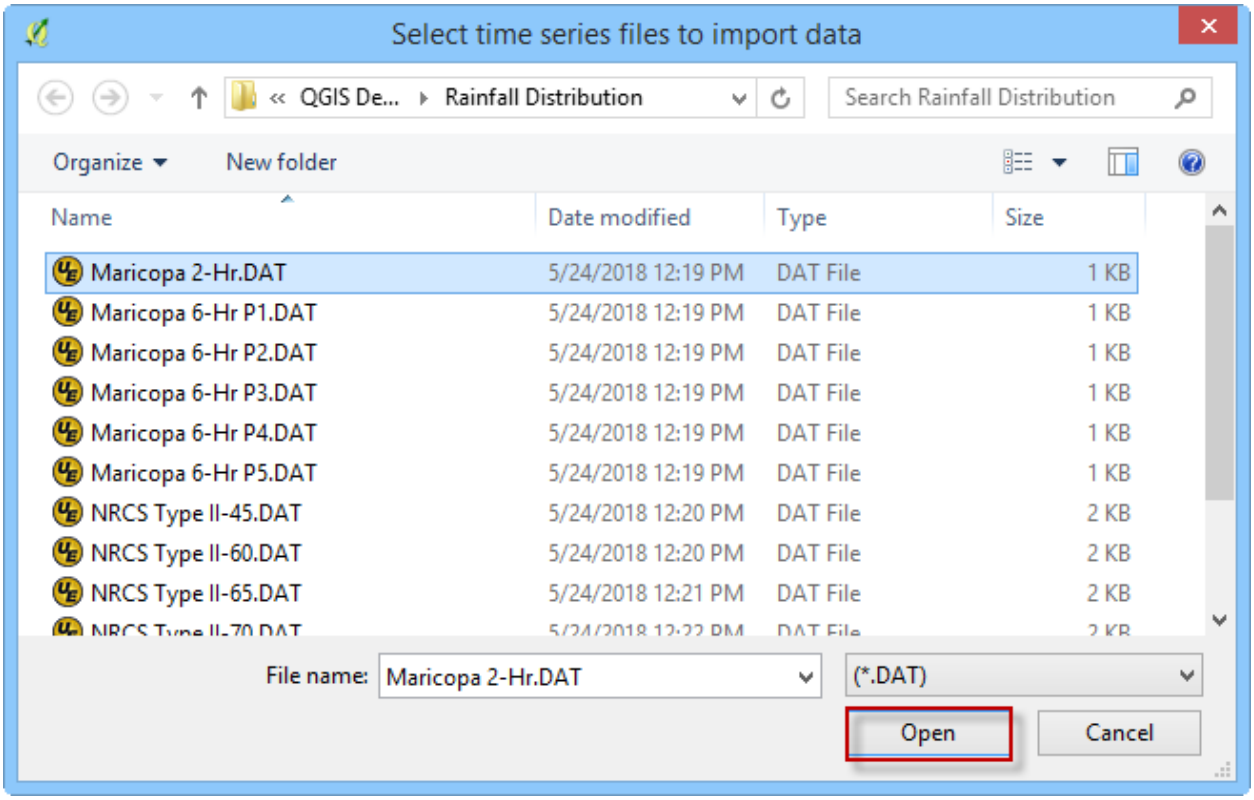
The rainfall editor is used to set up the RAINFALL.DAT file and the RAINCELL.DAT file. This section will describe how to set up uniform rainfall, spatially variable depth area reduction factors, and spatially and time variable rainfall.

Uniform Rainfall

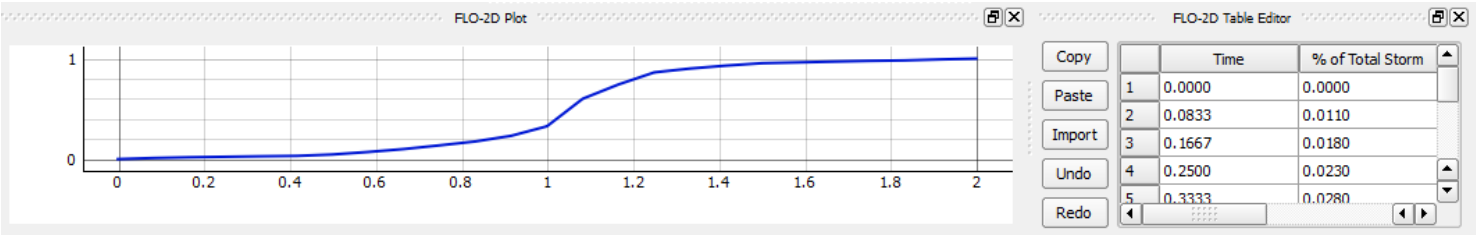
Uniform rainfall requires the total rain in inches or millimeters and a rainfall distribution. In this example, the total rainfall is 3 inches. To find Total Storm Rainfall, zoom to Rainfall raster with Computational Domain layer, check Identify Features icon, clicks within the Computational Domain to find highest derived value. The Building Rain switch is selected so that rain on totally blocked ARF cells will allow rain to run from the blocked areas. The initial abstraction is default of zero inches. The rainfall distribution is imported from a data file.



Select the rainfall distribution using the Folder menu. Import the rainfall distribution file from a set of predefined files. The files are installed in the FLO-2D Documentation folder. C:\users\public\public documents\FLO-2D Pro Documentation\flo_help\Rainfall Distributions



The rainfall data is imported into the FLO-2D Table Editor.



11. Load the currently selected time series into the editor table.

12. Add a new rainfall time series to the project.

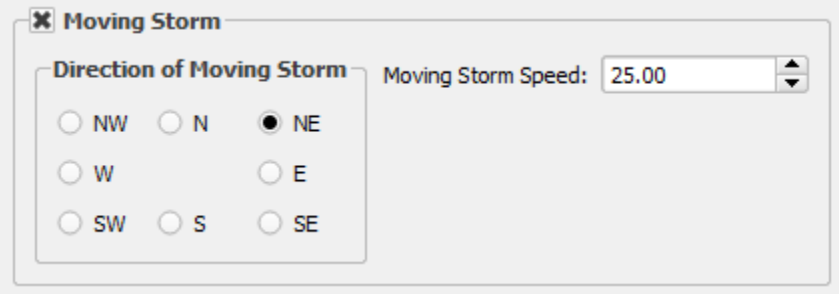
13. Open a time series data file.

14. Delete a time series table.

15. Rename a time series data set.

Moving Storm

To simulate a moving storm cell, select a storm direction and speed. This data must be assigned along with isohyetal contours that represent storm intensity as depth are reductions contours.

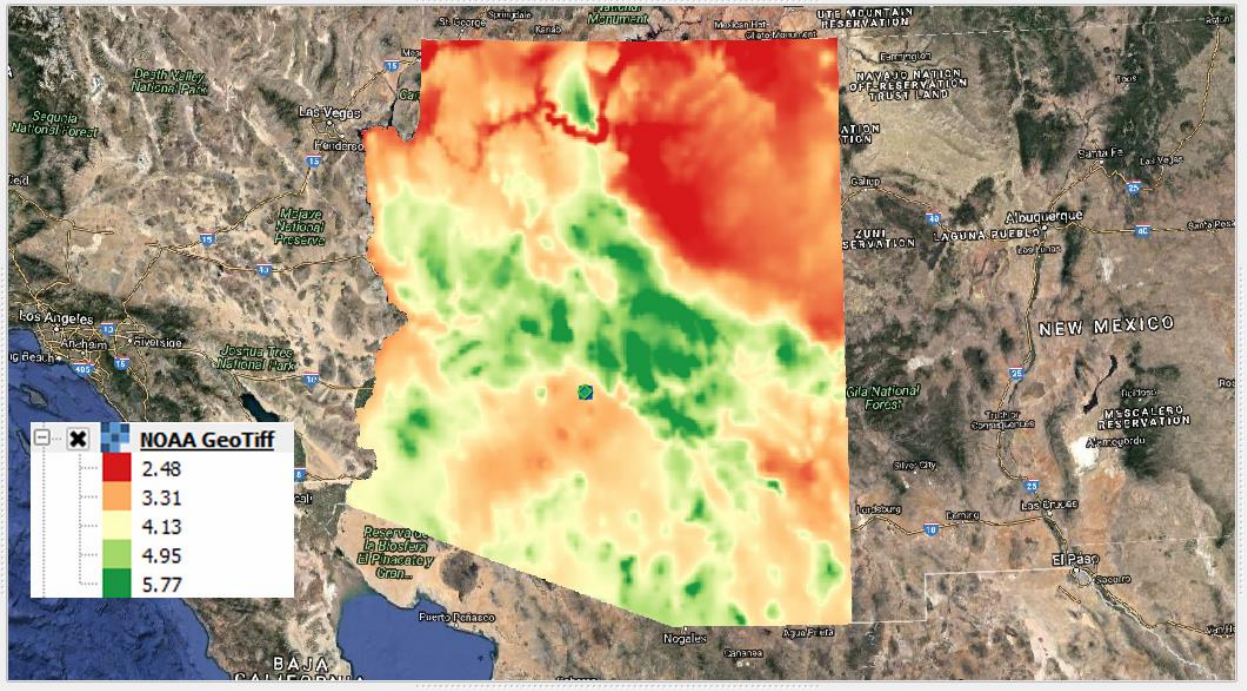


Spatially Variable Rainfall NOAA Atlas 14

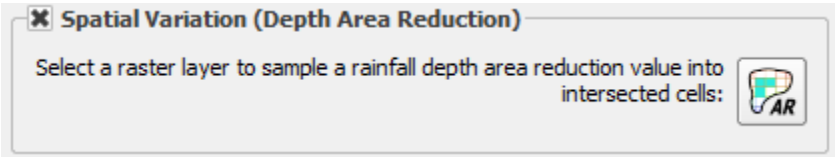
Spatially variable rainfall data can be set up using depth area reduction factors in the RAINFALL.DAT file. The data format is inches (millimeters) at a point over total rainfall in inches (millimeters). The point is the centroid of each cell. The total rainfall area is the computational domain.

The rainfall data is obtained from precipitation rasters. In the United States the rasters are calculated from isohyetal contours located the NOAA precipitation

prediction website (NOAA, 2017). One map is required for each precipitation frequency. The following example is a map of the 100yr 6hr storm prediction for Arizona. The legend represents total rainfall in inches.

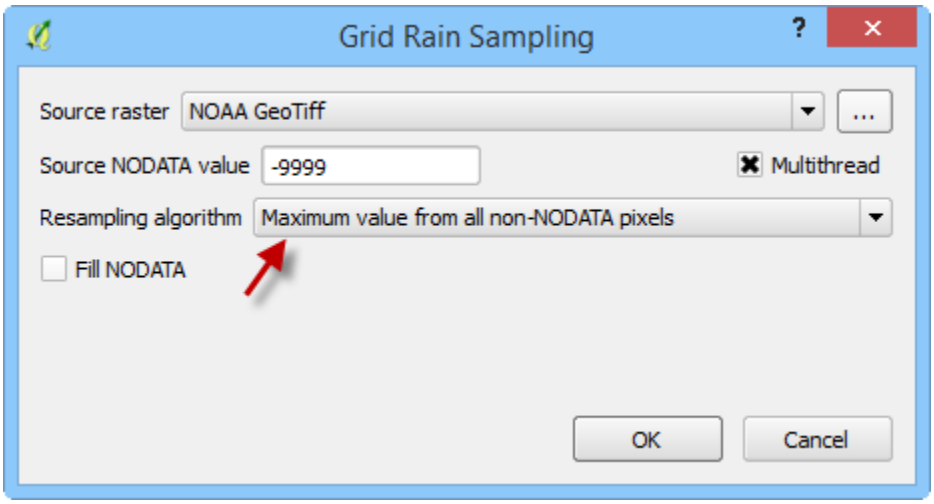


To perform the calculation, use the Area Reduction calculator.



1. Click the *Area Reduction* button.
2. Fill the form and click OK.

The raster pixels are typically 1000 by 1000 ft or larger. It is not necessary to average the data. Select the maximum value to set the cell value.



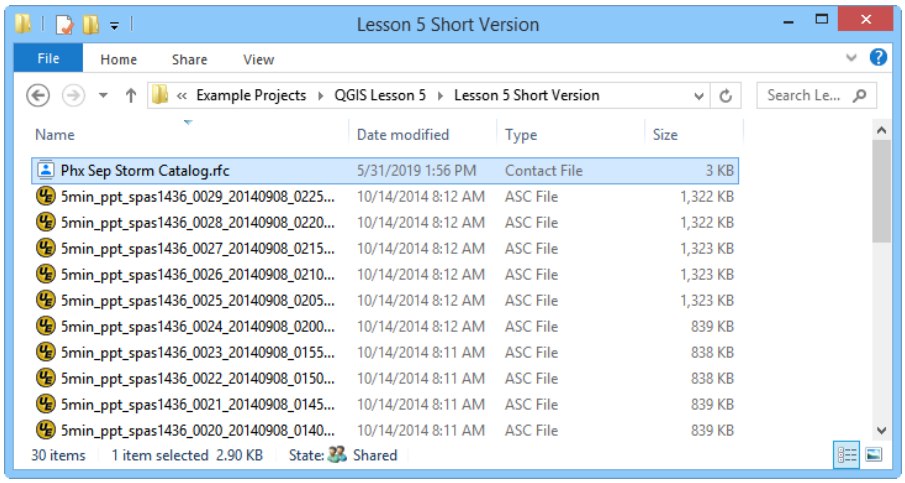
Realtime Rainfall NEXRAD

Realtime rainfall data is computed from NEXRAD *.ASC grid files. In the United States, the files are derived from calibrated radar reflectivity maps from National Oceanic and Atmospheric Administration (NOAA) and rainfall gages in local areas. The data calibration is typically conducted by local agencies or consultants. Datasets can be downloaded directly from NOAA's NEXRAD Archive (NOAA, 2017). NEXRAD rainfall data and rainfall gage data are available from NOAA.

The realtime rainfall calculator imports the *.ASC files, interpolates them to the grid and creates the rainfall data in two formats. The formats are RAINCELL.DAT and RAINCELL.HD5.

Rainfall Catalog

The project directory must contain all the rainfall *.asc files and a catalog file *.rtc. Both filetypes are text files. The *.asc files are rainfall coverage for the project area at a specified time interval. In this case the time interval is 5 minutes.



The rainfall catalog is a text file that contains specific information about the lay-out and timing of the data. The first line of data is a control line that sets up the timing of the storm and the interval.

Line 1: 9/9/2013 04:05 9/9/2013 23:55 5 239

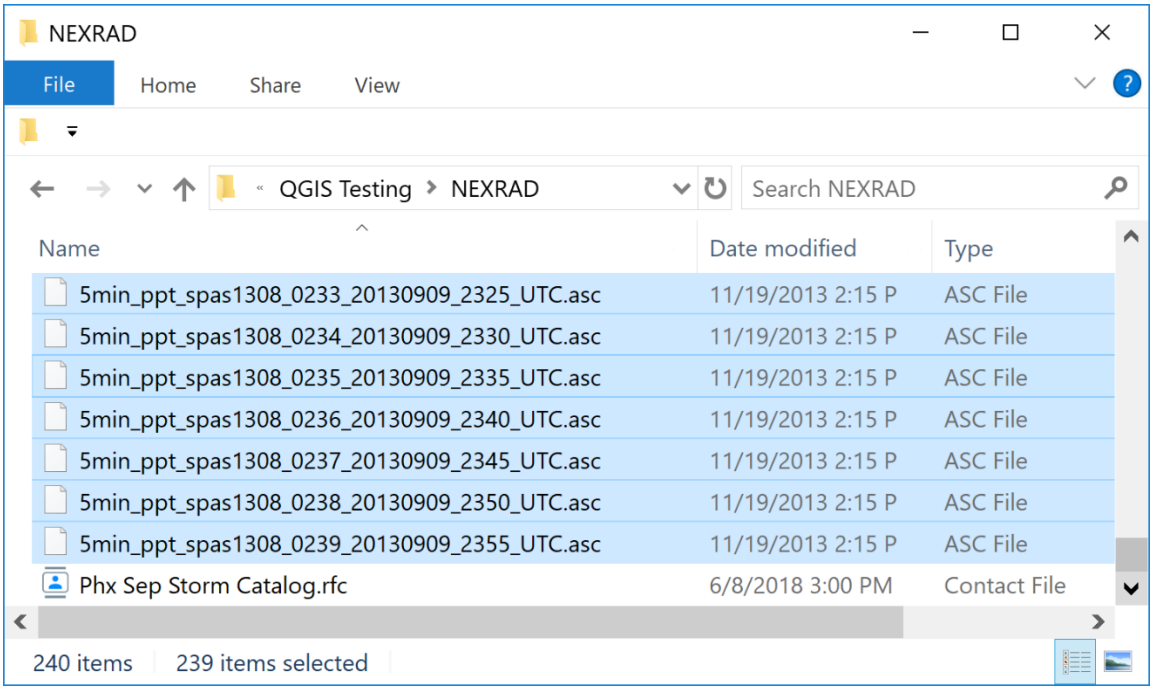
Line 1: Start Time, End Time, Time Interval, *.asc File Count

*****Important Note*****

Line 2 of the *.rfc isn't necessary. The Rainfall calculator will use all *.ASC files in the project directory. Make sure they are named in ascending order by time.

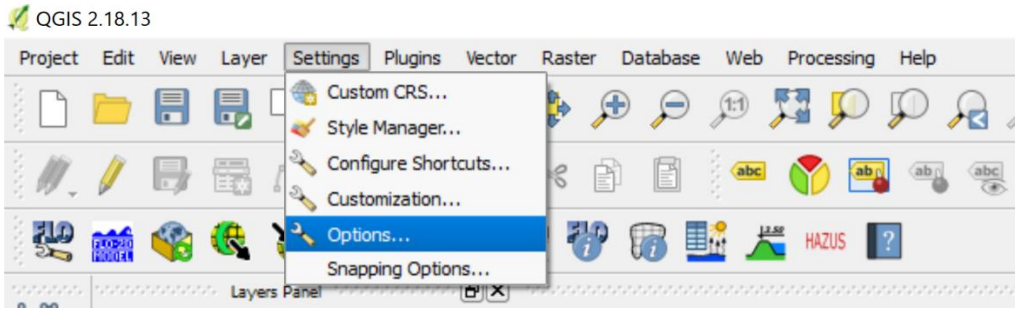
Time = 0 is not necessary. The engine will start the simulation at time and rainfall = zero. The rainfall data will interpolate from time = 0 rainfall = 0 to the first interval.

*****Important Note*****

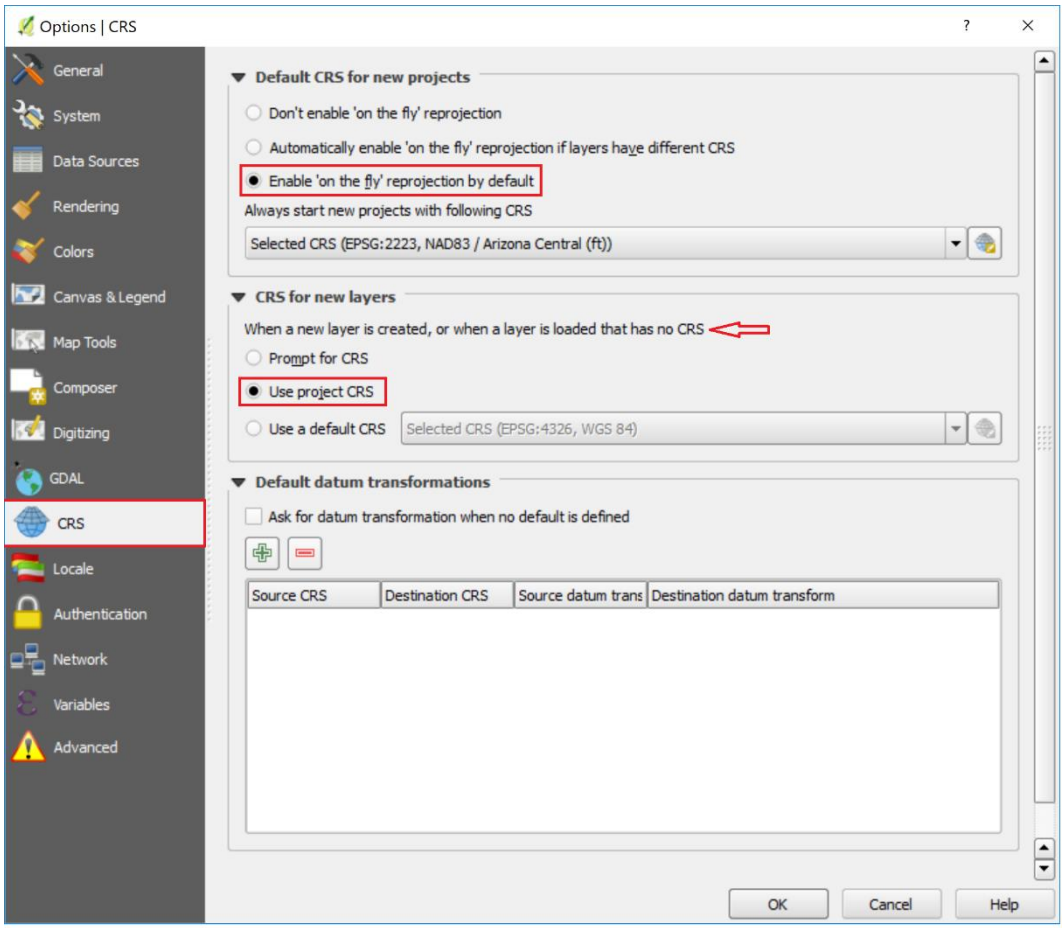


Setup Import Options

The rainfall data does not have a specific coordinate reference system (CRS). Set the import options for QGIS to automatically assign all imported data with no knows CRS to the Project CRS. Click Settings Options

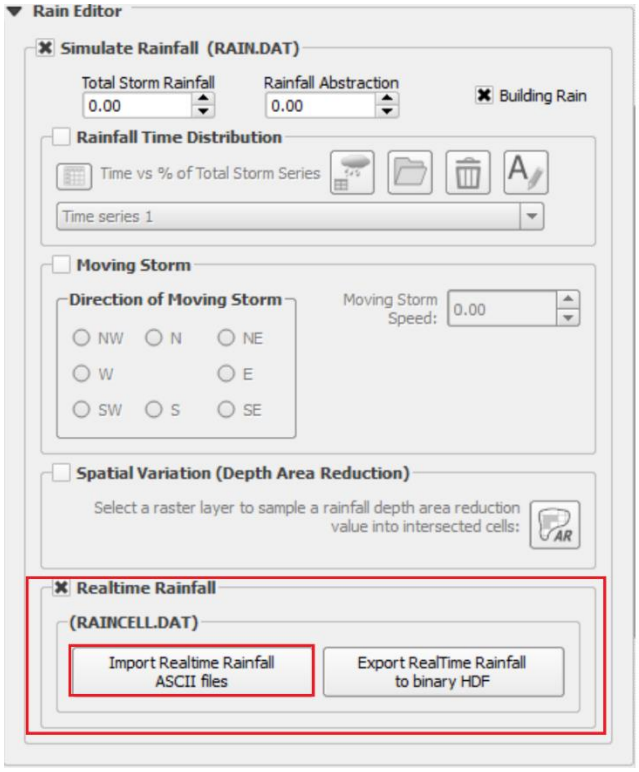


Select the following options and click OK.

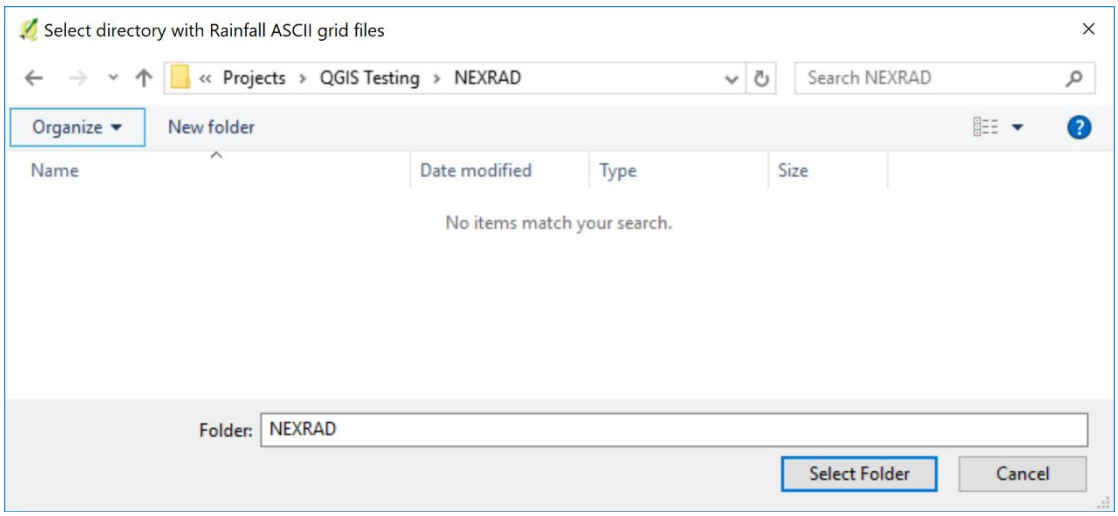


Import Realtime Rainfall

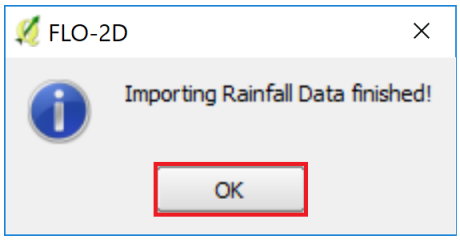
To interpolate realtime rainfall data, activate the Realtime Rainfall Editor. Click the Import Reatime Rainfall ASCII files.



The plugin requests the directory where the NEXRAD data is stored. Navigate to the correct directory and click Select Folder.

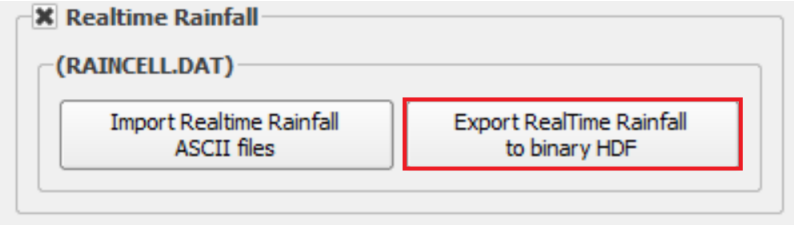


Click OK once the process is complete.

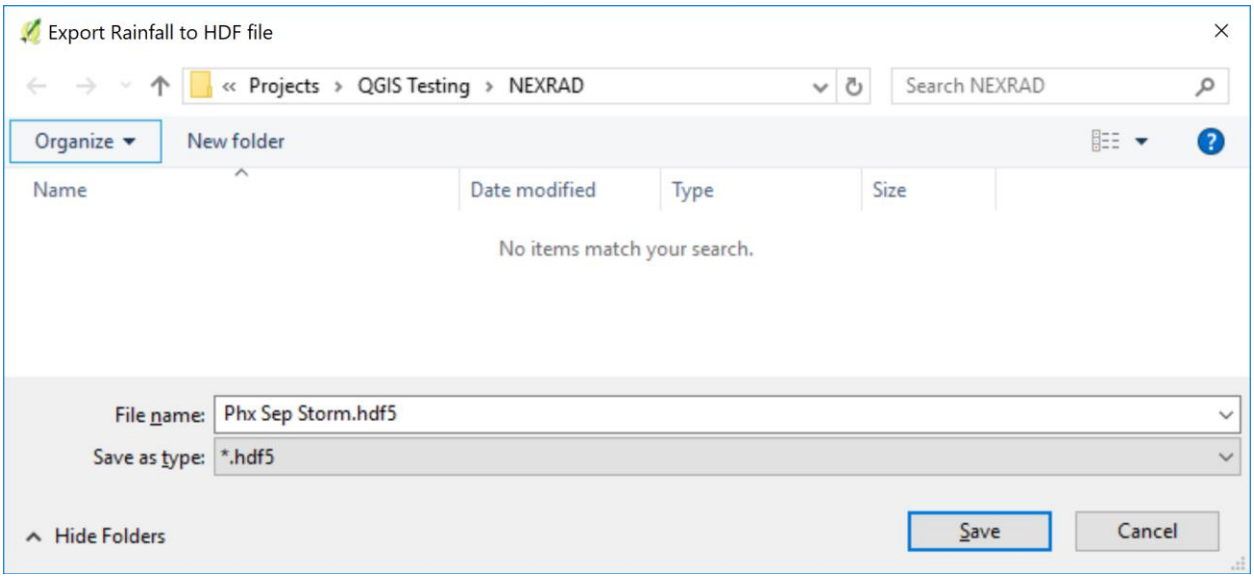


Export the rainfall data.

Click Export the Rainfall Data



Name the storm and click Save.

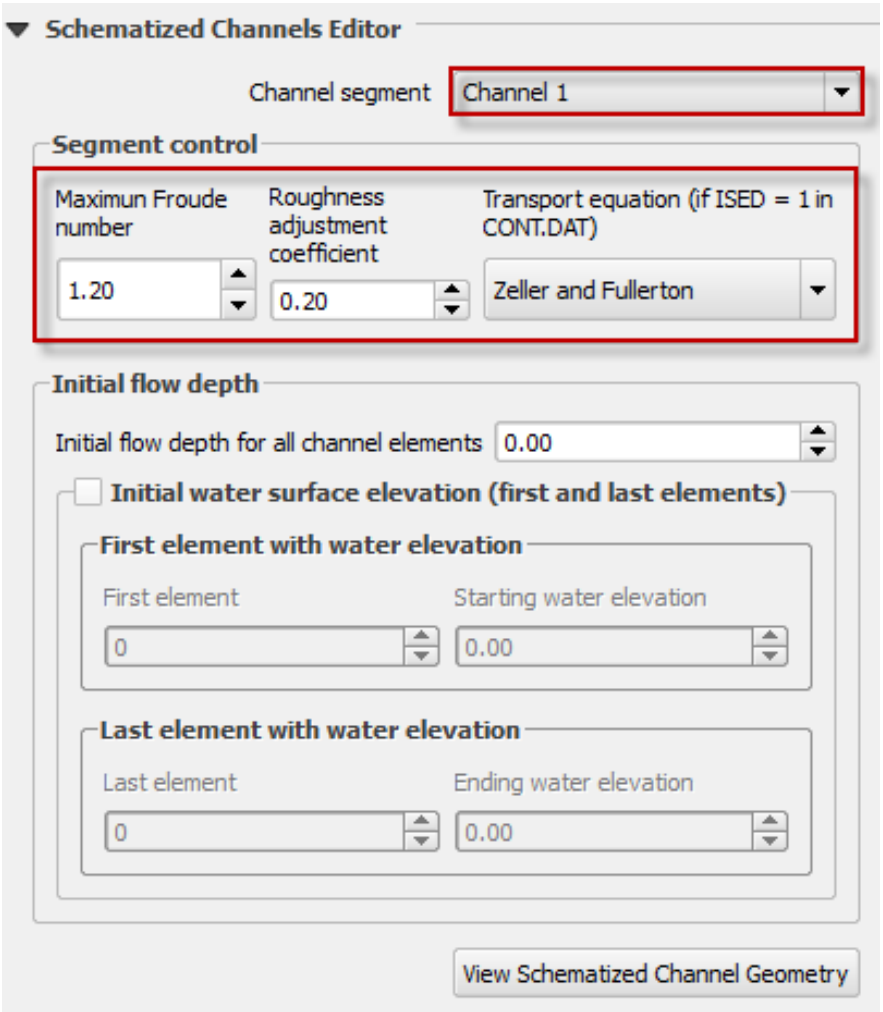


Schematized Channel Editor

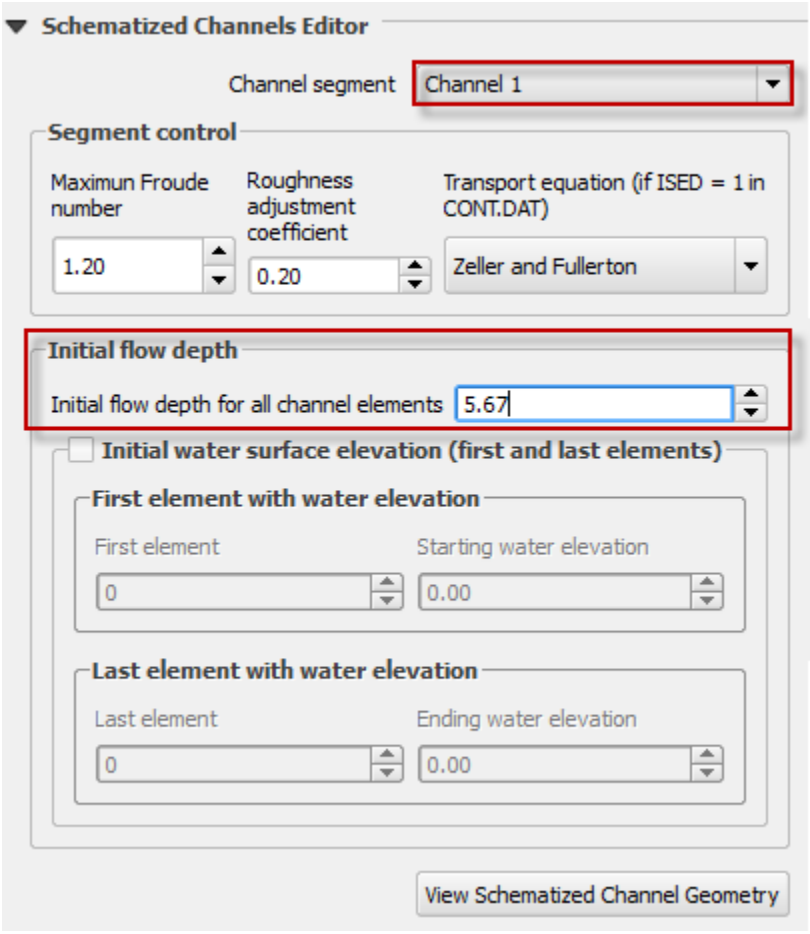
This editor widget is used to edit the control data and tabular data of the schematized channels. It works on any schematized channel data including, imported data, imported RAS data and digitized data. This data is written to the CHAN.DAT and CHANBANK.DAT files.

Segment Control

Select a channel segment to load the control data. The data is different for each channel segment. Review the Channel Modeling Guidelines (FLO-2D 2018) to assign the parameters. The sediment transport data will not be used unless the channel option is used in the SED.DAT file.

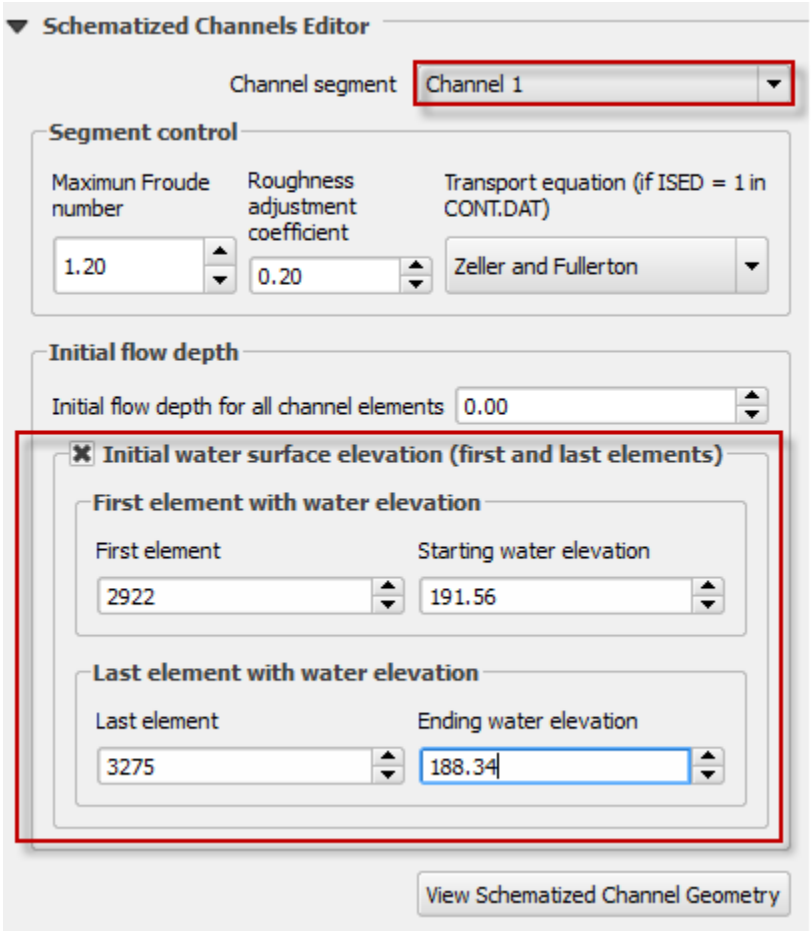


The global initial depth value can be set in the channel segment control data. This is a single value that is assigned to every cell in the channel segment when the model starts.



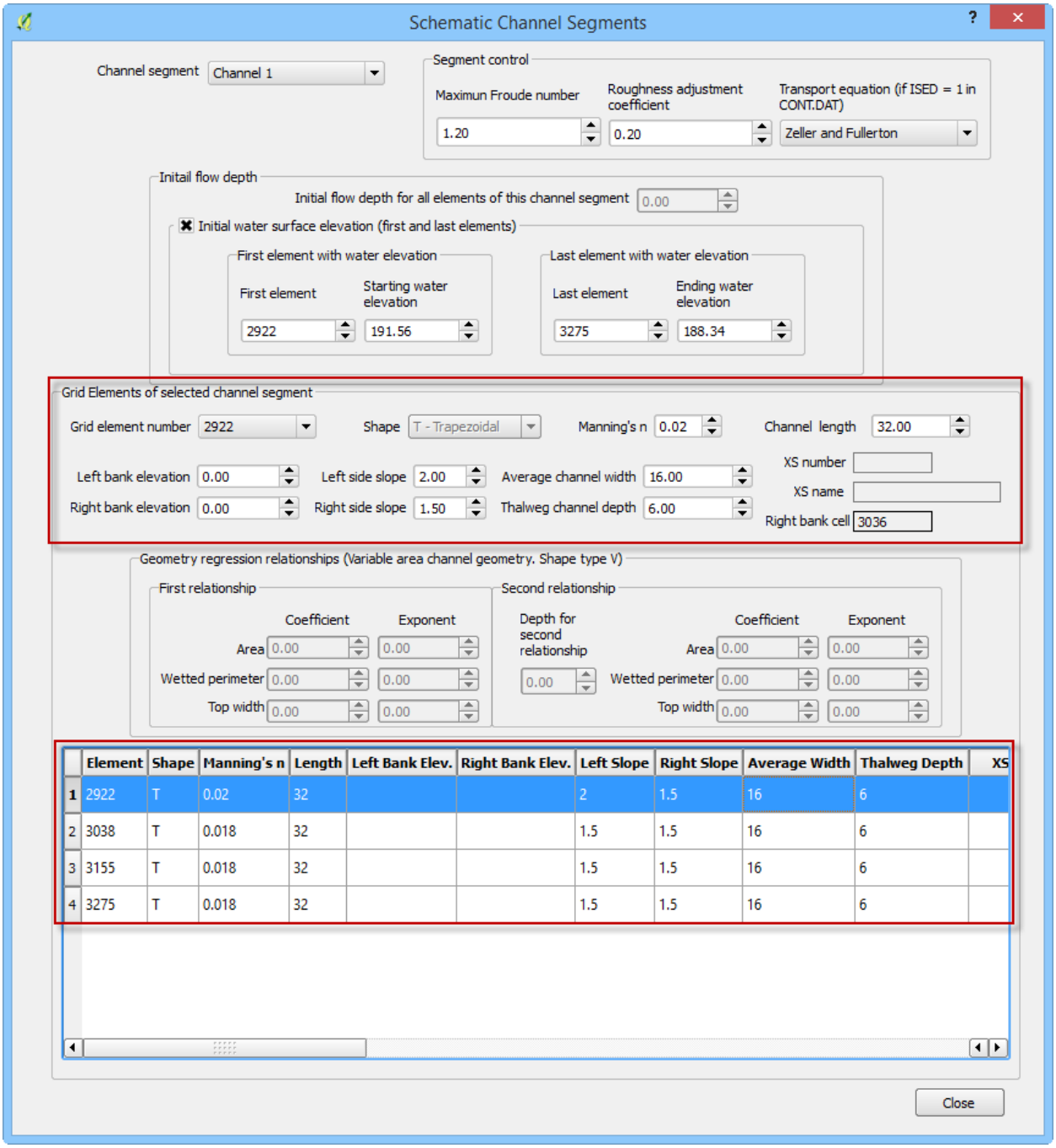
Initial Water Surface Elevation

The initial water surface elevation is used to set the variable initial conditions. The values are assigned to the first and last element in a segment and interpolated to each cell in the channel at runtime.



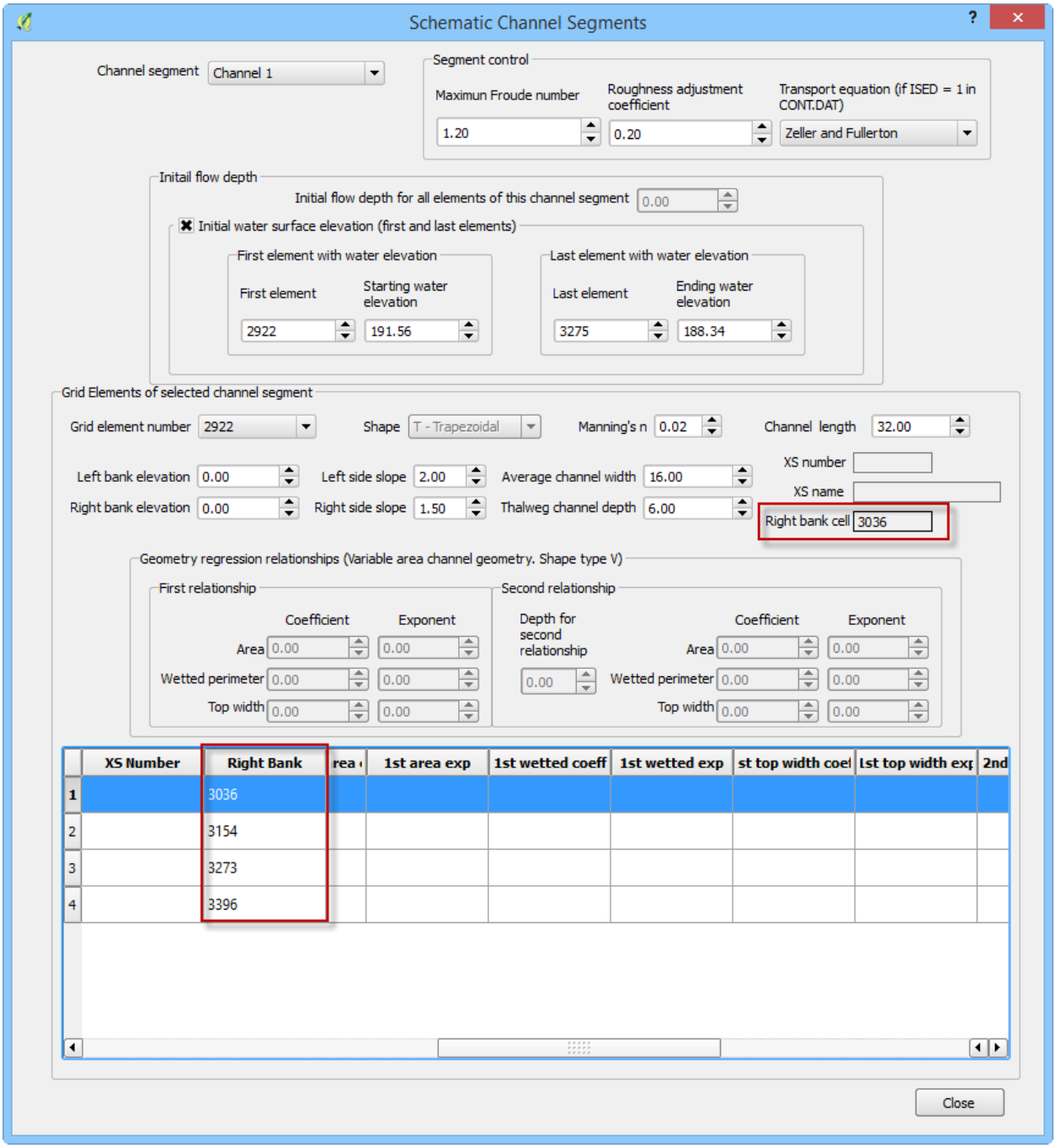
Channel Geometry

The tabular channel data is edited in the Schematized Channel Geometry dialog box. This data can be edited by loading any grid element into the editor box or by editing the table directly. Copy paste options are active in the table editor.



Right Bank

A right bank editor is available in the Schematic Chanel Segments dialog box. Edit the columns directly and click close to apply.



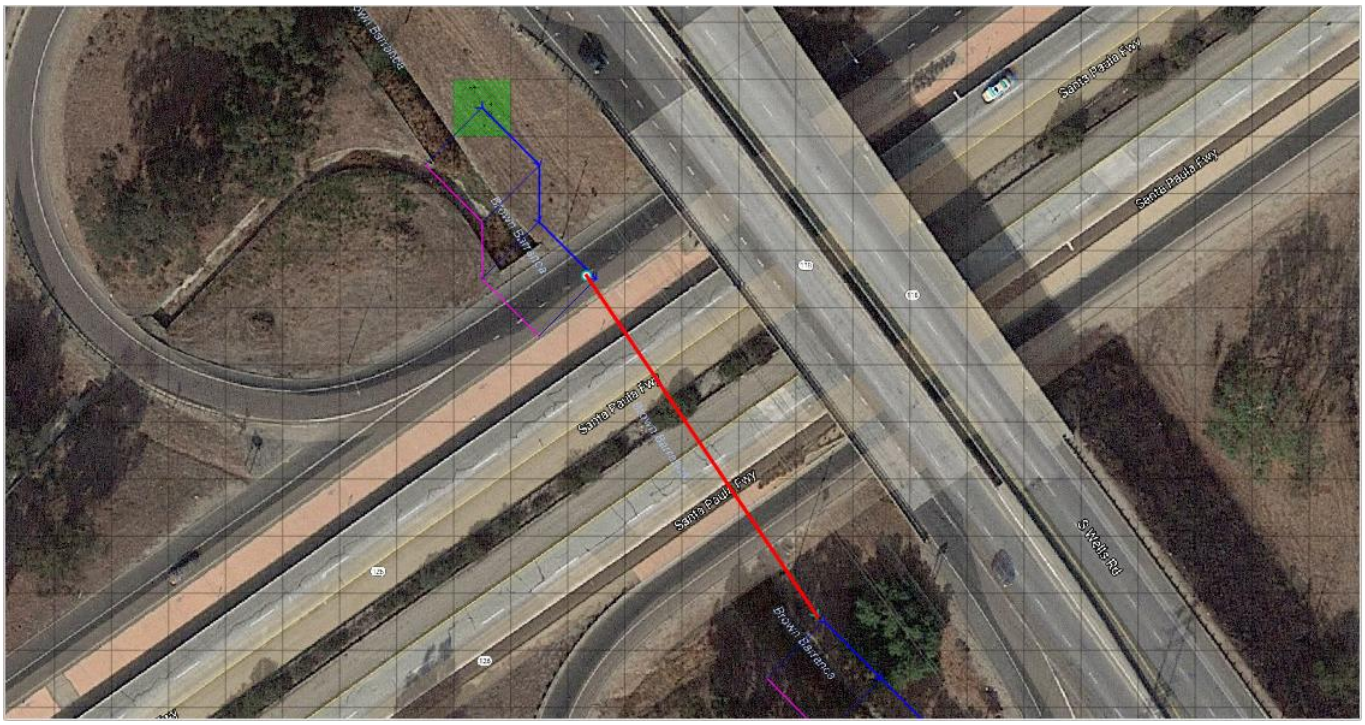
Hydraulic Structure Editor

The hydraulic structures editor is used to set up the data for the HYSTRUCT.DAT file. This section will describe how to set up several different hydraulic structures.

Channel to Channel Structure

This structure will simulate discharge through a box culvert. This example has a box culvert that is longer than the grid element. The channel segments are split up to allow for the width of the roadway.

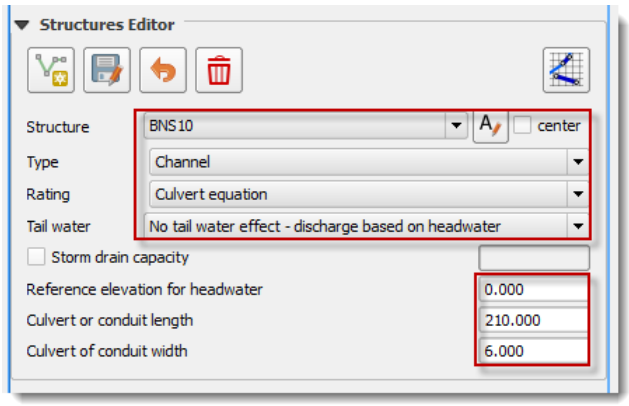
1. Click the Add Structure button.
2. Digitize a culvert by clicking on the two blue left bank elements that represent the beginning and end of the hydraulic structure.

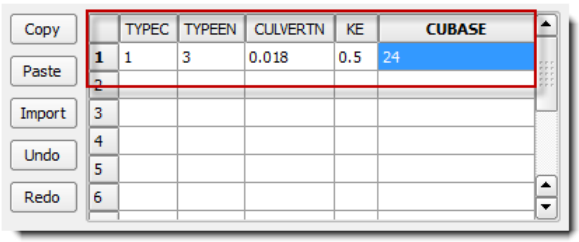


3. Click Save.

4. Fill in the data on the Structure Editor and the Table Editor. Here are the variables for this structure.

   a. Name BSN10
   b. Channel to Channel
   c. Discharge setting is Culvert Equation
   d. Tailwater condition is none.
   e. Head Reference elevation is not needed 0.00 will default to the channel inverts.
   f. Conduit length is 210 ft.
   g. Conduit width is a little confusing. For a Culvert Equitation with circular shape, it will be diameter. For a box culvert it will be culvert height.
   h. TYPEC is 1 for Box.
   i. TYPEEN is 3 entrance type.
   j. CULVERTN is 0.018 for the n-value.
   k. CUBASE is 24 ft. This culvert is a box culvert with 3 8 ft boxes.





5. Click Schematize to write the data to the schematic layers.

Floodplain to Floodplain Structure

This example will model a culvert system between two floodplain nodes. The invert elevation of the inlet node is set by the grid element elevation.

1. Click the Add Structure button.
2. Digitize a culvert by clicking on two cells closest to the inlet and outlet on the map. Use the elevation values to make sure that the invert elevations are correct.

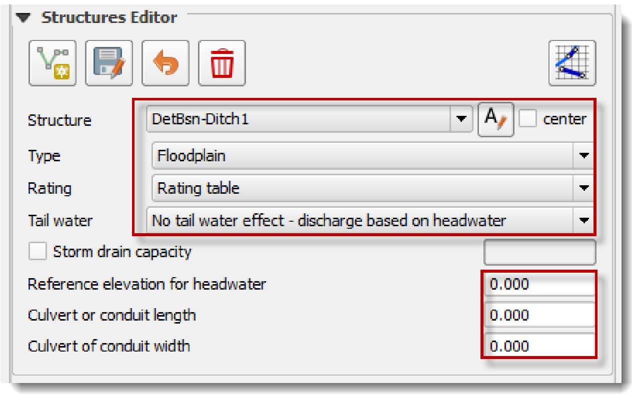
3. Click Save.

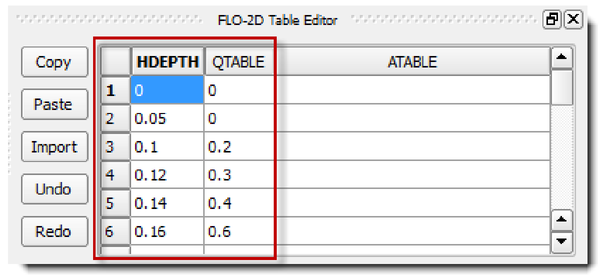
4. Fill in the data on the Structure Editor and the Table Editor. Here are the variables for this structure.

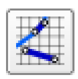
    a. Name DetBasin-Ditch1
    b. Floodplain to Floodplain
    c. Discharge setting is Rating Table
    d. Tailwater condition is none.
    e. Head Reference elevation is not needed 0.00 will default to the grid element elevation.

f. Conduit length, width, and area are zero. (not needed for this setup)

g. Fill in Rating Table data. It can be copied and pasted from most other sources.





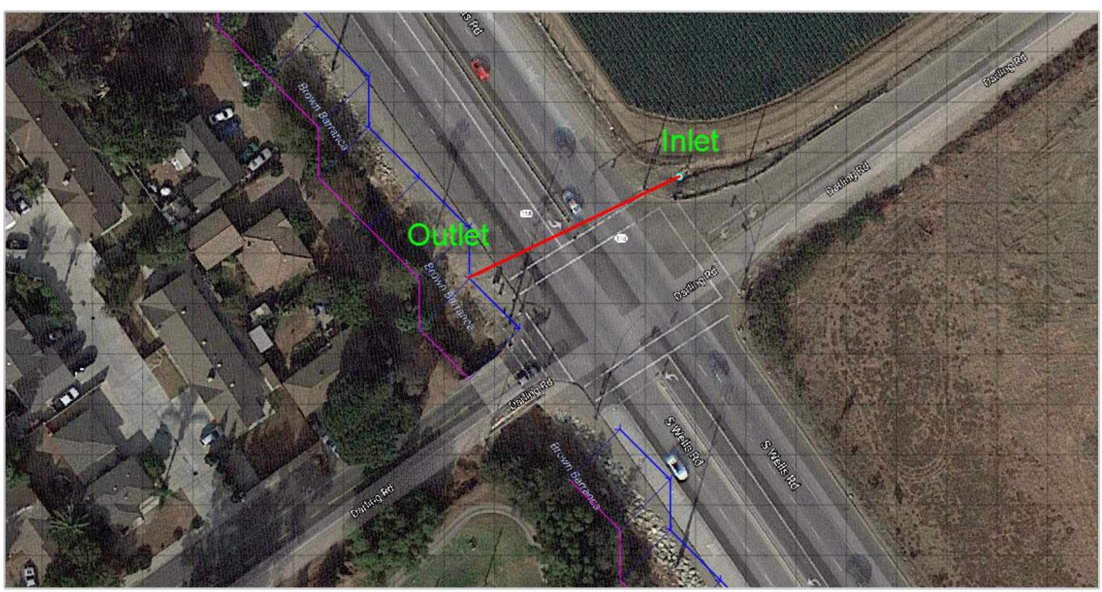


5. Click Schematize to write the data to the schematic layers.

Floodplain to Channel Structure

1. Click the Add Structure button.

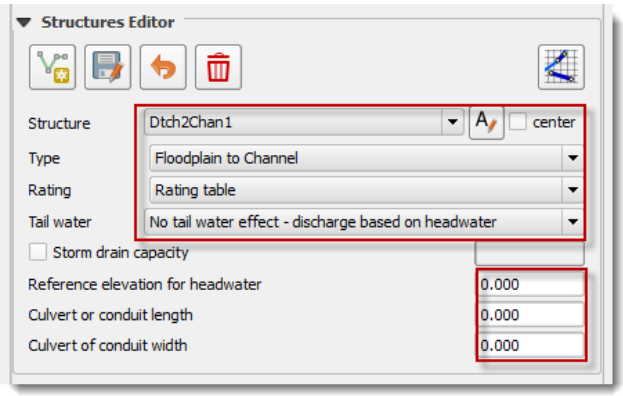
2. Digitize a culvert by clicking on two cells closest to the inlet and outlet on the map. The outlet should be assigned to a blue channel node. Use the elevation values to make sure that the invert elevations are correct.

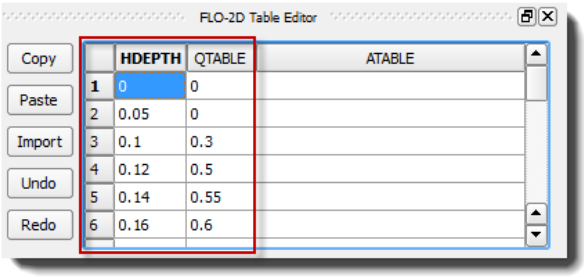


3. Click Save.

4. Fill in the data on the Structure Editor and the Table Editor. Here are the variables for this structure.

   a. Name DitchtoChan1
   b. Floodplain to Channel
   c. Discharge setting is Rating Table
   d. Tailwater condition is none.
   e. Head Reference elevation is not needed 0.00 will default to the grid element elevation.
   f. Conduit length is 0 ft. (not needed for this setup)
   g. Conduit is 0 (not needed for this setup)
   h. Fill in Rating Table data. It can be copied and pasted from most other sources.





5. Click Schematize to write the data to the schematic layers.

Street Editor

The street editor is used to define street data on low resolution urban areas. This data is written to the STREET.DAT file. It is used when the grid element size is larger than the width of curb and gutter streets. This tool works best for streets that collect and distribute water over the grid system. The street should be able to convey water like a small channel.

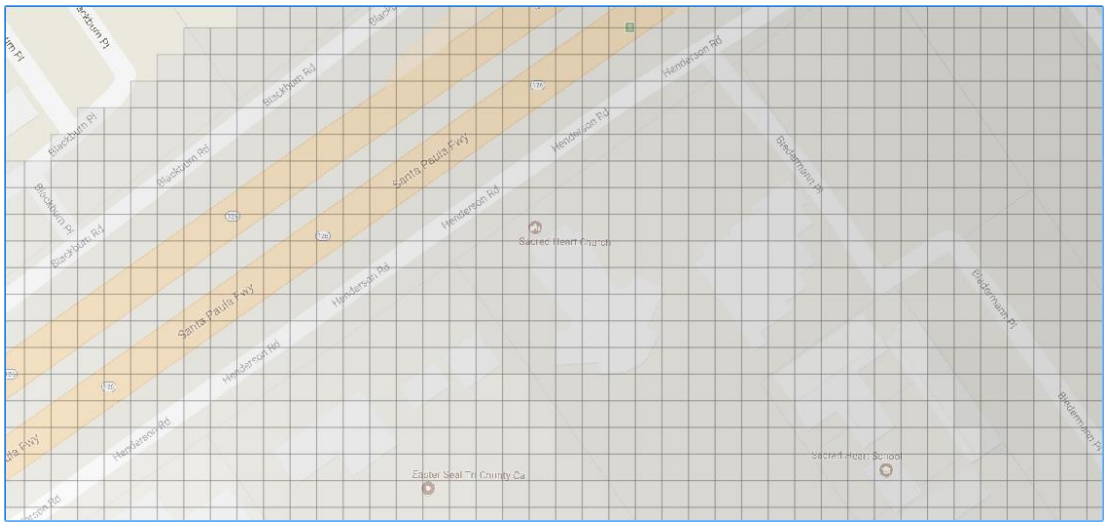
1Source: iStock

Build a street

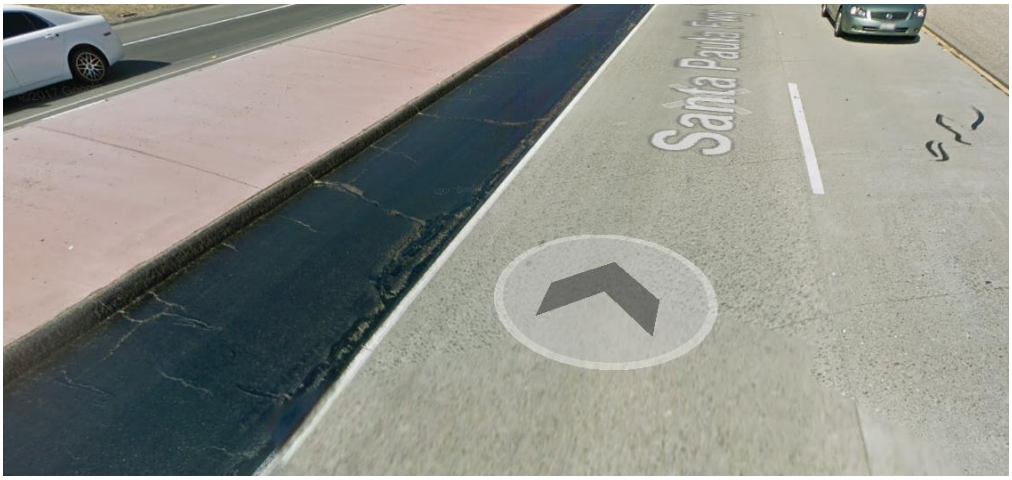
Identify a street that meets the criteria

Setup a FLO-2D project map with easy to identify roads, an elevation raster and a grid layer view.

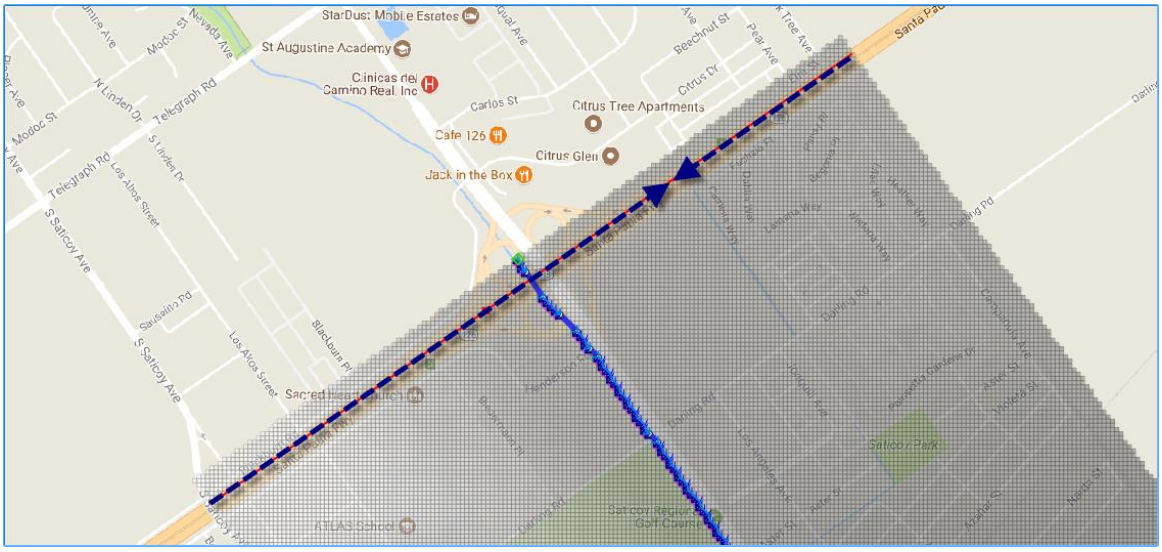
1. Identify a street that is larger than the grid element size. The Santa Paula freeway lanes meet the grid element size criteria.

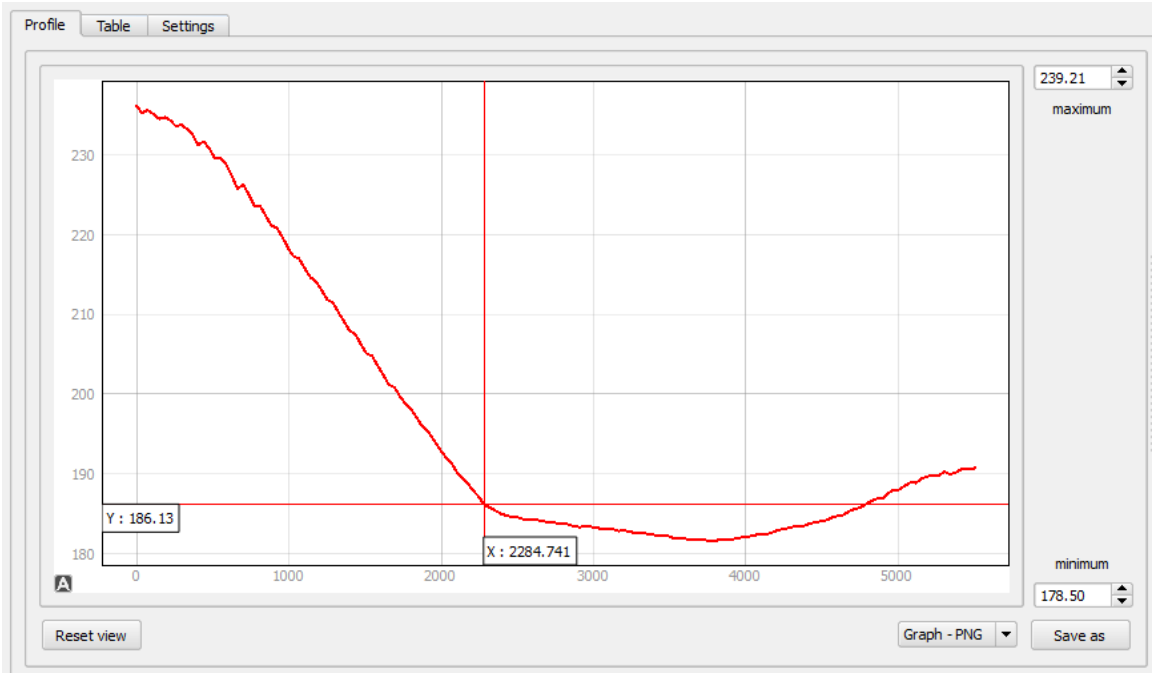


2. Determine if the street can convey water. The Santa Paula freeway has a well-defined curb and gutter on the outside lane.



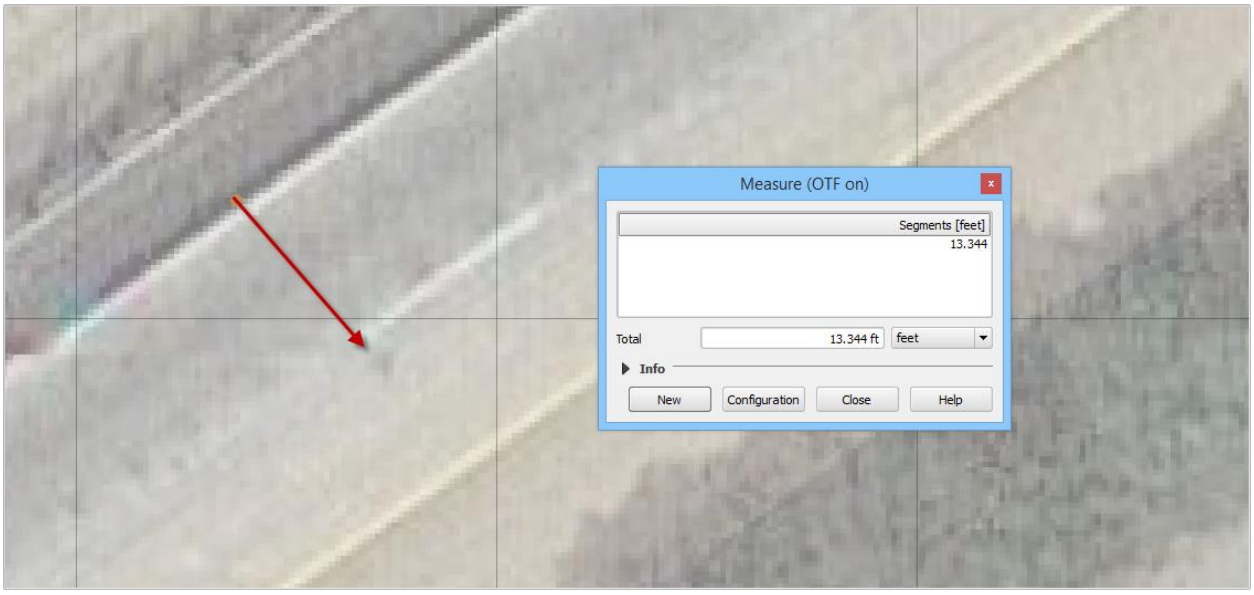
3. Determine the flow direction. The Santa Paula freeway has a well-defined slope leading up to the overpass with no storm drain features to divert water off the road. This information was collected from creating a profile of the raster along the street and reviewing the Google Maps Street View tool via a web browser. The water will flow toward the point where the two arrows meet. Then it will overtop the channel onto the floodplain.



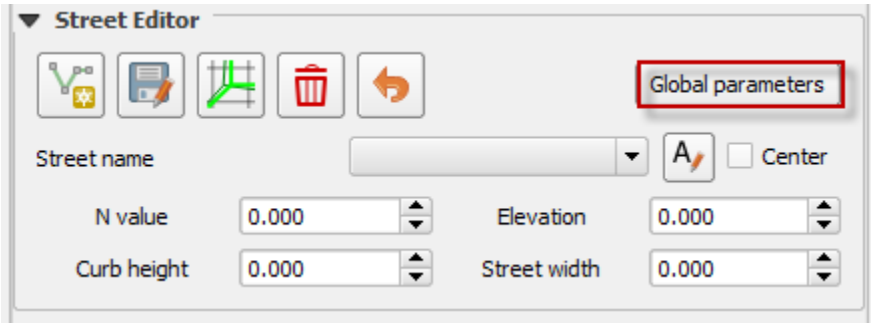


Assign Global street values.

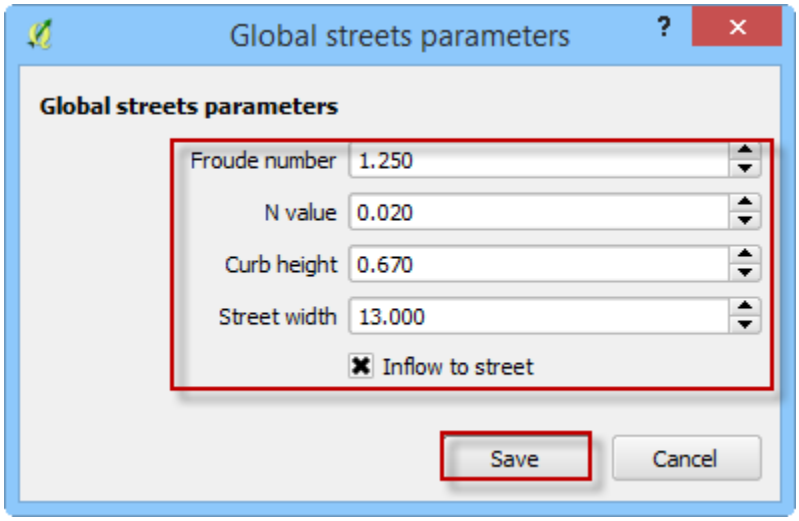
1. Define the street width, curb height and n-value. This street is 13 ft wide from the curb to the crown. There is only a gutter on one side so the other side will not convey water.
2. The n-value is 0.020 and the curb height is 0.67 ft and the limiting Froude number is 1.25 because the street has a steep slope.



3. Click the Global Parameters button.



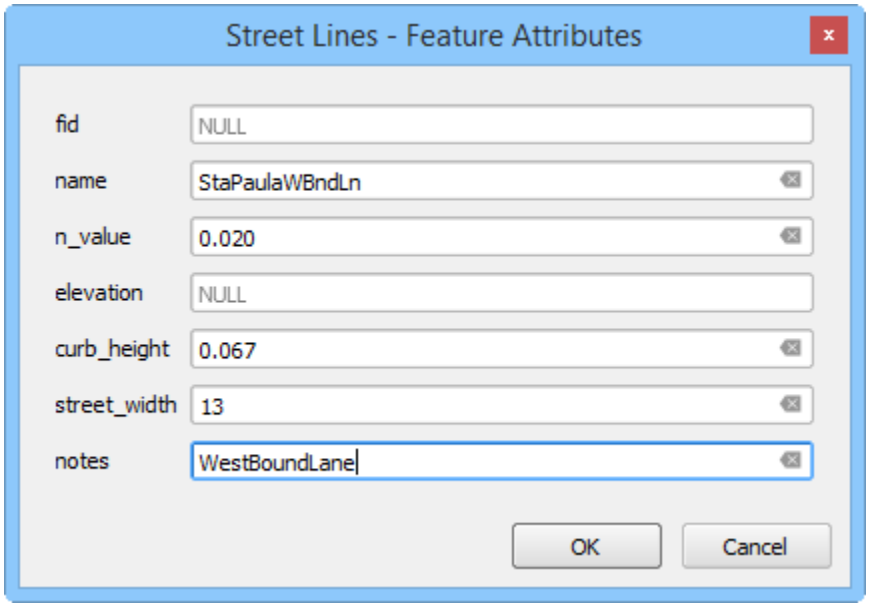
4. Fill the form and click Save.



Digitize a street segment.

The streets are a collection of cells that share water as a small rectangular channel. Each segment should represent a single street with an ability to distribute water from one end to the other. Any place where the street can no longer route water should be eliminated.

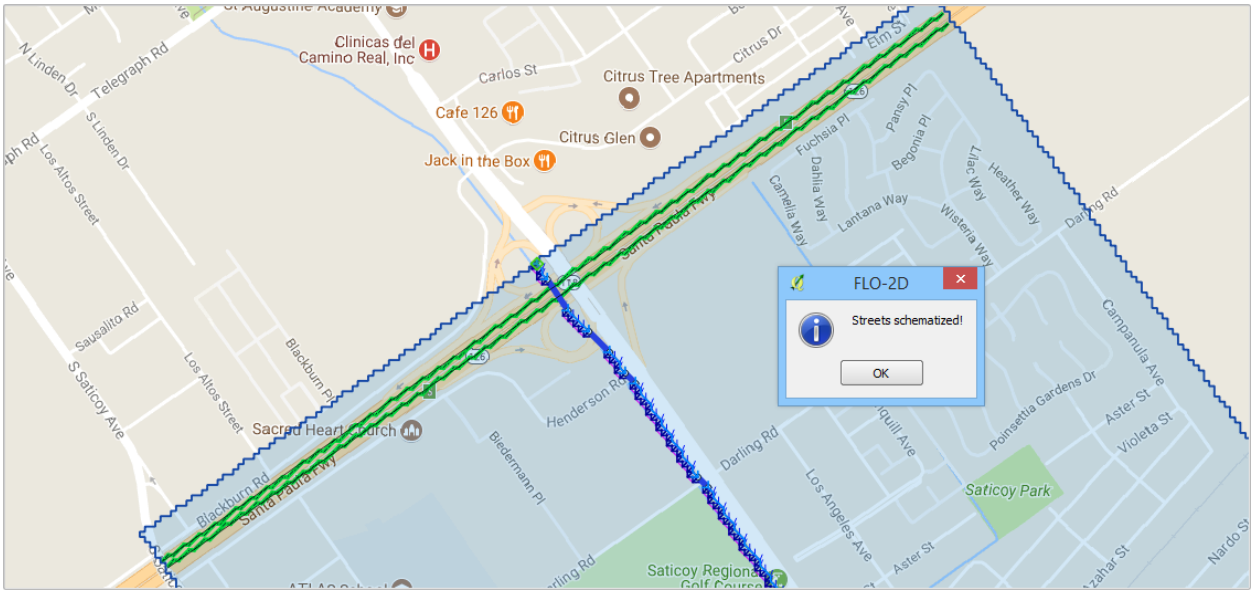
1. Click the Add a Street Line button.
2. Digitize the street segment on the map.
3. Enter the street data and click OK.



4. Save the street segment.

5. Repeat the process for the East Bound Lane.

6. Click the Schematize Button. The streets are schematized. Click ok to close the dialog box.



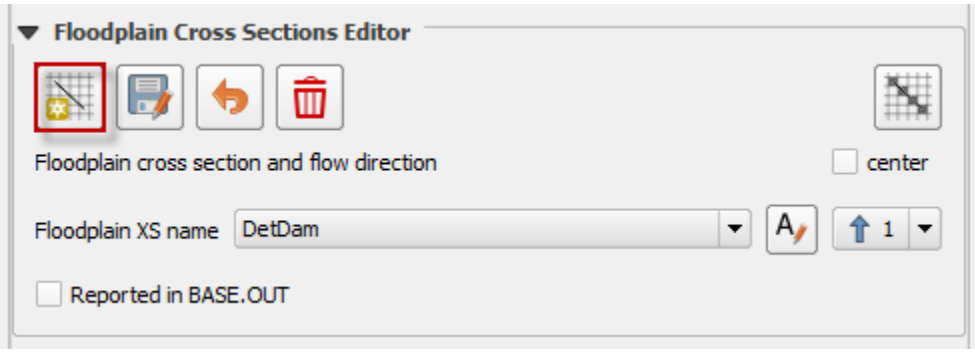
Troubleshooting

1. The street alignment can be adjusted by editing the street line in the user layers.
2. If the data is not written to the STREET.DAT file correctly. Check the schematic layer, it can be edited in the schematic layer attribute table.
3. Street intersections do not typically convey water. The street crown is designed to keep water out of the intersection. Set up intersections by stopping the connecting streets on cell back from the intersection.

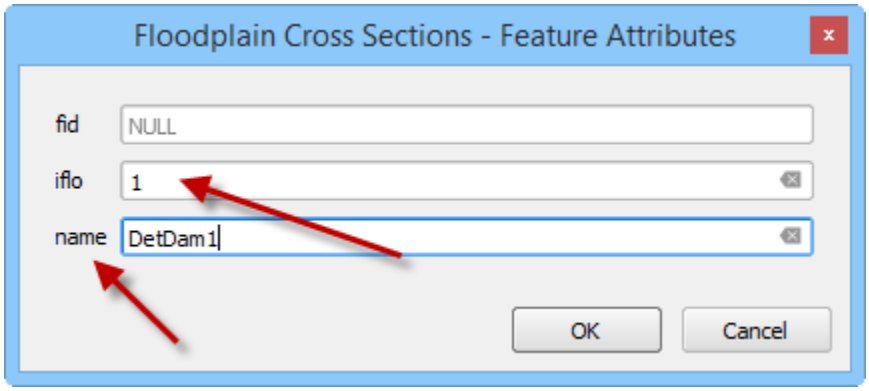
Floodplain Cross Section Editor

The floodplain cross section editor is used to set up the FPXSEC.DAT file. This section will describe how to digitize and schematize the data.

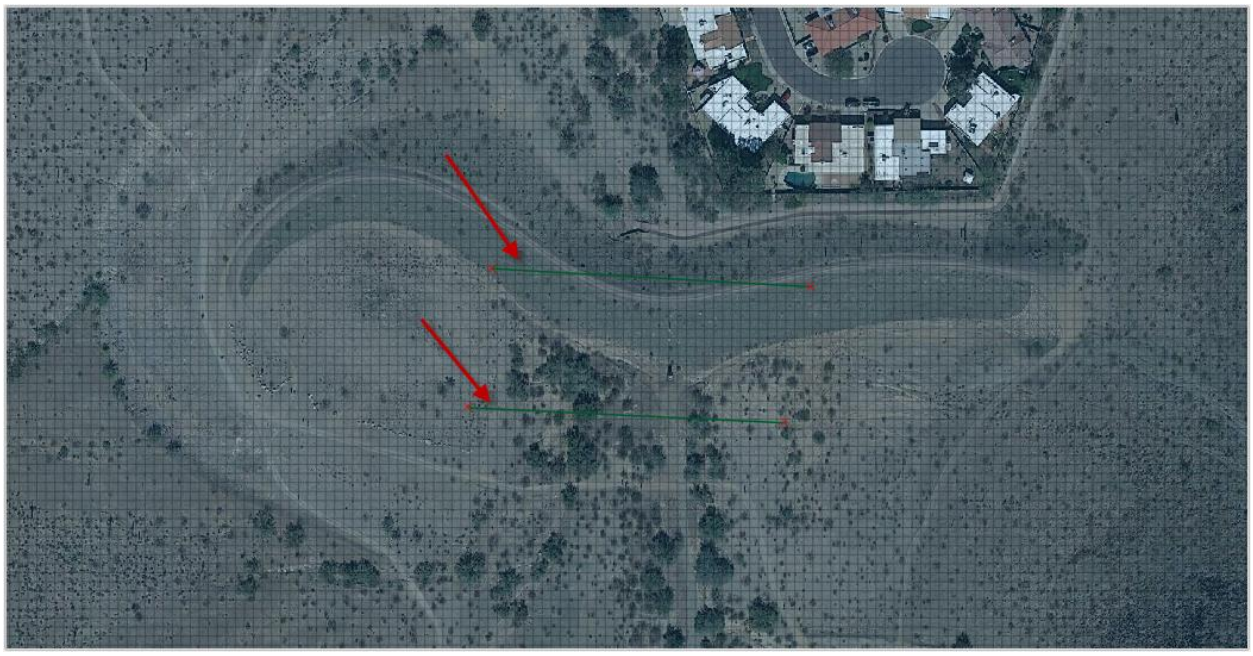
Use the editor widget to create the cross section.



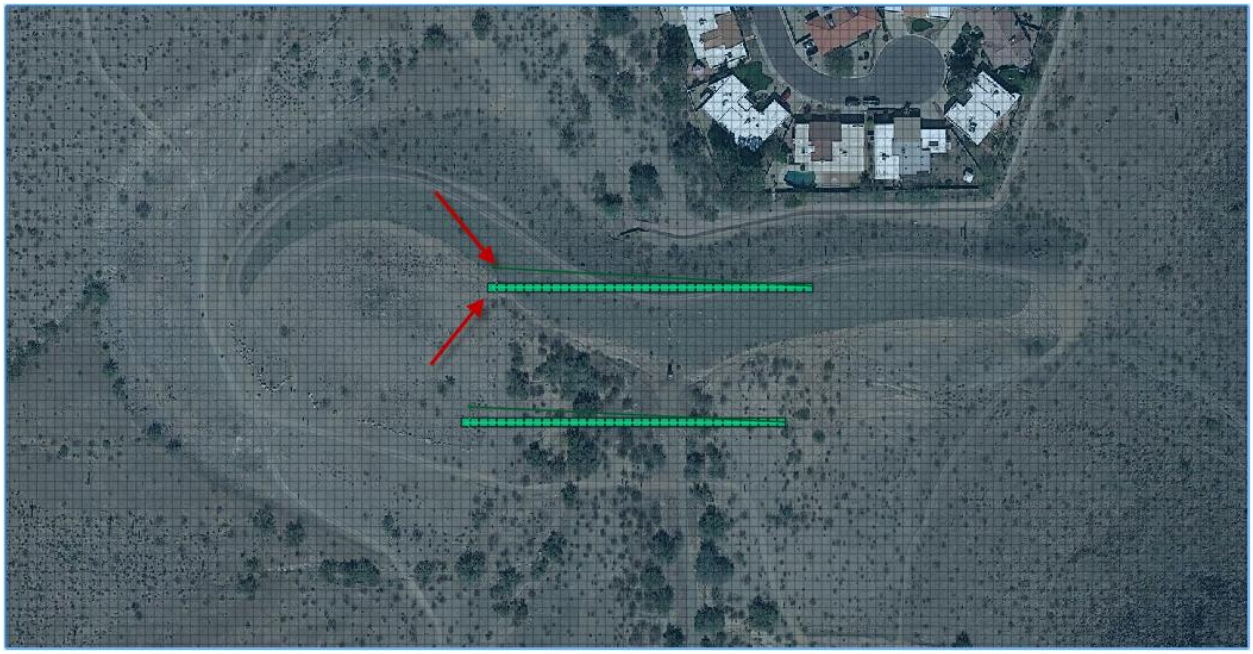
1. Click the floodplain cross section button.
2. Digitize the cross section and fill the dialog box. Click OK.



3. Click save to preserve the data. The user cross section is a green line.



4. Click Schematize to save the data. The schematized cross section is corrected to meet the FLO-2D criteria for Floodplain cross sections.

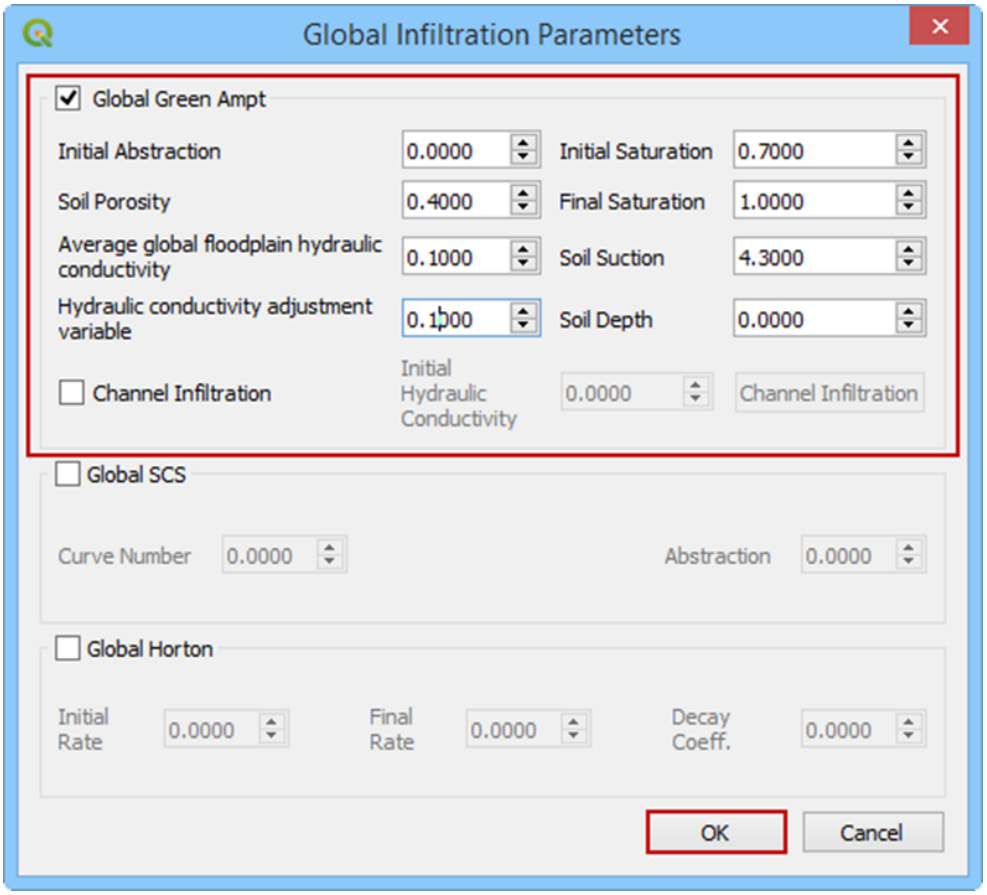


Green-Ampt

The Green-Ampt infiltration editor can add global or spatially variable infiltration data to the INFIL.DAT file.

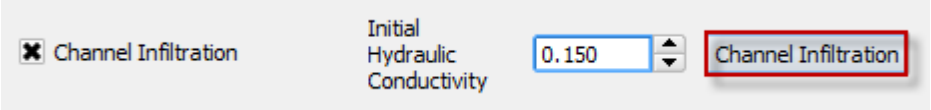
Global

Global data is set up using the Global Infiltration button. Click the button to open the editor dialog box.

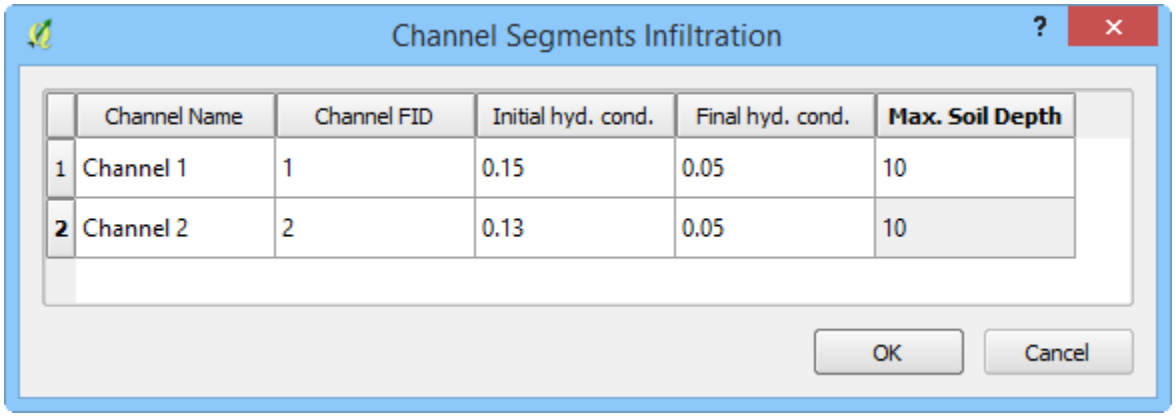


Channel Infiltration

To assign channel infiltration, use the channel infiltration editor. Set a global hydraulic conductivity for all channel elements and click the Channel Infiltration button.



Local channel infiltration is set by segment in the dialog box.



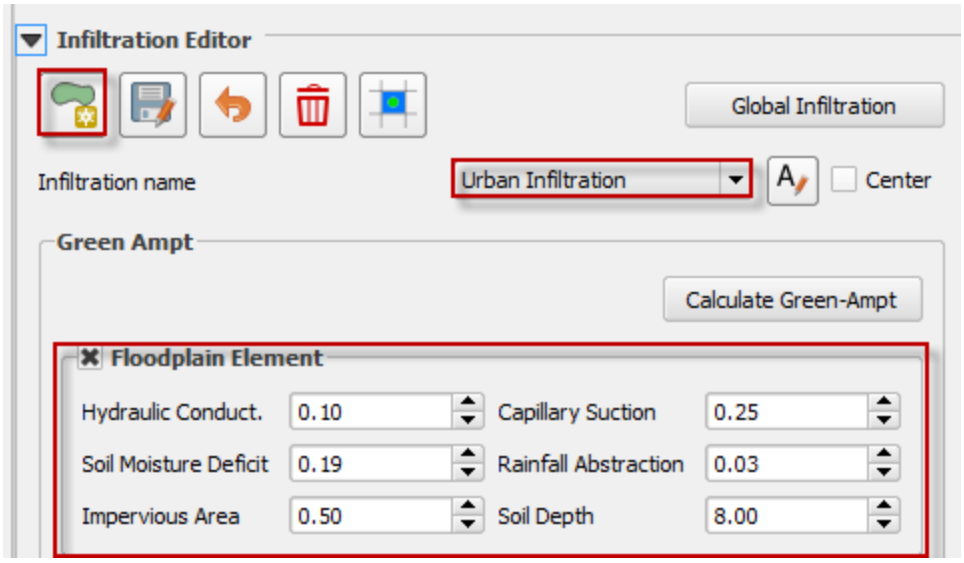
Green-Ampt from Infiltration Polygons

Spatially variable floodplain infiltration is set by digitizing infiltration polygons or importing infiltration polygons. Use the polygon editor to digitize spatially variable infiltration. Create a polygon to represent an area of infiltration.

8. Click the create a polygon tool and digitize a polygon.
9. Name the infiltration polygon.

10. Fill the table for the infiltration data.
11. Click *Save*.

12. Click Schematize.



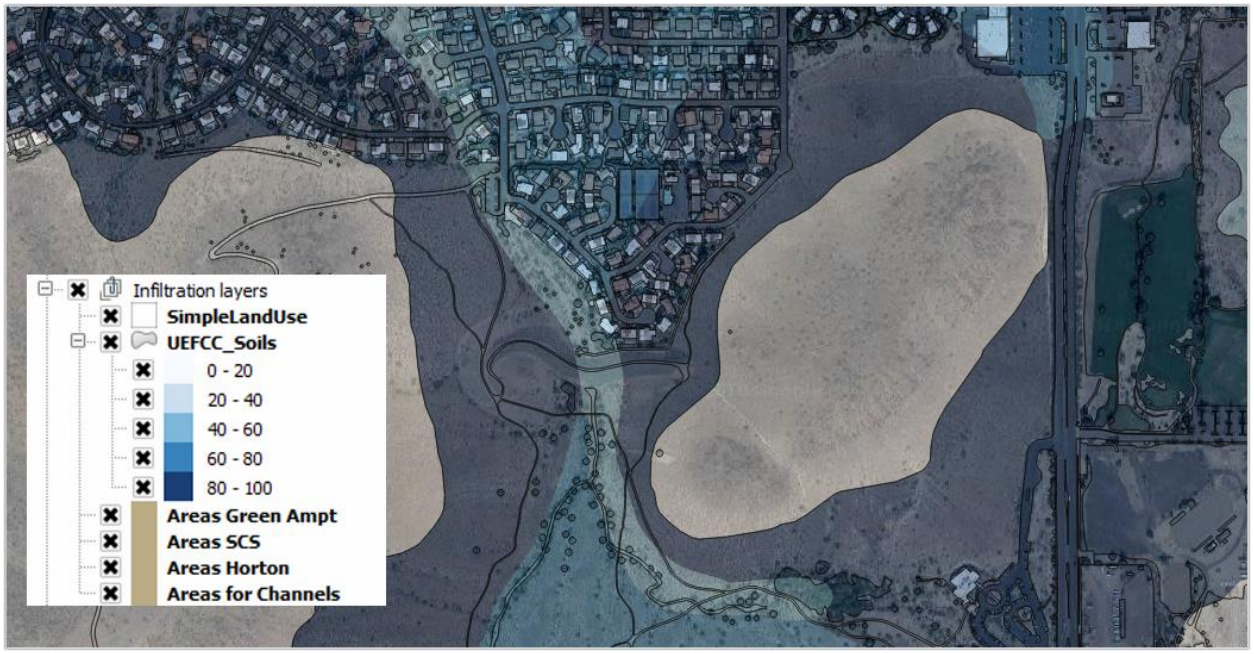
The infiltration polygons outline areas of cells that have similar infiltration characteristics. In the following image, the infiltration areas are different for urban, desert and desert drainage.

Green-Ampt from Infiltration Calculator

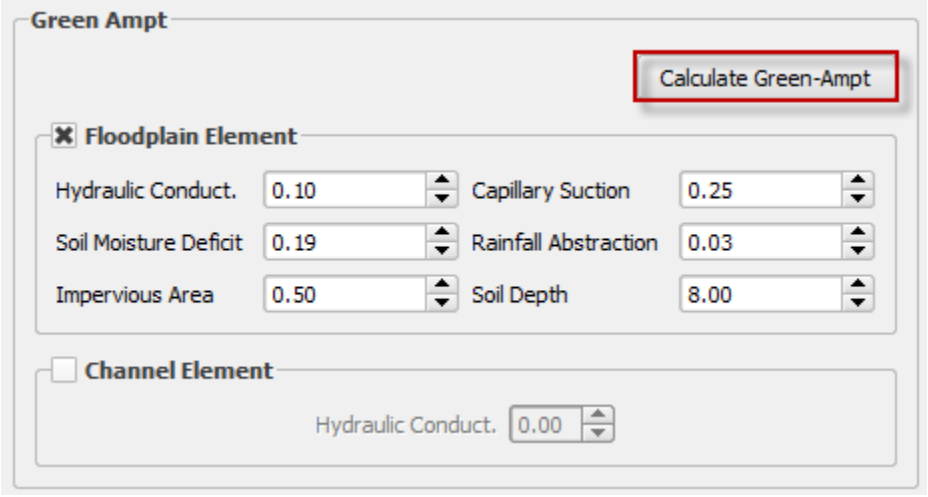
To use the Green-Ampt calculator, the user must prepare soil and landuse shapefiles. The soils data is acquired from the United States Department of Agriculture Web Soil Survey (USDA, 2017). The data is organized by soil group.

The land use data can be acquired from various sources but is generally available from the United States Geological Survey Land Cover website (USGS, 2017). The land use data can account for vegetative cover and impervious cover.

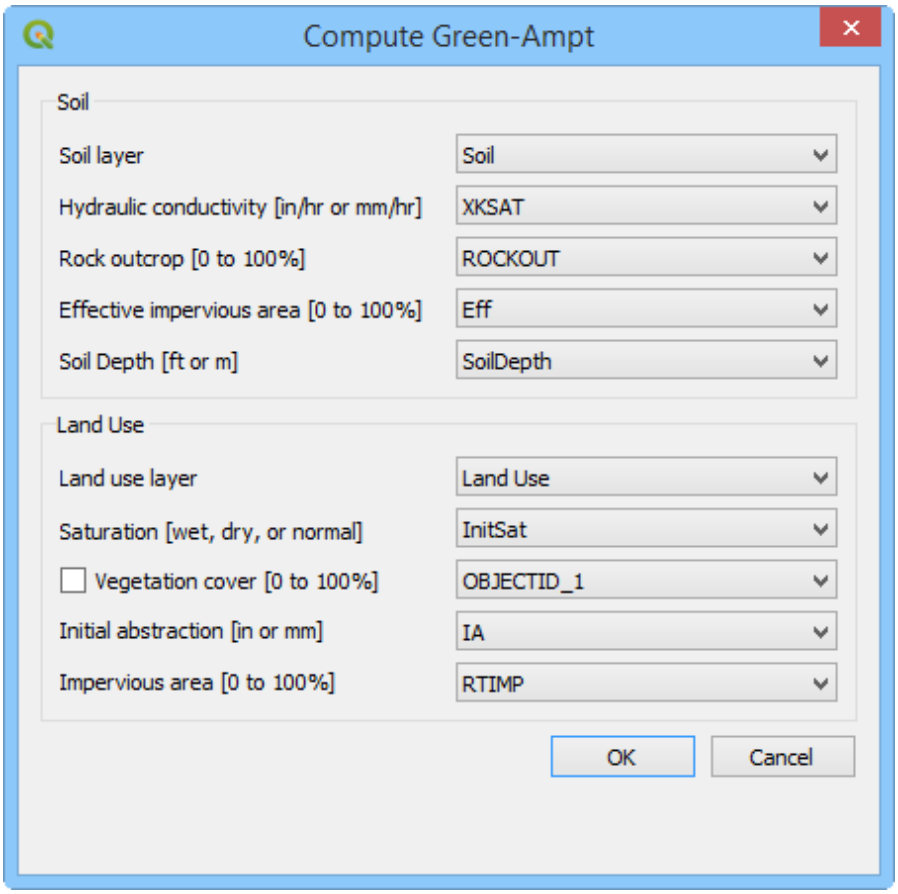
Prepare the data into shapefiles using QGIS or import them into QGIS.



To run the calculator, click the Calculate Green-Ampt button.



Use the following dialog box to assign the required shapefiles and fields to the calculator. Click OK to run the calculation. The plugin will assign spatially variable infiltration data to each cell. The data is exported to the INFIL.DAT file.

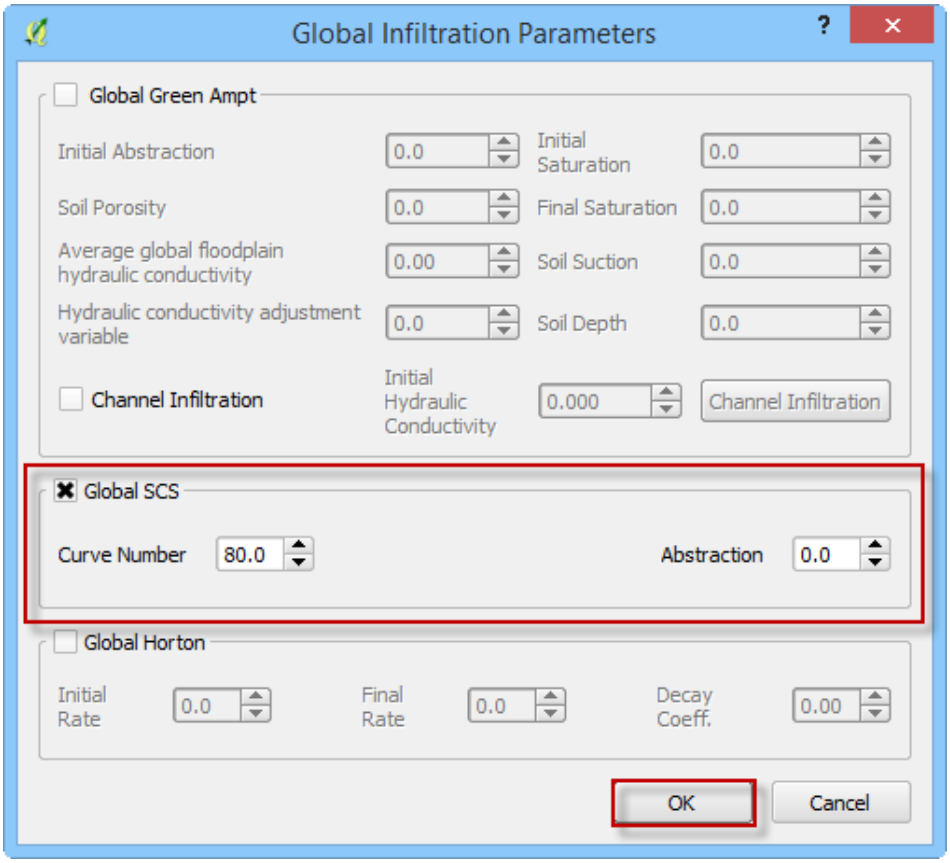


Global

The SCS infiltration editor can add global or spatially variable infiltration data to the INFIL.DAT file for infiltration curve numbers.

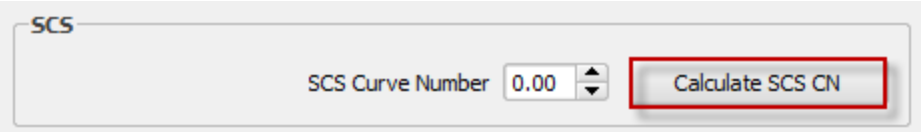
1. Set up the Global Infiltration first. Click Global Infiltration.

2. Fill the Global Infiltration dialog box.

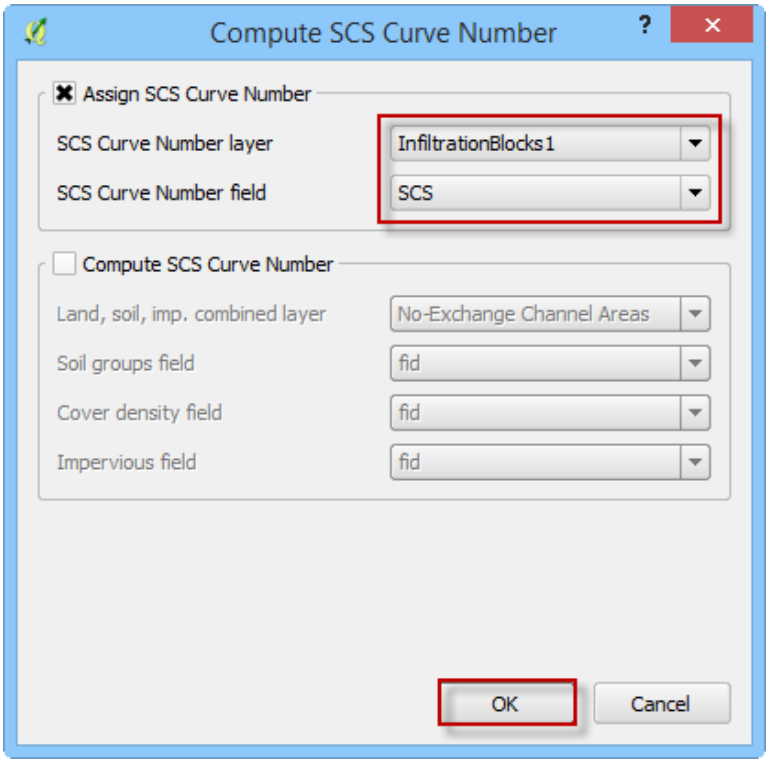


SCS Calculator Single Shapefile

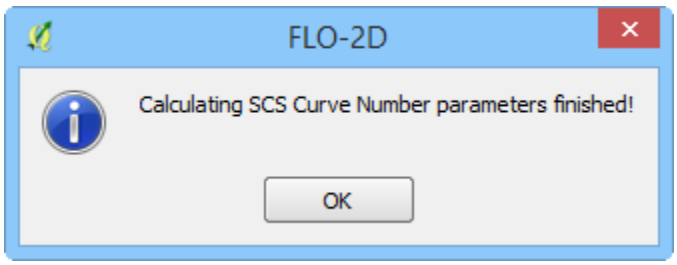
1. Click the Calculate SCS CN button.



2. Select the layer and field with the infiltration data and click OK to run the calculator.



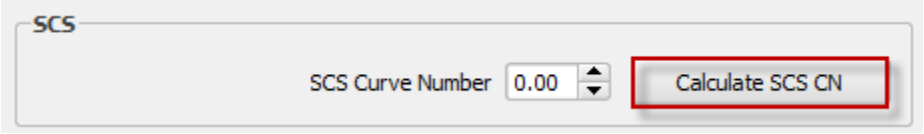
3. When the calculation is complete, the following box will appear. Click OK to close the box.



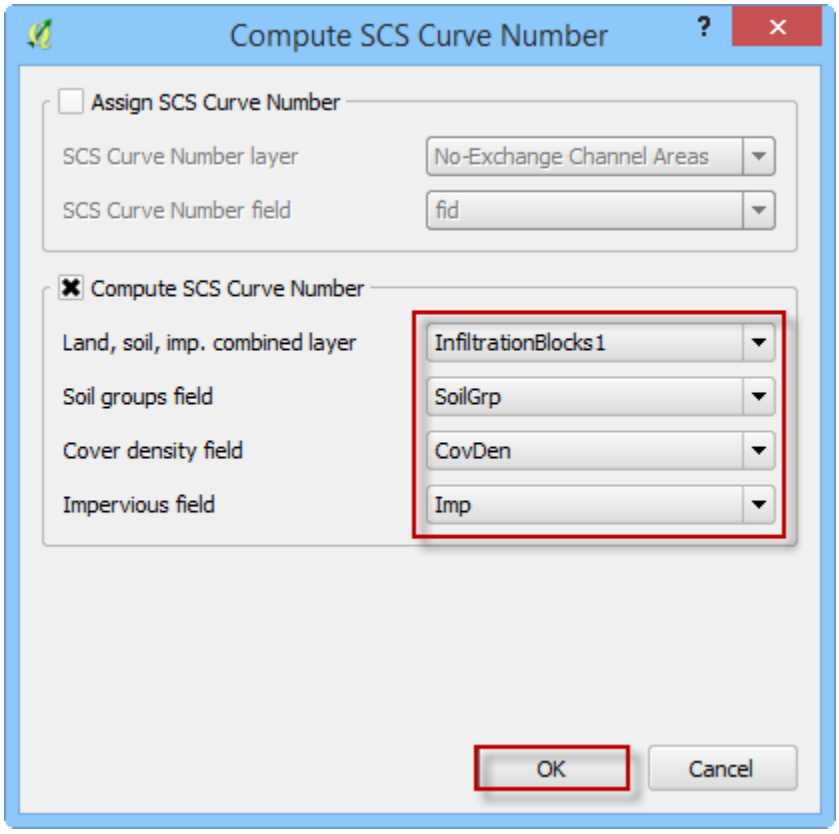
SCS Calculator Single Shapefile Multiple Fields

Use this option to calculate SCS curve number data from a single layer with multiple fields. This is a vector layer with polygon features and field to define the soil group, vegetation coverage and impervious space. This option was developed specifically for Pima County.

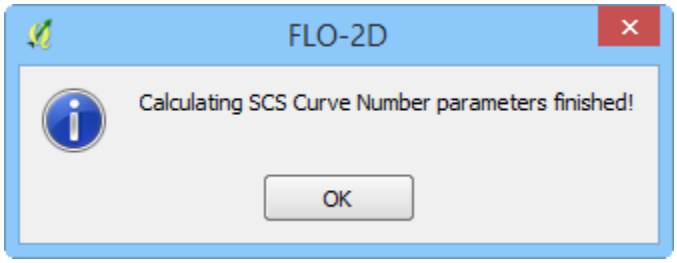
1. Click the Calculate SCS CN button.



2. Select the layer and fields with the infiltration data and click OK to run the calculator.



3. When the calculation is complete, the following box will appear. Click OK to close the box.



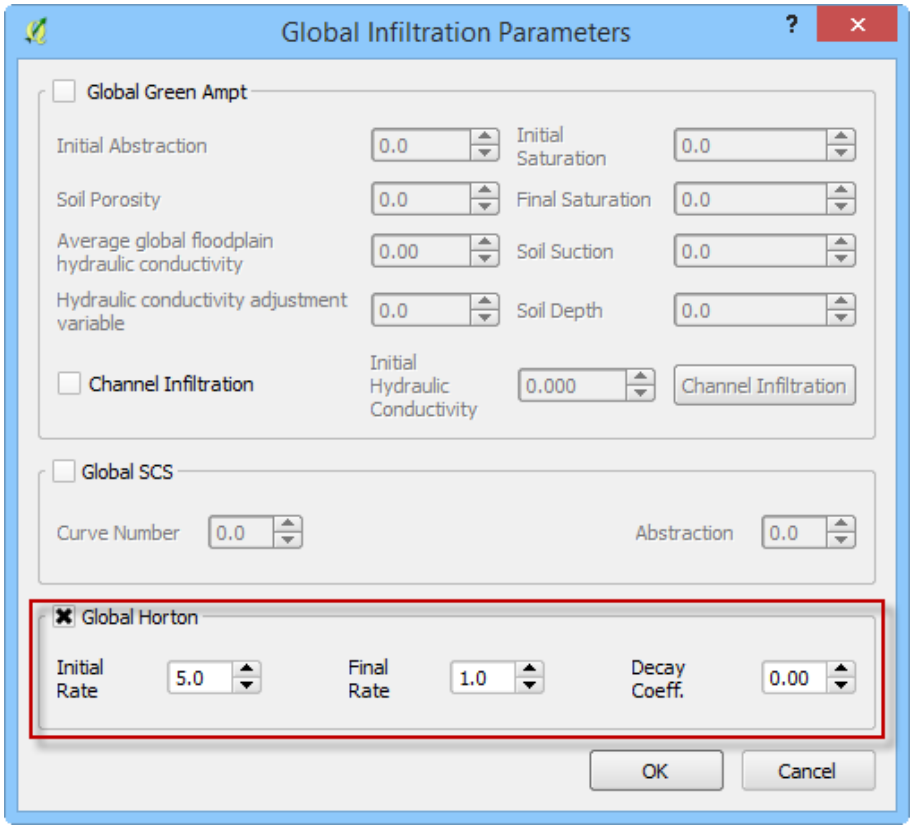
Horton

Global

The SCS infiltration editor can add global or spatially variable infiltration data to the INFIL.DAT file for infiltration curve numbers.

1. Set up the Global Infiltration first. Click Global Infiltration.

2. Fill the Global Infiltration dialog box.

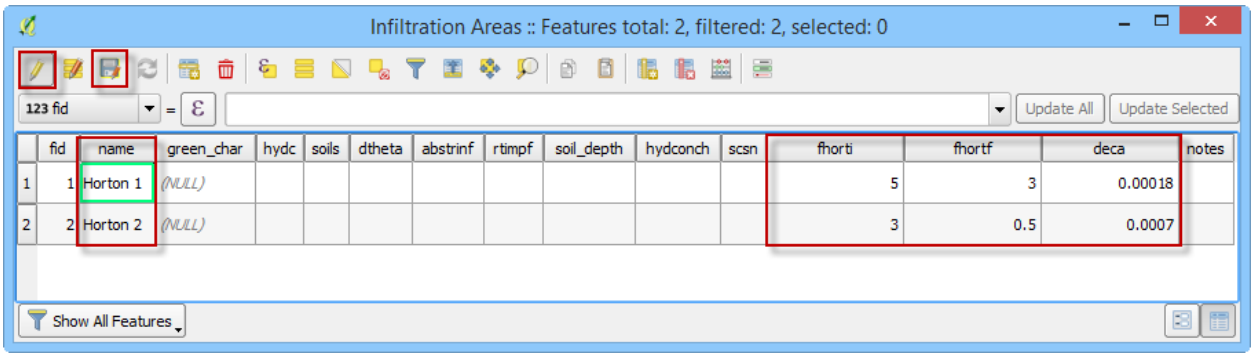


Horton Spatially Variable Method

Spatially variable Horton infiltration is created by digitizing infiltration polygons. Use the polygon editor to digitize spatially variable infiltration. Create a polygon to represent an area of infiltration.

3. Click the create a polygon tool and digitize a polygon.

4. Click *Save*.
5. Right Click the Infiltration Areas layer (User Layers) and open the Attributes Table. Click the Editor Pencil button.
6. Name the infiltration polygons and fill out the data for fhorti, fhori, and deca.
7. Click the Save button and Editor Pencil button.



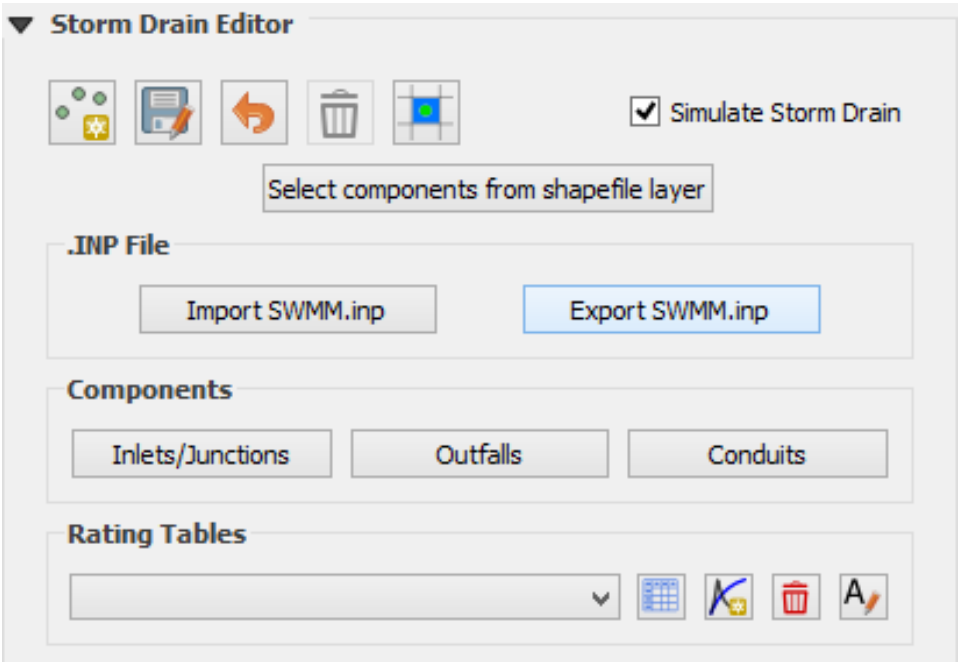
8. Click Schematize.

Storm Drain Editor

The Storm Drain Editor tool has several functions to prepare the data that integrates the FLO-2D surface water model with the storm drain model:

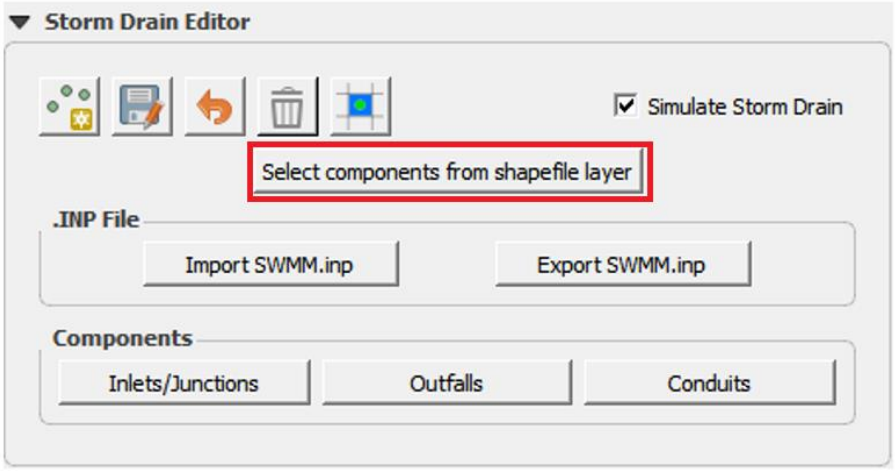
- Reads/writes data from the SWMM.inp file, and associates inlets/junctions, outfalls, and conduits with FLO-2D cells.
- Creates and edit the SWMM.INP file by importing inlets/junctions, outfalls, and conduits from shapefiles.
- Saves and edits the FLO-2D SWMMFLO.DAT, SWMMRT.DAT and SWMMOUTF.DAT files containing the inlet and outlet data.
- Displays the inlets and outlets and the piping network connections.

The storm drain editor widget has the following functionality:



Create storm drain components from shapefiles

The select component from shapefile feature is used to build a storm drain network from a set of shapefiles.



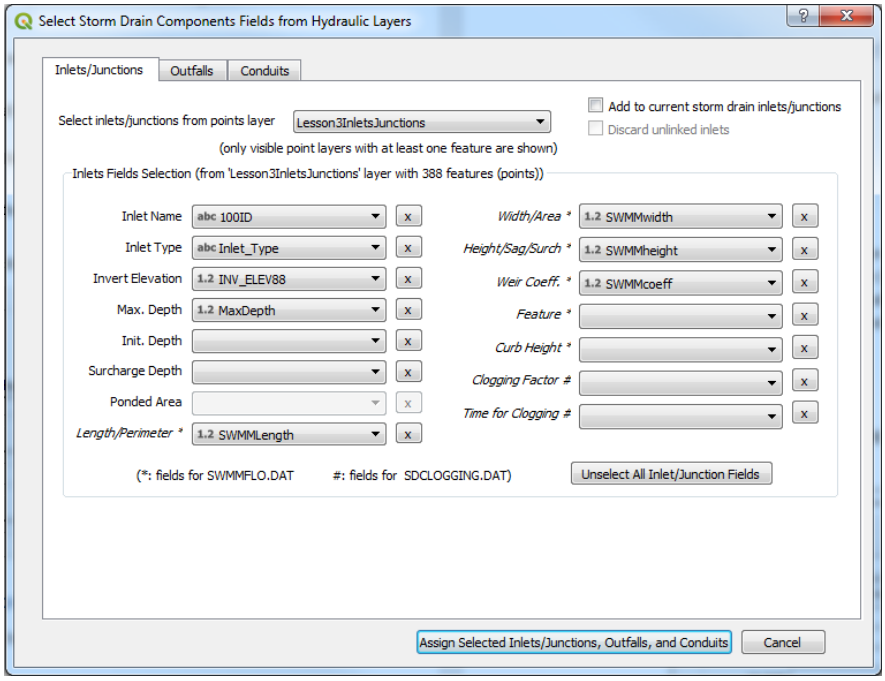
The following information summarized the data that is needed in the Shapefiles for the creation of the *.INP. The collection of As-Built always help when assumptions are needed in the creation of the *.INP file.

Inlets/Junctions Shapefiles

The following data is needed for inlets/junctions. Data identified as optional is not required.

1. Name
2. Location (X- and Y- Coordinates)
3. Invert Elevation
4. Rim Elevation or Maximum Depth
5. Initial Depth (Optional)
6. Surcharge Depth (Optional for inlets, Non-optional for Manholes)
7. External Inflows receive at inlet/junction (Optional). Typically, inflow coming from out of the domain can be imposed as an inflow condition to the Inlets/Junctions next to the boundary.
8. Inlet Geometry for Inlets type 1,2,3, 4 and 5. For Inlet type 4, geometry is needed for the creation of the rating table.
9. Length (1 or 2) / Perimeter (3 or 5)
10. Width (2)/ Area (3 or 5)
11. Height (1 or 2) / Sag (3) / Surcharge (5)
12. Vertical Opening (1) / Horizontal Opening (0).
13. Curb Height (Optional)

Click the button to open a dialog box that calls shapefiles attributes and assign them to the storm drain features.

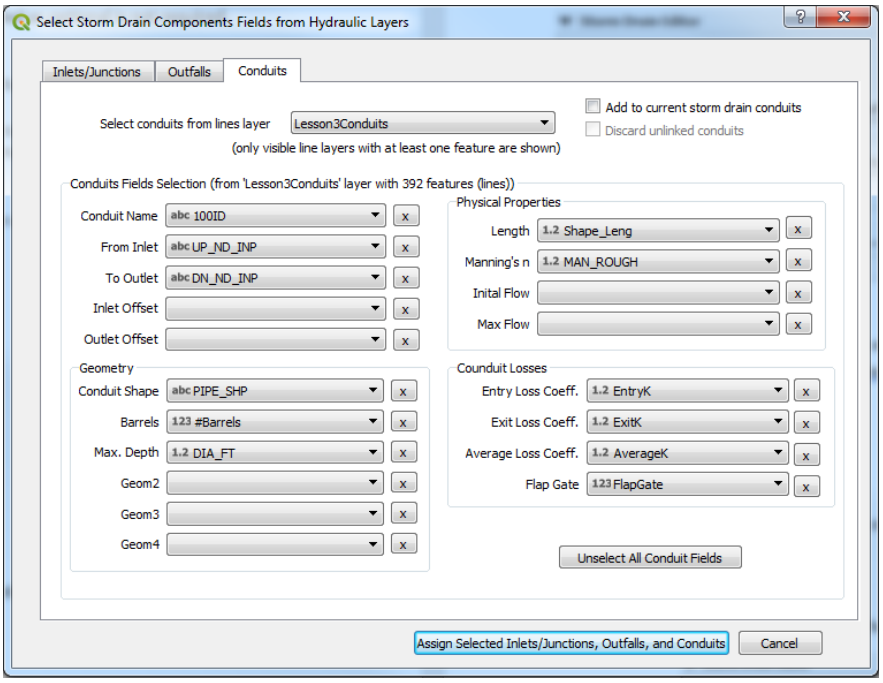


Conduits Shapefiles

The following data is needed for conduits. Data identified as optional is not required.

1. Name
2. Inlet and Outlet Node: upstream and downstream inlets/junctions connected to the pipe.
3. Shape, example: Circular.
4. Max Depth of Cross Section, example: diameter for circular shape. Other conduits shapes will require additional geometry data, see Storm Drain Manual for detailed information.
5. Number of Barrels.
6. Pipe Length.
7. Manning's Roughness Coefficient: typical values can be assigned based on the pipe material and conditions.
8. Inlet/Outlet Offset (optional)
9. Entry Loss Coefficient (optional)
10. Exit Loss Coefficient (optional)
11. Average Loss Coefficient (optional)
12. Flap Gate (optional)
13. Initial Flow (optional)
14. Maximum Flow (optional)

Click the button to open a dialog box that calls shapefiles attributes and assign them to the storm drain features.

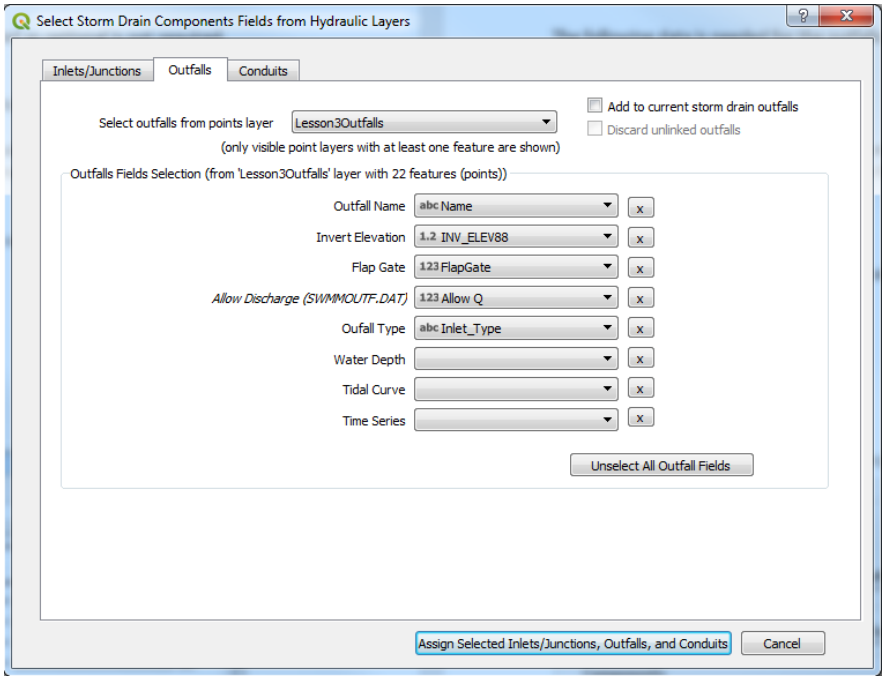


Outfall Shapefiles

The following data is needed for the outfalls. Data identified as optional is not required.

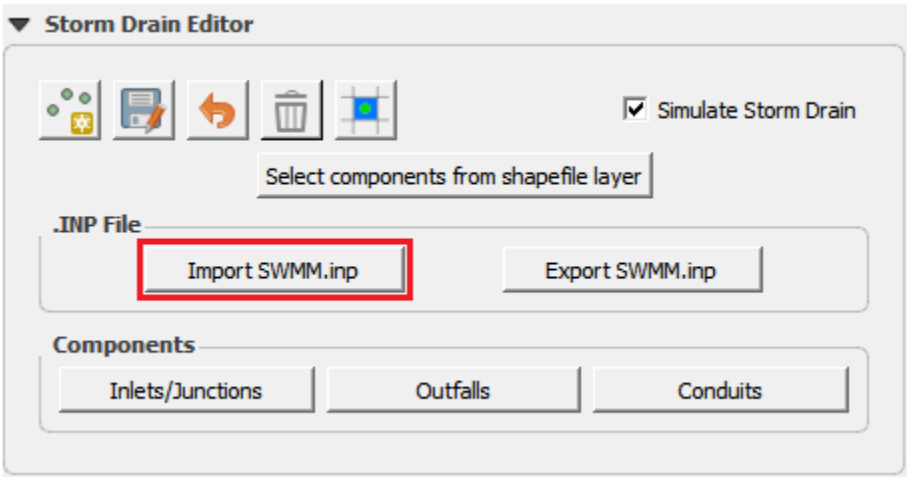
1. Name
2. Location (X and Y Coordinates)
3. Invert Elevation
4. Type: FREE to be connected to the surface model. Normal, Fixed, Tidal or Timeseries can also be modeled but they will not be connected to the surface flow model.
5. Flap gate (optional)

Click the button to open a dialog box that calls shapefiles attributes and assign them to the storm drain features.



Import SWMM.inp

An existing SWMM.inp project can be imported in a FLO-2D Surface System. Click on Import SWMM.inp and browse the project folder that contains the file.



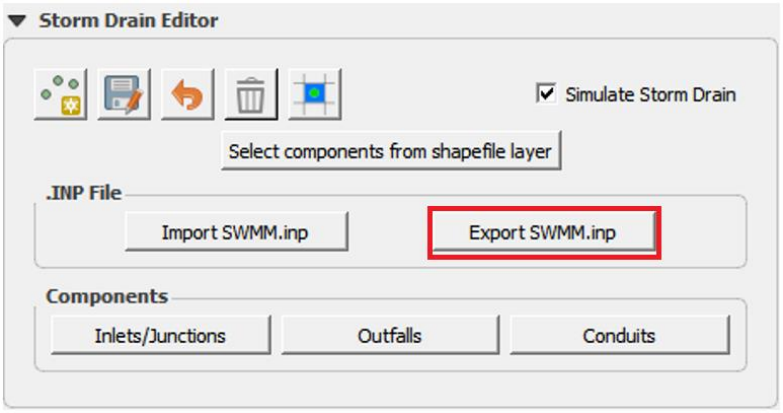
This button loads inlets/junctions, outfalls and conduits from an *.INP file. The Storm Drain data needs to be schematized, the table components that can be opened from the Storm Drain Editor will contain the variables from the SWMM.inp file. Additional data is needed for the SWMMFLO.DAT, SWMMOUTF.DAT and SWMMFLORT.DAT files.

QGIS FLO-2D layers will be filled up with the data from the following *.INP groups:

- Inlets/Junctions
- Outfalls
- Conduits
- Cross sections
- Losses
- Coordinates (required coordinates only)

Export SWMM.inp

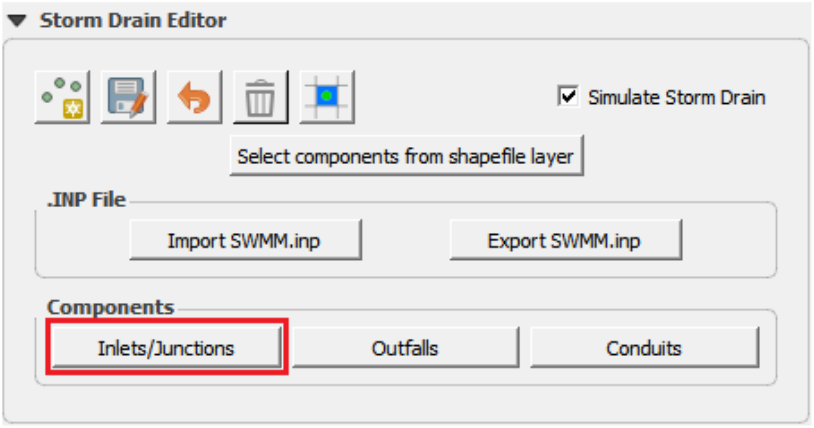
Export SWMM.inp file in a FLO-2D format prior to running. The SWMM.INP can be created from shapefiles and then exported or it might be modified from an existing SWMM.INP.



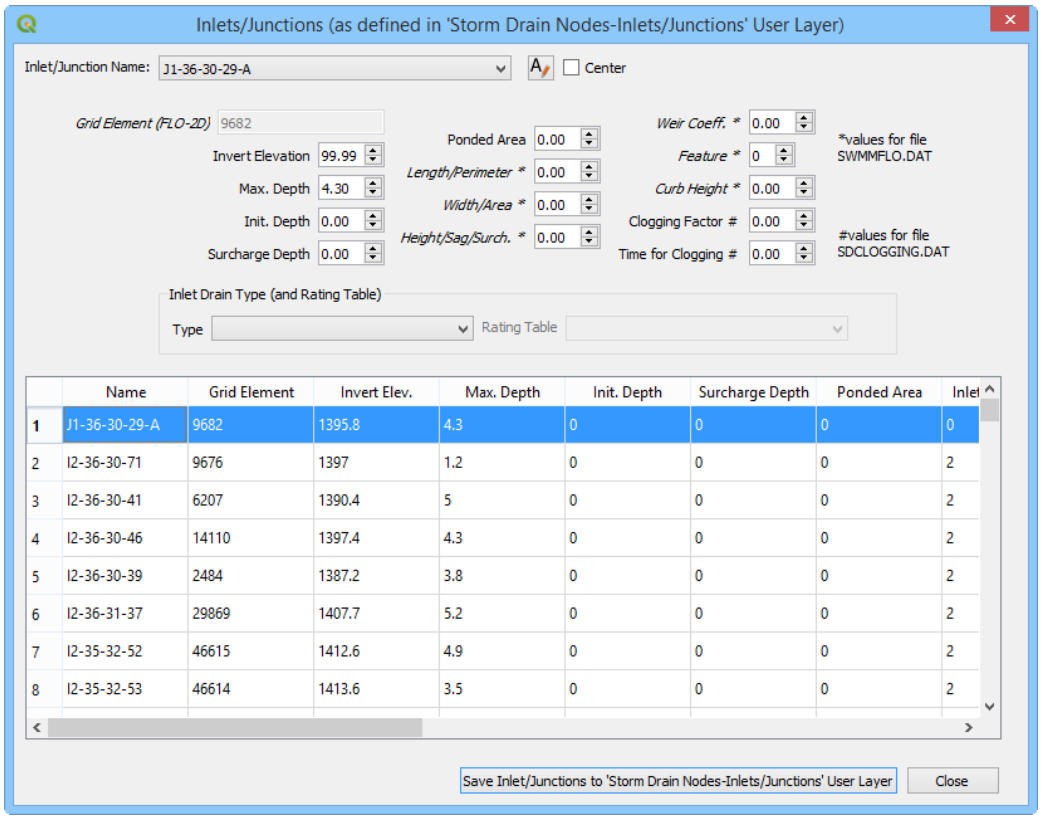
Data save in the Components tables is written to the .INP file using this function.

Components: Inlets/Junctions

Edit components that already exist using the Components editors for Inlets/Junctions, Outfalls and Conduits.

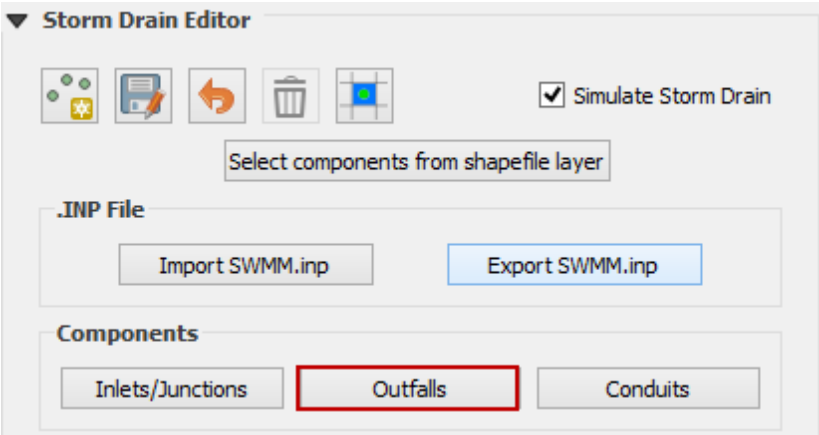


A dialog is shown with data for the selected component, in this case the Inlets/Junctions were selected, the user can edit the tables.

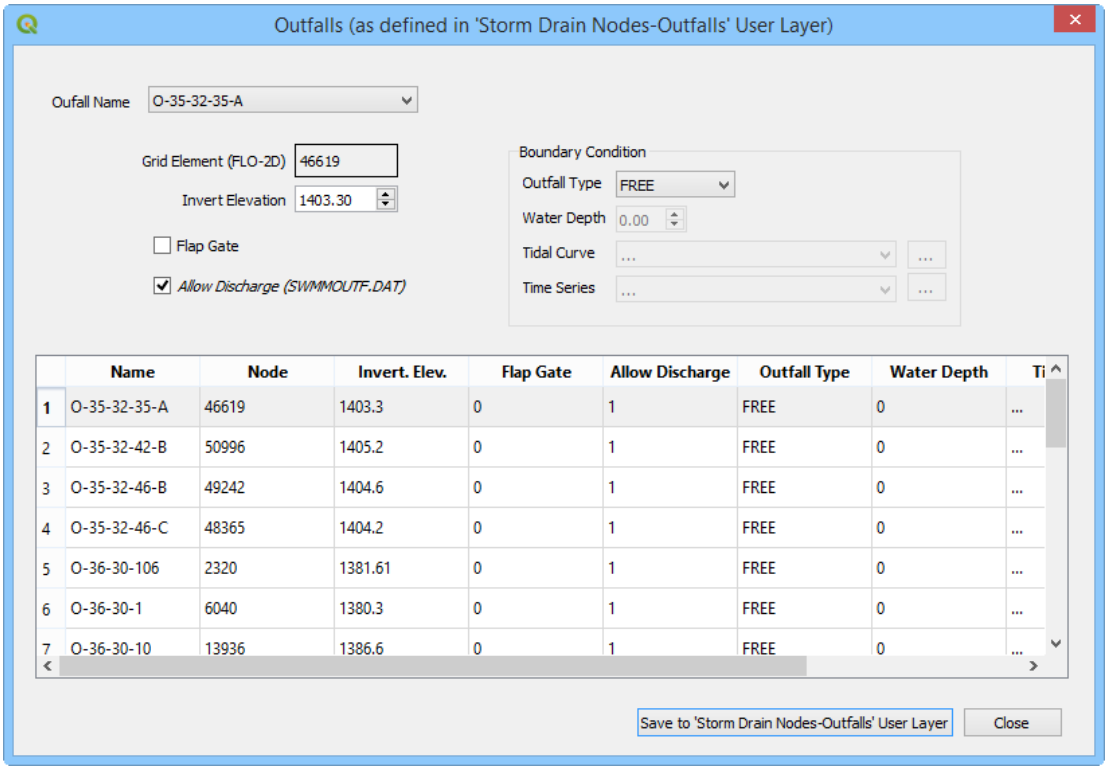


Components: Outfalls

Edit components that already exist using the Components editors for Inlets/Junctions, Outfalls and Conduits.

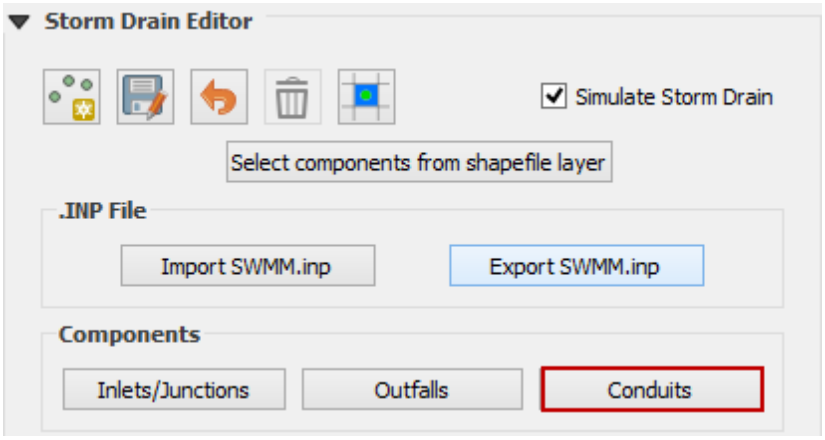


A dialog is shown with data for the selected component, in this case the Outfalls were selected, the user can edit the tables.

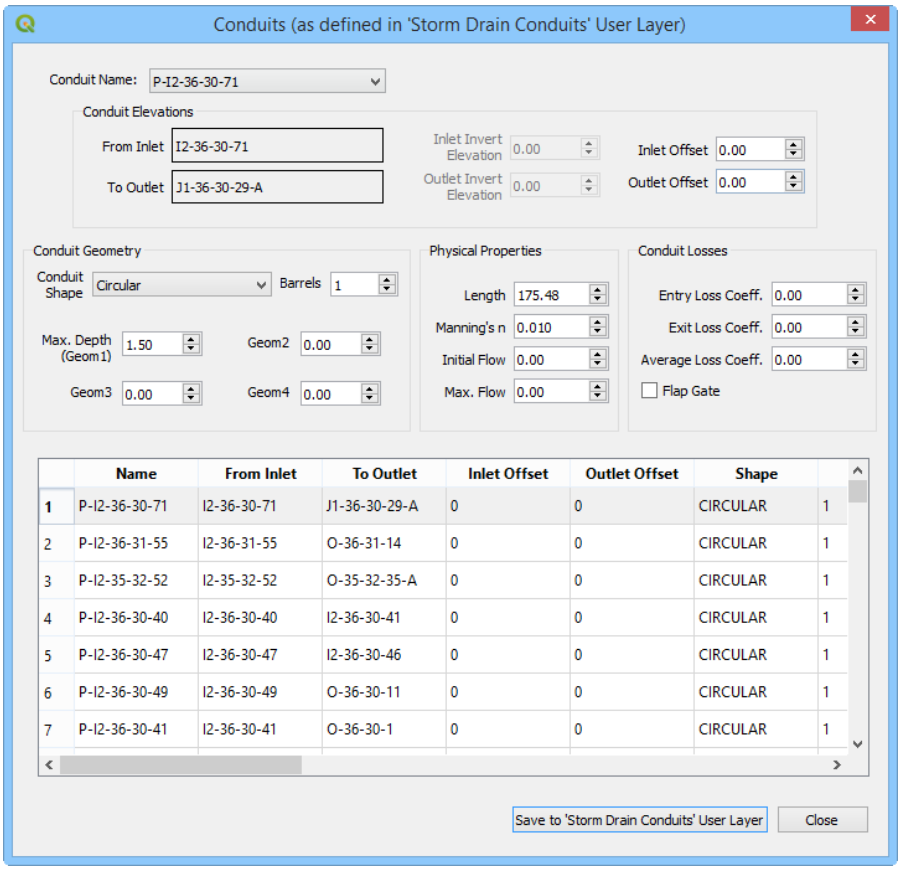


Components: Conduits

Edit components that already exist using the Components editors for Inlets/Junctions, Outfalls and Conduits.

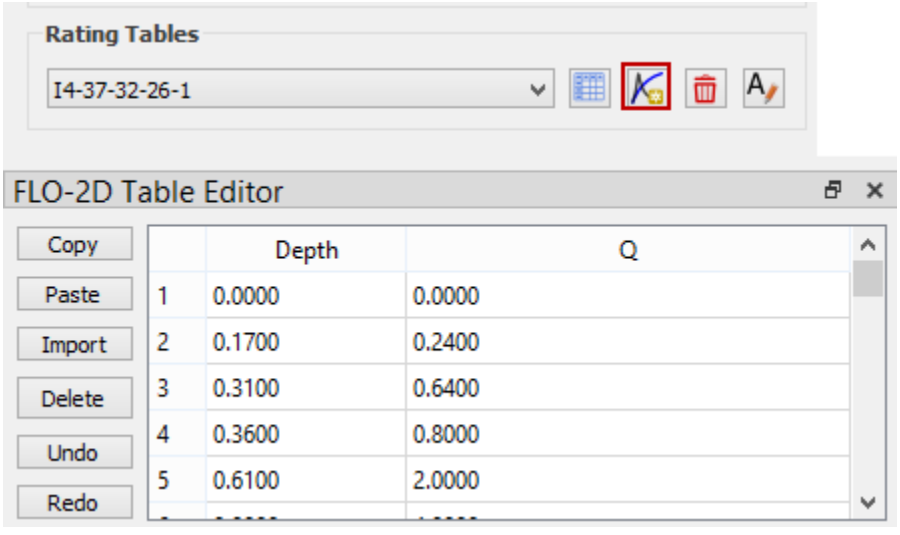


A dialog is shown with data for the selected component, in this case the Outfalls were selected, the user can edit the tables.

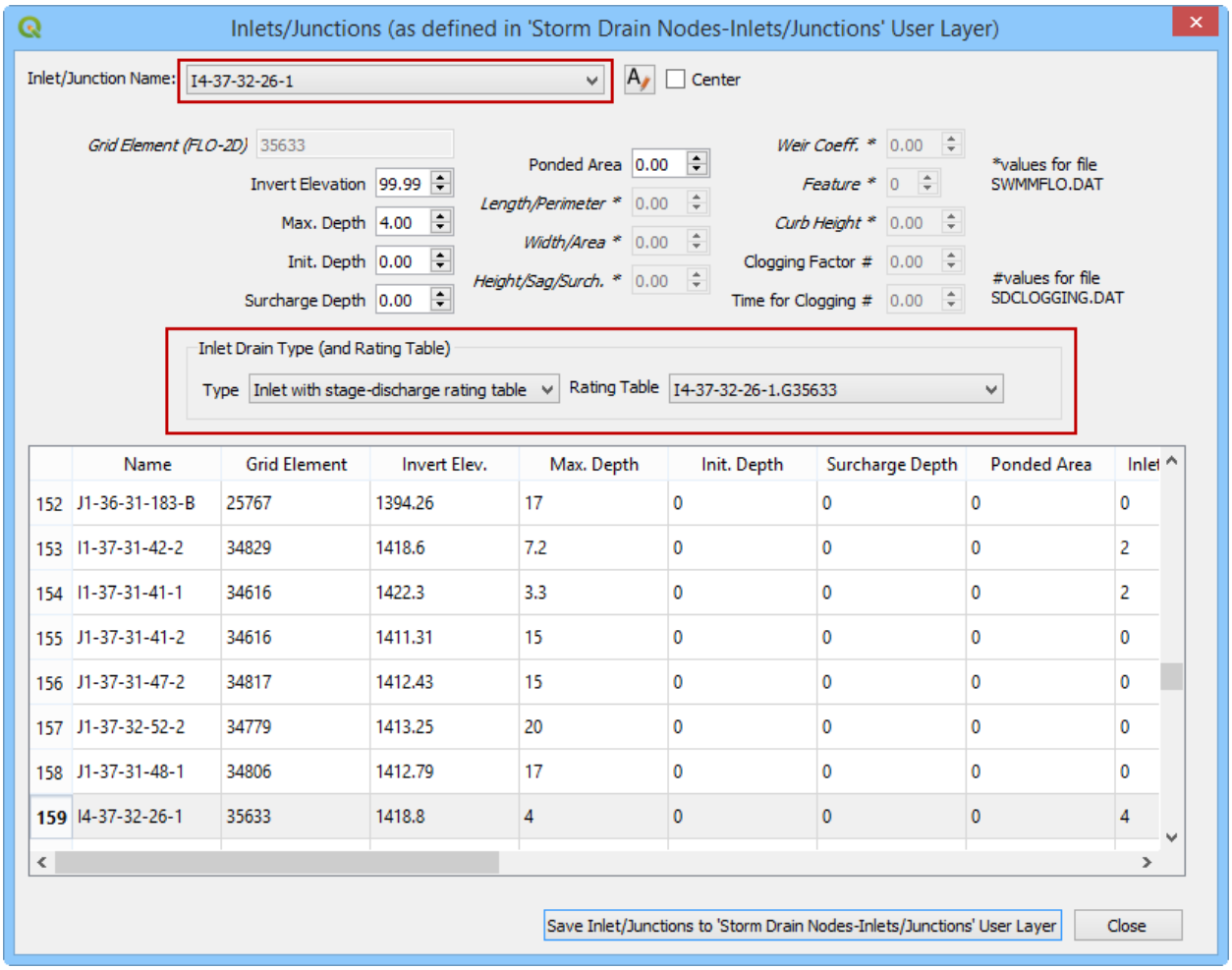


Rating Tables

Set up the rating tables by adding a rating table to the table editor and assigning the table to the correct inlet.



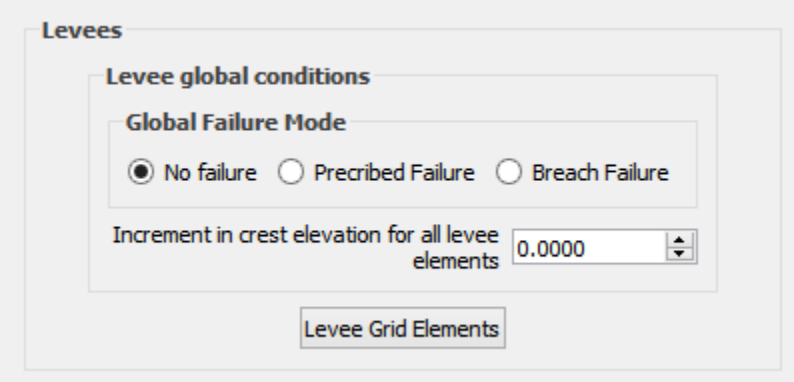
Use the Inlet Editor to assign the table to the Type 4 inlet.



Levees Breach Editor

Levees – Levee Global Conditions

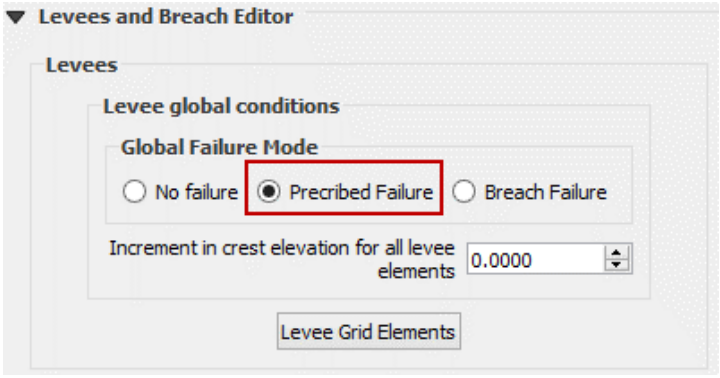
The levee global conditions are used to set the levee failure switch and to add a global elevation increment for all levee elements. Both variables are assigned to LEVEE.DAT.



Set the breach failure mode with the radio buttons.

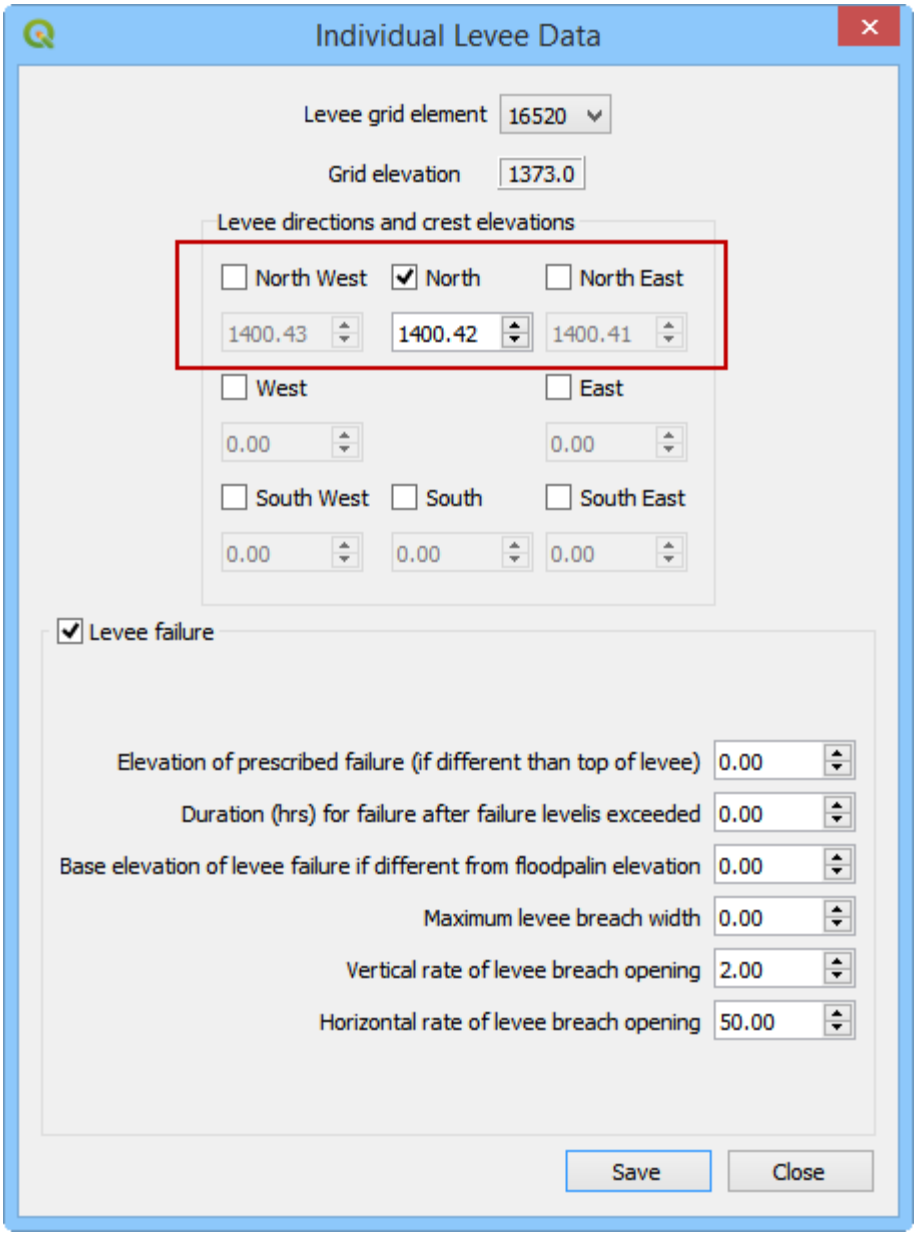
Breach Prescribed

To create Prescribed Breach data, set the Failure Mode to Prescribed Failure.



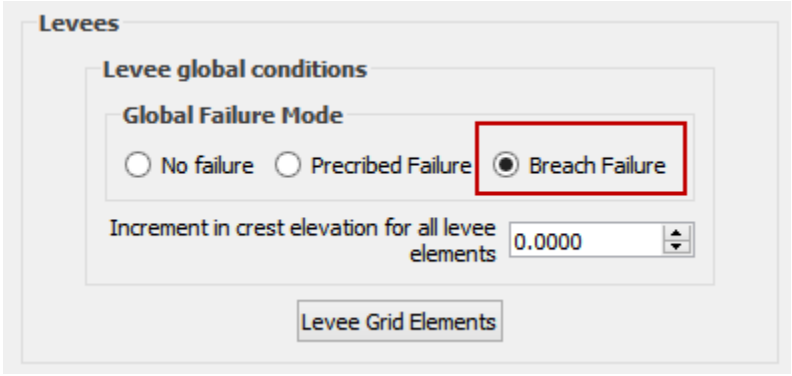
Use the Grid ID tool to pick the levee element to assign a prescribed breach to.

Click the Levee Grid Elements button and assign the levee breach data. ***Note It is important to uncheck all but one levee side. Note*** Choose the levee from the dropdown box. Use the cursor to enter the levee number.

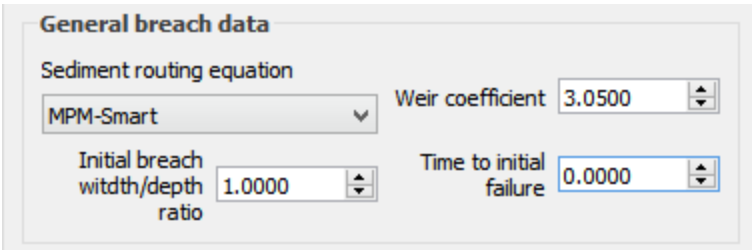

Individual Levee Data
Levee grid element
16520
Grid elevation
1373.0
Levee directions and crest elevations
North West
North
North East
1400.43
1400.42
1400.41
West
East
0.00
0.00
South West
South
South East
0.00
0.00
0.00
Levee failure
Elevation of prescribed failure (if different than top of levee)
0.00
Duration (hrs) for failure after failure levelis exceeded
0.00
Base elevation of levee failure if different from floodpalin elevation
0.00
Maximum levee breach width
0.00
Vertical rate of levee breach opening
2.00
Horizontal rate of levee breach opening
50.00
Save
Close

Breach Erosion

To create Breach Erosion data for the embedded Fread Dam Breach model, set the Failure Mode to Breach Failure.

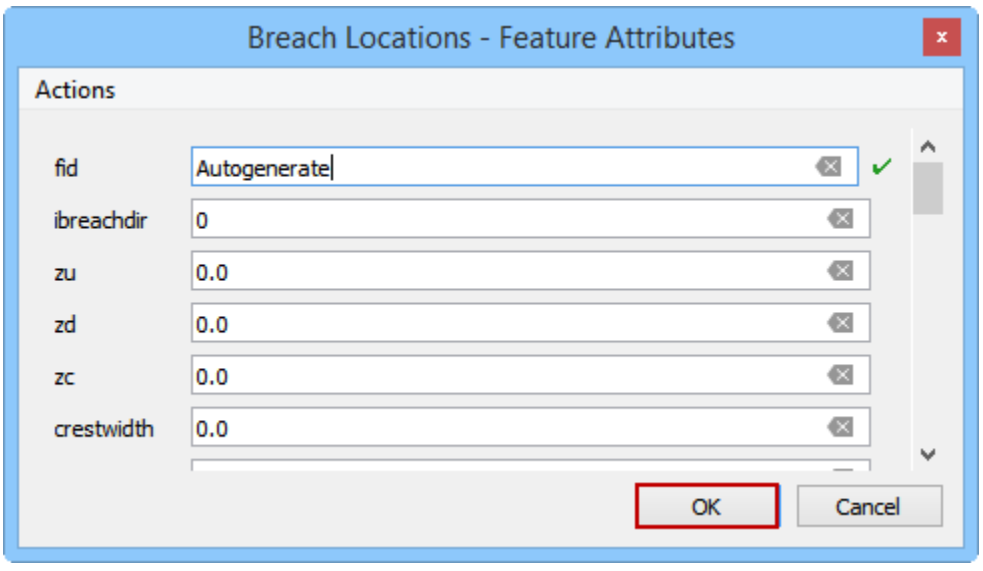


The General Breach Data is set in the Widget.



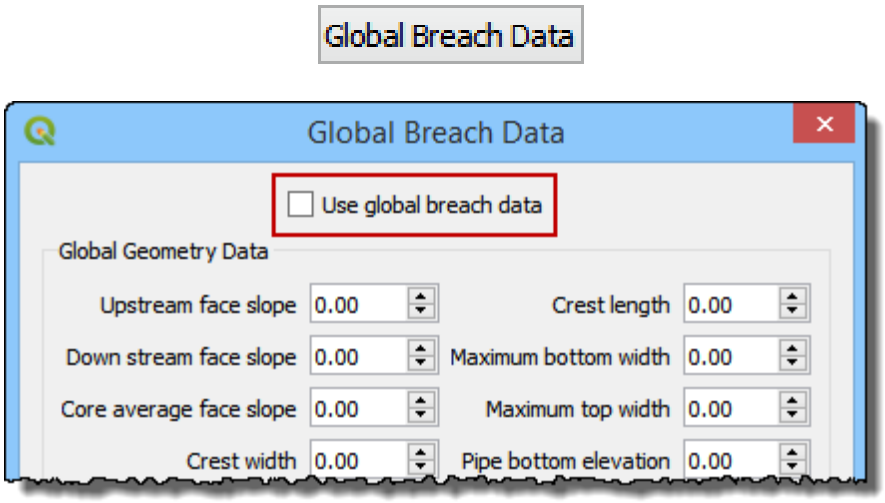
Use the Point button to create a Dam Breach Node. Click the button and then click the breach location on the map.

Click OK to close the Dialog box. It is not necessary to fill the data here.

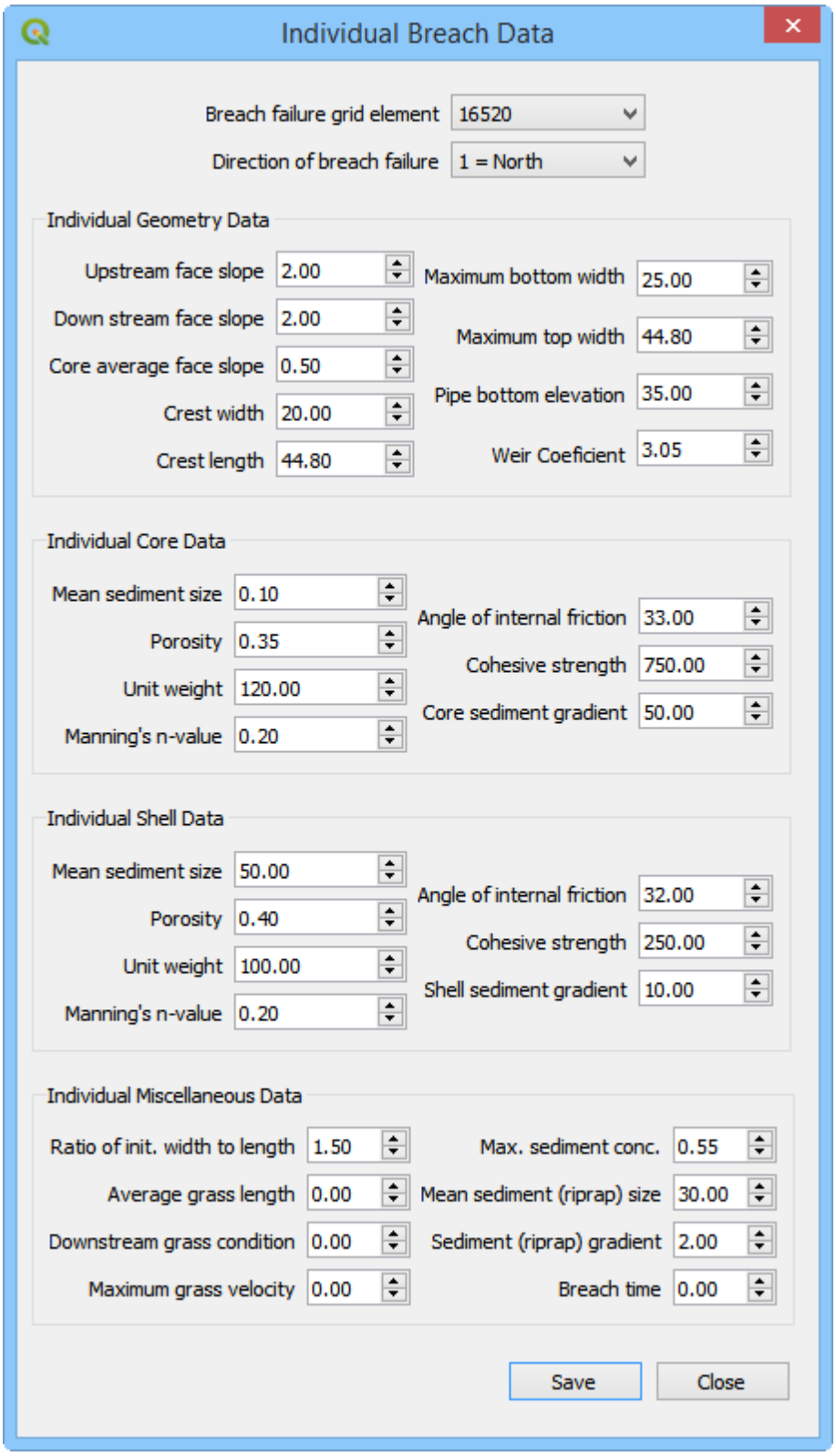


Click the Save icon on the Widget to close the data and load the data tables.

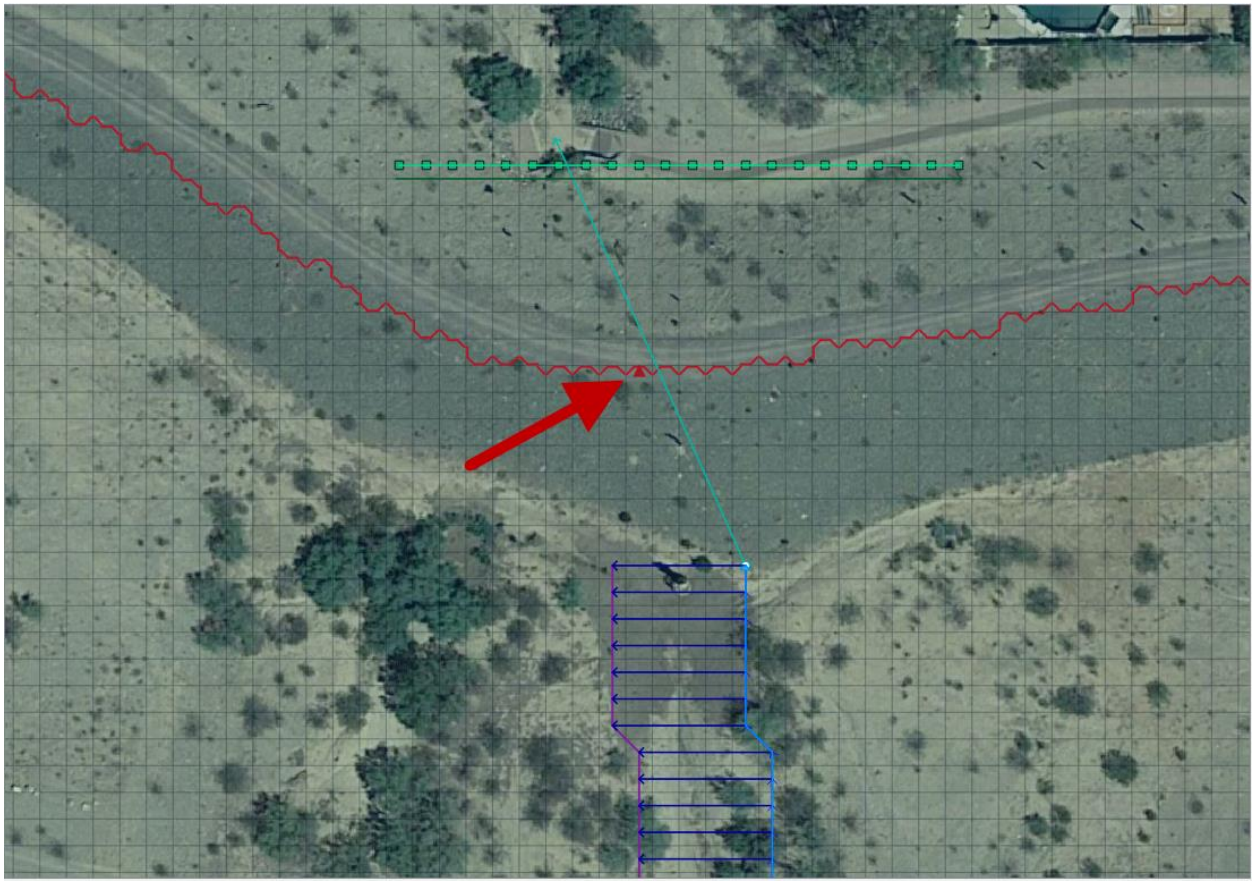
Click the Global Breach Data button and make sure the Use global breach data is unchecked. Click Save to close the dialog box.



Click the Individual Breach Data Button to fill the dam and breach data into the dialog box. ***Note: The breach data for this example is not the real data for this project. Note***

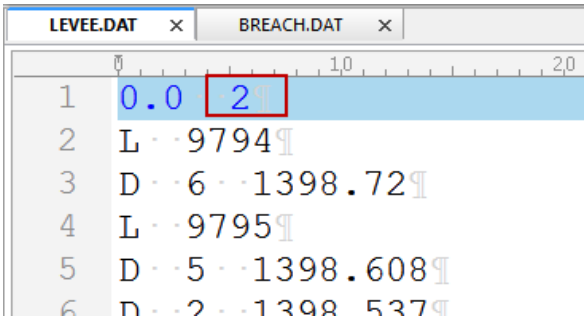


The Breach location is shown with a small red triangle. It is important to place the point in a cell that contains a levee.

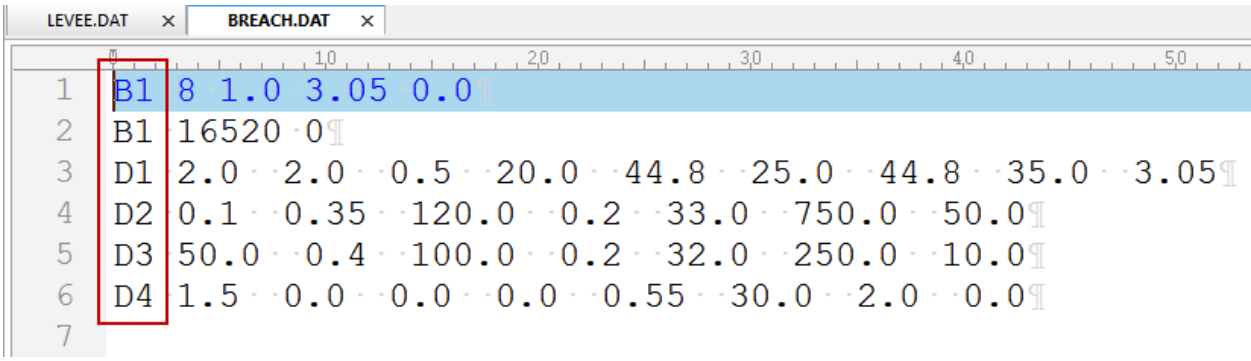


Export the data and check the BREACH.DAT and LEVEE.DAT data files.

LEVEE.DAT should include the Breach Erosion Switch.



BREACH.DAT should have only the B lines and D lines for general and individual breach data.



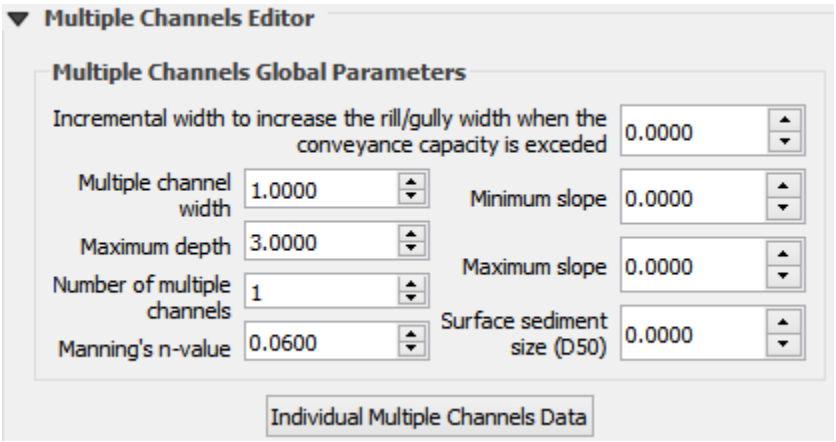
Important notes for Dam Breach Modeling.

1. The cell elevation should be reset to the base floodplain elevation for any cell that represents the breach flow path. See the Elevation Correction section for more details.
2. The breach node should be assigned to a node with a levee and the breach direction should be set so that the breach occurs in the downstream direction.
3. It is also important to choose a flow direction that contains a levee cutoff.
4. The levee crest elevation is used as the dam crest elevation. The base elevation is set by the levee cells where the breach occurs.

Multiple Channels Editor

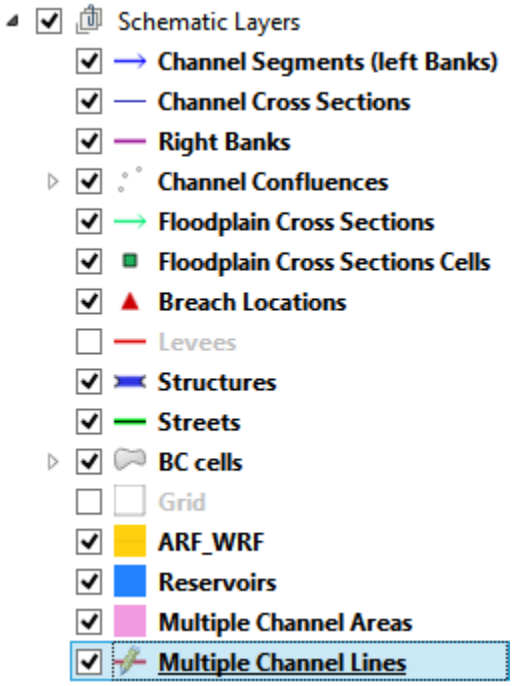
Multiple Channel Editor

The multiple channel editor is used to set up the global parameters for the MULT.DAT file.

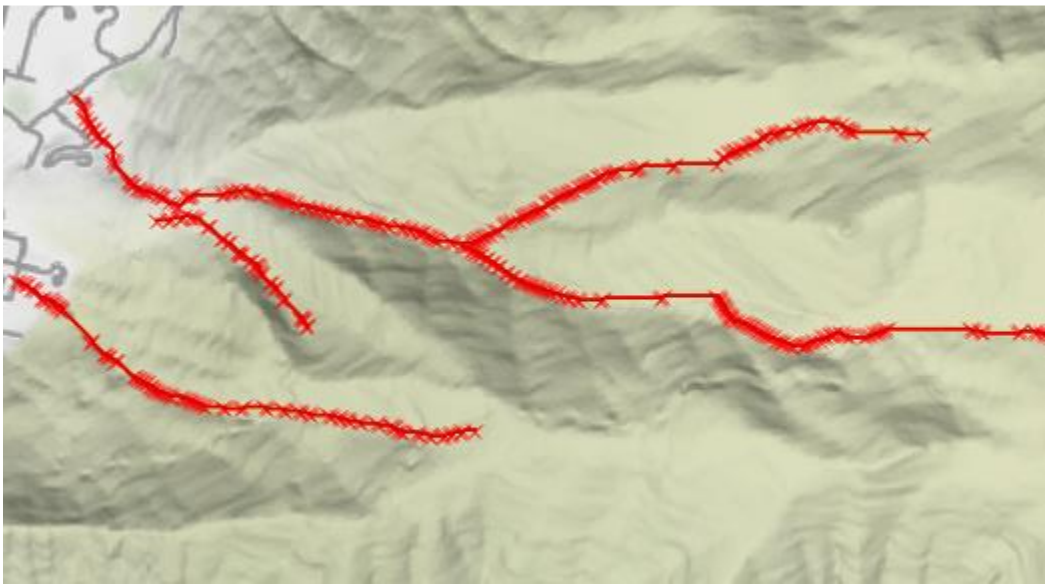


Multiple Channel Lines

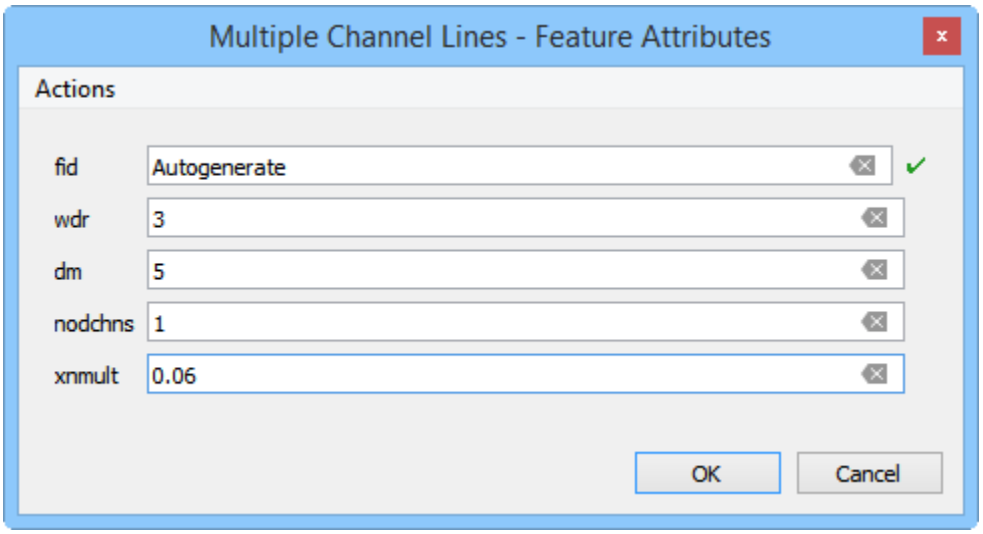
The Multiple Channel Lines layer is used to set the path of the multiple channels. This layer is in the Schematic Layer group. Enter the data using the QGIS vector editor in the toolbar.



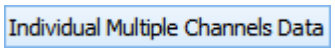
Digitize the drainages, rills or gullies in the project area.



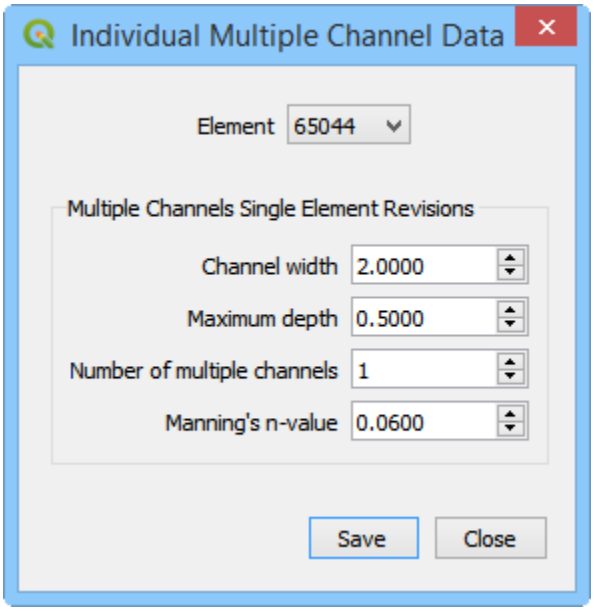
Assign the spatially variable data to each line. The multiple channel lines are split into segments wherever a new channel begins or ends or when new data is required.



Save and close the editor to commit the data to the geopackage. This will automatically write the data to the Multiple Tables and assign the grid elements. It also enables the Individual Multiple Channel Data button.



Individual cells can be edited with this dialog box.



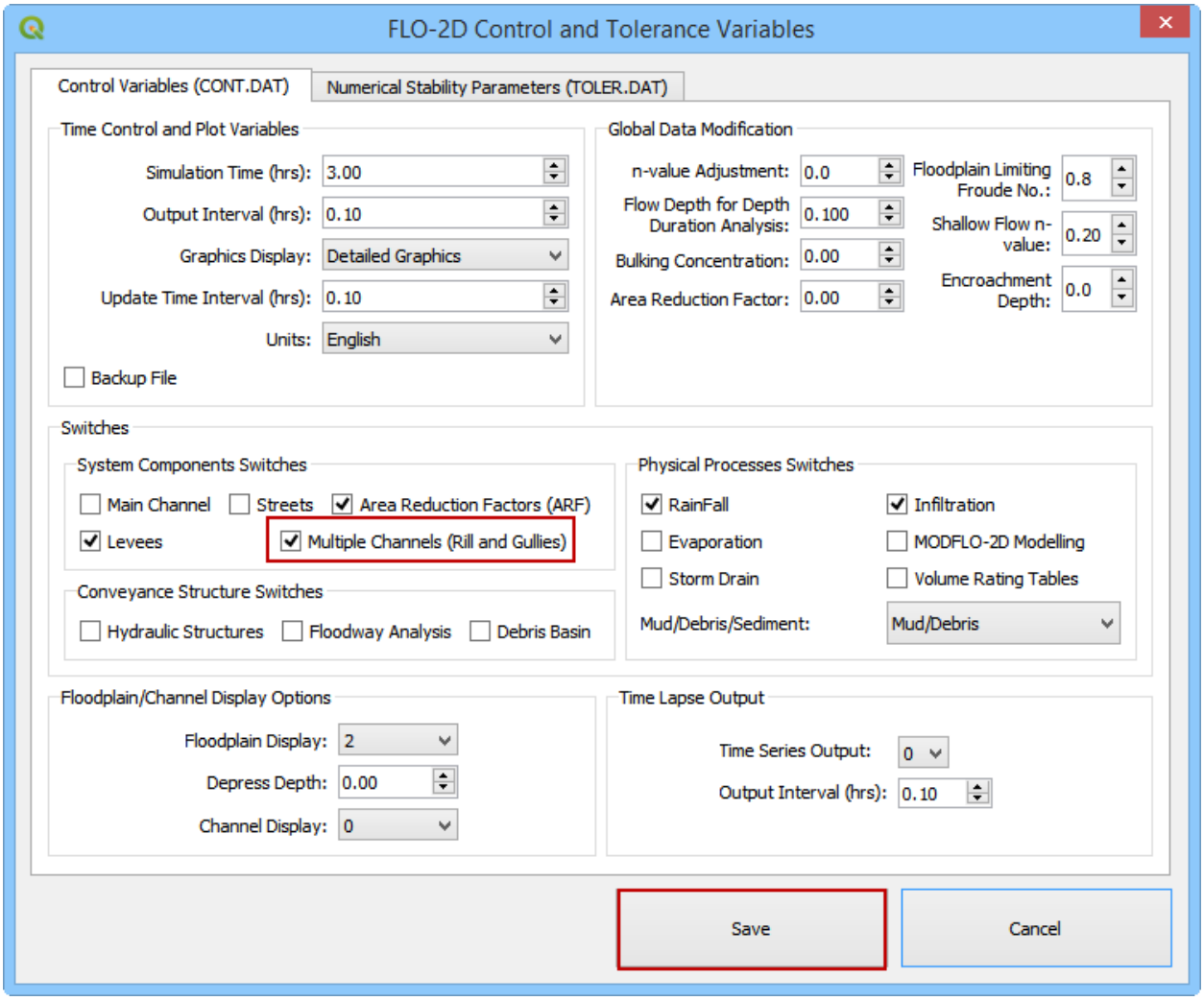
If multiple layer edits are required, they should be made in the Multiple Channel Line layer.

Multiple Channel Areas

When a channel is imported into QGIS using the Import GDS button, the data is written to the Multiple Channel Areas layer as grid element sized polygons.

Export MULT.DAT Files

To export the MULT.DAT file, check the Multiple Channel checkbox and click save. The MULT.DAT file will be written the next time the project data is exported.



Post Development Grid Modification

This section outlines a process to port data after the domain has been changed or the grid elements size has been changed. The process will be outlined based on the available layers and procedures in the QGIS FLO-2D Plugin order.

- Computational Domain Layer
  - Update polygon to cover complete project extent.
  - Set cell size.
- Grid
  - Create the grid.
- Interpolate Elevation
  - Import full coverage raster.
    - If a new area is needed, it is easy to create a mosaic of the two rasters.
    - Export the grid layer as a raster and use a mosaic tool to combine the two rasters.
    - It is a good idea to use the new grid element resolution and alignment as the extent and size for the raster.
  - Sample full coverage raster with the Sampling grid elevation from raster layer.
- Roughness
  - Recalculate from a full coverage shapefile.
  - If shapefile isn't available,
    - Export the old grid and intersect it with roughness polygons for the new grid.
    - Use a Dissolve tool to simplify the shapefile so it combines like polygons with the same n-value.
- Buildings
  - Recalculate from Blocked Areas layer.
  - It may be necessary to add buildings from additional area.
- Extra Grid features
  - Run separate tools for Spatial Tol, Spatial Froude, Spatial Shallow n, and Gutters
- Infiltration
  - Recalculate from external layers.
  - It may be necessary to add infiltration areas to the infiltration shapefile.
    - If extra data is needed, intersect the new data into the Soil and Landuse shapefiles.
- Hydraulic Structures
  - Copy polyline features from Structures Schematic Layer.
  - Paste into Structures User Layer.
  - Run structures schematize tool to refresh data.
  - Add new structures using the editor tool and table editor after refreshing the data.
  - Rerun schematize tool to add new data to final Schematic Layer.
- Levees

- If User Layers Levee Lines are available, use them to recalculate the levees.
- If Levee Lines User Layer is not present use this simple process to copy them from the Schematized Levees. (This doesn't work for walls)
  - Create levee lines using simple polylines along the center of the levee.
  - Make certain these lines cross the Schematic Levees frequently.
  - Intersect the Levee Lines Layer and Levees Layer to generate a point file with crest elevation.
  - Copy that to the Elevation Points Layer.
  - Run the Levee Calc tool with Levee Lines and Points.
- Walls
  - Recalculate walls from wall data.

- Boundary Conditions
  - Run schematize button.
  - The downstream boundary might need to be edited.
- Rain
  - Rerun depth Area Reduction sampler.
- Save and export data.
- Channels
  - Rerun schematize.
  - Run Interpolator.
  - Import interpolated channels.
  - It may be necessary to make left and right bank alignment corrections.
- Floodplain cross sections
  - Rerun schematize.
  - It may be necessary to add new cross sections.
- Storm Drain
  - Add new data to the User Layers and tables if necessary.
  - Rerun storm drain schematizer.
  - Export swmm.inp.
- Export Data
- Run Model

References

FLO-2D Software, 2018, Channel Modeling Guidelines. Nutrioso, AZ.

Library of Congress, 2017, GeoPackage Encoding Standard (OGC), version 1.0, https://www.loc.gov/preservation/digital/formats/fdd/fdd000419.shtml.

Multi-Resolution Land Characteristics Consortium, 2011, National Land Cover Database, https://www.mrlc.gov/nlcd11_data.php.

National Oceanic and Atmospheric Administration, 2017, NOAA Atlas 14 Precipitation Frequency Estimates in GIS Compatible Format, https://hdsc.nws.noaa.gov/hdsc/pfds/pfds_gis.html.

National Oceanic and Atmospheric Administration, 2017, NOAA NEXRAD Archive, https://www.ncdc.noaa.gov/nexradinv/.

United States Department of Agriculture, 2017, Web Soil Survey, https://websoilsurvey.sc.egov.usda.gov/App/HomePage.htm.