OilFlow2D Tutorials

Oil Spill Model

August 2026

Hydronia LLC

OilFlow2D™, RiverFlow2D™ models, and documentation produced by Hydronia, LLC, Pembroke Pines, FL. USA.
Information in this document is subject to change without notice and does not represent a commitment on part of Hydronia, LLC. The software described in this document is furnished under a license agreement.
RiverFlow2D, OilFlow2D, RiverFlow2D, and RiverFlow2D GPU are copyrighted by Hydronia, LLC. 2011-2026.
OilFlow2D™ and RiverFlow2D™ are registered trademarks of Hydronia, LLC.
All other products or service names mentioned herein are trademarks of their respective owners.
All rights reserved. No part of this publication may be reproduced, stored in a retrieval system or transmitted in any form or by any means electronic, mechanical, photocopying, recording or otherwise, without the prior written permission of Hydronia, LLC.
Last document modification date: August 2026.
Technical Support: support@hydronia.com
Web site: www.hydronia.com

Content

List of Figures vi

1 Introduction 1

2 Creating your first river flow application with OilFlow2D 2
- 2.1 Starting a new project . . . 2
- 2.2 Start a new project . . . 5
- 2.3 Load elevation data . . . 8
- 2.4 Create the limits of the modeling area . . . 10
- 2.5 Generating the triangular-cell mesh . . . 12
- 2.6 Setting up the boundary conditions . . . 13
- 2.7 Assigning Manning's n . . . 15
- 2.8 Exporting the files . . . 17

3 Simulating overland oil spills with OilFlow2D 20
- 3.1 Tutorial data . . . 20
- 3.2 Starting QGIS . . . 21
- 3.3 Start a new OilFlow2D project . . . 22
- 3.4 Load elevation data . . . 23
- 3.5 Create the limits of the modeling area (Domain Outline) . . . 25
- 3.6 Generating the triangular-cell mesh . . . 27
- 3.7 Entering the spill data . . . 28
- 3.8 Assigning Manning's n roughness coefficients . . . 30
- 3.9 Setting the boundary conditions . . . 31
- 3.10 Exporting the files to OilFlow2D . . . 32
- 3.11 Creating OilFlow2D maps . . . 37
- 3.12 Utilizing the Heat Transfer option for oil on land spills in OilFlow2D . . . 40
- 3.13 Animating oil spill on land results in OilFlow2D . . . 41

4 Multiple Scenario Project Tool 43
- 4.1 New interface elements for managing multi-scenario projects in OilFlow2D . . . 43
- 4.2 Creating a new OilFlow2D multiple-scenario project using QGIS . . . 44
- 4.3 Create a new scenario . . . 46
- 4.4 Switching scenarios . . . 46
- 4.5 Deleting scenarios . . . 47
- 4.6 Import an OilFlow2D project into Multi-scenario mode . . . 47

5 Simulating Oil Pipeline Breaks **48**
5.1 Tutorial data . . . 48
5.2 Starting QGIS and Creating a new *OilFlow2D* project . . . 49
5.3 Setting up the required layers . . . 51
5.4 Create the Pipeline by adding the *Oilpipeline* layer . . . 54
5.5 Running the *Oil Pipeline Module* within the OilFlow2D plugin . . . 56
5.6 Generating the triangular mesh and exporting to the OilFlow2D model . . . 58
5.7 Configuring parameters in the Hydronia Data Input Program . . . 59
5.8 Creating maps of all sources from results data . . . 62

6 Simulating overland oil spills from a tank break **65**
6.1 Start a new OilFlow2D project . . . 66
6.2 Load elevation data . . . 67
6.3 Create the limits of the modeling area (Domain Outline) . . . 69
6.4 Setting the boundary conditions . . . 71
6.5 Assigning Manning's n roughness coefficients . . . 73
6.6 Using the Weirs layer to create the tank . . . 73
6.7 Generating the triangular-cell mesh . . . 75
6.8 Filling the tank with oil utilizing the *InitialWSE* layer . . . 76
6.9 Exporting the files to OilFlow2D . . . 77
6.10 Creating OilFlow2D maps . . . 82
6.11 Animating oil spill on land results in OilFlow2D . . . 85

7 Oil spills in Rivers. Part I: Hydrodynamics **87**
7.1 Tutorial data . . . 87
7.2 Starting QGIS . . . 88
7.3 Start a new OilFlow2D project . . . 89
7.4 Load elevation data . . . 91
7.5 Create the limits of the modeling area (Domain Outline) . . . 92
7.6 Generating the triangular-cell mesh . . . 94
7.7 Setting the boundary conditions . . . 94
7.8 Assigning Manning's n roughness coefficients . . . 97
7.9 Exporting the files to OilFlow2D and running the model . . . 98
7.10 Creating a velocity field map . . . 102

8 Oil spills in Rivers. Part II: Oil Spill **104**
8.1 Opening the project with results obtained in Part I . . . 104
8.2 Create the OilSpills layer and enter the spill location and data . . . 105
8.3 Export Oil Spill Data . . . 108
8.4 Enter additional data in the Hydronia Data Input Program and run OilFlow2D . . . 110
8.5 Creating a Oil Spill Trajectory Animation . . . 113

9 Oil spills in Coastal Areas. Part I: Coastal Hydrodynamics **116**
9.1 Tutorial data . . . 116
9.2 Starting QGIS . . . 117
9.3 Start a new OilFlow2D project . . . 117

9.4 Load elevation data . . . 119
9.5 Create the limits of the modeling area (Domain Outline) . . . 121
9.6 Generating the triangular-cell mesh . . . 122
9.7 Setting the boundary conditions . . . 123
9.8 Assigning Manning's n roughness coefficients . . . 125
9.9 Exporting the files to OilFlow2D and running the model . . . 127
9.10 Creating an Animation of the Velocity Field in the Bay . . . 131

10 Oil spills in Coastal Areas. Part II: Oil Spill 133
10.1 Opening the project with results obtained in Part I . . . 133
10.2 Create the OilSpills layer and enter the spill location and data . . . 135
10.3 Export Oil Spill Data . . . 137
10.4 Enter additional data in the Hydronia Data Input Program and run OilFlow2D . . . 139
10.5 Creating a Oil Spill Trajectory Animation . . . 144

11 Wind driven circulation 147
11.1 Open an existing project . . . 147
11.2 Wind velocity time-series data file . . . 148
11.3 Create the template for the wind layer and the wind speed polygons . . . 149
11.4 Generate the mesh . . . 152
11.5 Exporting files . . . 153
11.6 Running the model . . . 154
11.7 Check the wind output files . . . 157

12 Using Manning's n ESRI shape files 159
12.1 Open an existing project . . . 159
12.2 Load the shape file with the Manning's n polygons . . . 160
12.3 Import the Manning's n geometry and values to the Manning N layer . . . 161

13 Post-processing calculations 165
13.1 Open an existing project . . . 165
13.2 Create a template of the layers *ObservationPoints*, *CrossSections* and *Profiles* and draw the output controls . . . 166
13.3 Generate the mesh . . . 171
13.3.1 Exporting files to OilFlow2D . . . 172
13.4 Running the Model . . . 174
13.5 Review the output files . . . 175

14 Advanced digitization/snapping 179
14.1 Open an existing project . . . 180
14.2 Activate and configure the Snapping tool . . . 181
14.3 Draw contiguous or adjacent polygons using the Snapping tool . . . 181
14.4 Delete a polygon . . . 187

15 Creating raster elevations from X Y Z data sets 189

16 Setting up a Google Cloud VM for GPU-Accelerated Simulations 197

16.1 Before you begin . . . 197
16.1.1 Checking and Requesting GPU Quotas . . . 198
16.2 Creating the VM Instance . . . 198
16.3 Configuring Machine and GPU . . . 199
16.4 Configuring the Boot Disk and Firewall . . . 200
16.5 Create the VM and Handle Quota Requests . . . 201
16.6 Connect to the VM . . . 204

17 Creating high-impact graphics and animations using Paraview 205
17.1 Paraview basics . . . 205
17.2 Two-dimensional (2D) visualizations . . . 207
17.2.1 Create a 2D bed elevation map . . . 207
17.2.2 Creating 2D velocity vector fields . . . 209
17.3 Three-dimensional (3D) visualizations . . . 213
17.3.1 Create a 3D water elevation graphic adding a (h+z) layer . . . 218
17.3.2 Create a velocity field graphic . . . 220
17.4 Generating animations . . . 222
17.5 Streamlines representation . . . 223

List of Figures

2.1 Files with data required for the example. 2
2.2 QGIS interface indicating window areas. 3
2.3 Plugins window showing standalone OilFlow2D activated (pre-v10/standalone). . . 4
2.4 QGIS Plugins window showing Hydronia2DMS installed for version 10 and later. . 5
2.5 Domain Outline layer. 11
2.6 Resulting mesh. 12
2.7 Message panel of the registry with GMSH messages. 12

3.1 Files with data required for the example. 21
3.2 QGIS interface for OilFlow2D indicating window areas. 1: Menu bar, 2: Tool bar, 3: Layer panel, 4: Maps view area, 5: Status bar. 21
3.3 Domain Outline layer. 27
3.4 Mesh generated. 28
3.5 Polygon that covers the nodes defining the Inflow boundary condition segment. . . 31
3.6 Oil Properties vs. Time Maps menu button. 41
3.7 Output map options for Oil Properties vs. Time Maps. 41

4.1 OilFlow2D toolbar with the scenario list 43
4.2 OilFlow2D New Scenario menu item 43
4.3 OilFlow2D Import Single to MultiScenario Project menu item 44

5.1 Files with data required in this tutorial. 49

6.1 Files with data required in this tutorial. 66
6.2 Domain Outline layer. 71
6.3 Boundary-condition polygon at the lower end of the mesh. 72
6.4 Open DIP dialog showing QGIS DIP selected by default and More options expanded with Classic DIP. 78
6.5 QGIS DIP Control Data panel for Simulating Tank Breaks. 79
6.6 Legacy configuration reference for the Hydronia Data Input Program. 79

7.1 Files with data required in this tutorial. 88
7.2 QGIS interface for OilFlow2D indicating window areas. 1: Menu bar, 2: Tool bar, 3: Layer panel, 4: Maps view area, 5: Status bar. 88
7.3 Raster data import dialog. 91
7.4 Mesh generated. 94

8.1 Resulting project from the tutorial Oil Spill in Rivers Part I 105

9.1 Files with data required in this tutorial. . . . 116
9.2 QGIS interface for OilFlow2D indicating window areas. 1: Menu bar, 2: Tool bar, 3: Layer panel, 4: Maps view area, 5: Status bar. . . . 117
9.3 Raster data import dialog. . . . 119
9.4 Digital elevation model in raster format created by interpolation. . . . 120
9.5 Domain Outline layer. . . . 122
9.6 Mesh generated. . . . 123

10.1 Resulting project from the tutorial Oil Spill in Coastal Areas Part I . . . 134

11.1 Model window. . . . 157
11.2 Map with the speed field for the time 20 hours. . . . 158

12.1 Attribute table of the Manning N layer. . . . 164

13.1 Legacy Classic DIP Control Data panel. . . . 174

14.1 Areas with overlap and empty spaces in manual digitization of adjacent polygons. . 179
14.2 Project screen loaded in QGIS. . . . 180
14.3 Snapping configuration panel. . . . 181

15.1 File containing terrain elevation points. . . . 190
15.2 Window to change the render style of a raster layer. . . . 195
15.3 Digital elevation model with color render. . . . 196

16.1 Configuring the machine with a custom N1 type, 4 vCPUs, 32 GB of memory, and one NVIDIA V100 GPU. . . . 200

17.1 Main Paraview window. . . . 206
17.2 View after opening the 'BridgeTutorial..vtk' file. . . . 206
17.3 VCR and Time Control toolbars. . . . 207
17.4 View for Time = 2. . . . 207
17.5 Variable selector. . . . 208
17.6 Bed elevation z representation. . . . 208
17.7 Predefined color maps. . . . 209
17.8 Bed elevation z representation. . . . 209
17.9 XRay Color Map. . . . 210
17.10 White-to-blue color map. . . . 210
17.11 White-to-blue color map. . . . 211
17.12 Water depth (h). . . . 211
17.13 Velocity vector field on water depth. . . . 212
17.14 Velocity vector field detail. . . . 213
17.15 Bed elevation z. . . . 214
17.16 Viewpoint Toolbar. . . . 217
17.17 Alternative 3D bed elevation representation. . . . 218
17.18 Camera Control Toolbar. . . . 218
17.19 Water surface elevation representation. . . . 219

17.20 Threshold Filter application. 220
17.21 Three dimensional water depth h representation. 220
17.22 Three-dimensional velocity field representation. 222
17.23 Animation. 222
17.24 StreamLines representation. 224

1 Introduction

The main purpose of these tutorials is to assist model users in performing hands-on applications of the OilFlow2D model using QGIS.

OilFlow2D is a two-dimensional model that simulates spreading of crude oils and other viscous fluids on the land surface, and trajectory and fate of crude oils on water. The oil spill on land module can simulate oil depths and velocity over irregular surfaces considering fluid properties such as density, viscosity that can vary in time. The model can use 3 different rheogological formulations for Newtonian and non-Newtonian fluids that can have yield stress. The OilFlow2D model also accounts for oil evaporation and that process affects density and viscosity and its subsequent impact on fluid flow. The oil spill on water module uses a particle-tracking approach to simulate oil trajectories of oil spill, and includes oil fate algorithms to account for evaporation, emulsification, dissolution, dispersion, and interaction with shores.

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

2

Creating your first river flow application with OilFlow2D

This section provides a step-by-step guide to help you get started with a OilFlow2D project using the QGIS interface. The example illustrates the model application to simulate flow in a river with a single inflow upstream and a single outflow downstream. It includes instructions to enter the terrain elevation data, create the mesh, prepare the layers with the input information, and run OilFlow2D.

2.1 Starting a new project

The files required to follow this tutorial can be extracted from the 'ExampleProjects' zip file under the 'Hoh_QGIS_Metric_Units' folder. This zip file is downloaded separately from your installation materials.

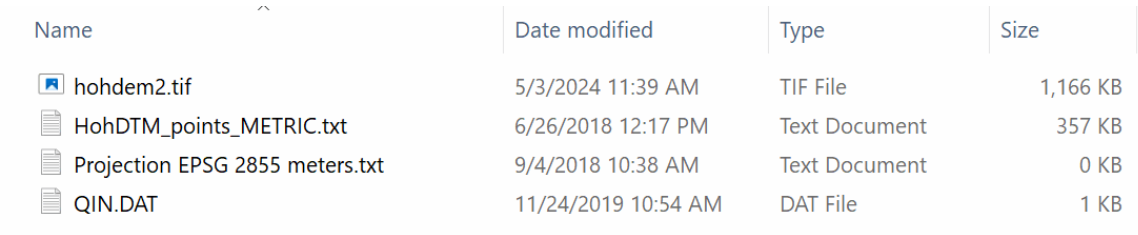


Figure 2.1 – Files with data required for the example.

The first step is to start the QGIS software clicking the QGIS desktop icon 

. If this icon is not available, you can run the 'qgis-bin.exe' executable on the QGIS 'bin' subdirectory. After loading, you will see a window similar to the one shown below:

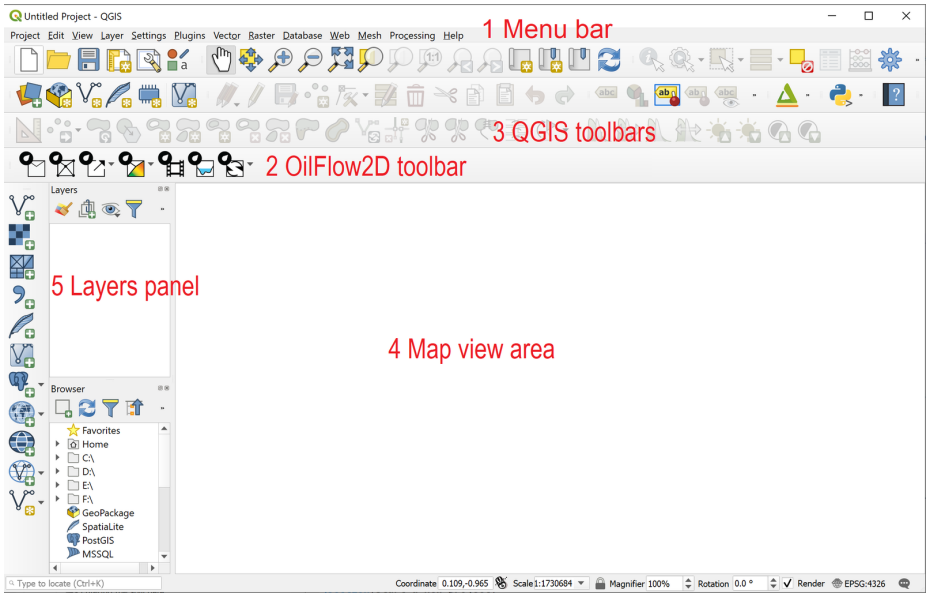


Figure 2.2 – QGIS interface indicating window areas.

For pre-v10 or standalone OilFlow2D installations, if you don't see the toolbar with the model icons as shown, activate the standalone plugin using the *Manage and Install Plugins…* command under the *Plugins* menu.

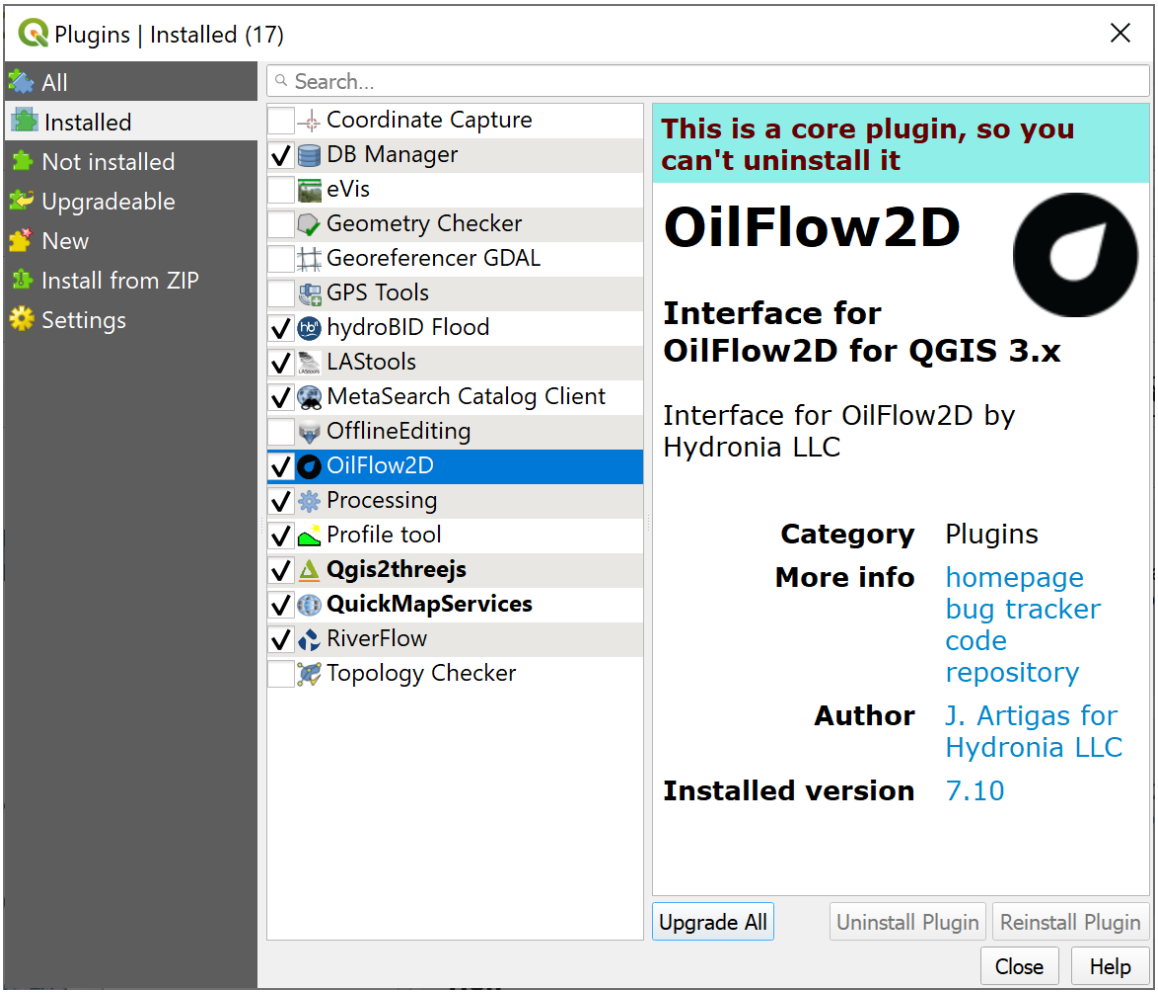


Figure 2.3 – Plugins window showing standalone OilFlow2D activated (pre-v10/standalone).

For version 10 and later, use the unified Hydronia2DMS plugin. In *Manage and Install Plugins…*, search the *Installed* list for Hydronia2DMS, confirm it is enabled, and verify that the OilFlow2D-flavored toolbar is visible before continuing. Hydronia2DMS exposes the OilFlow2D-flavored toolbar.

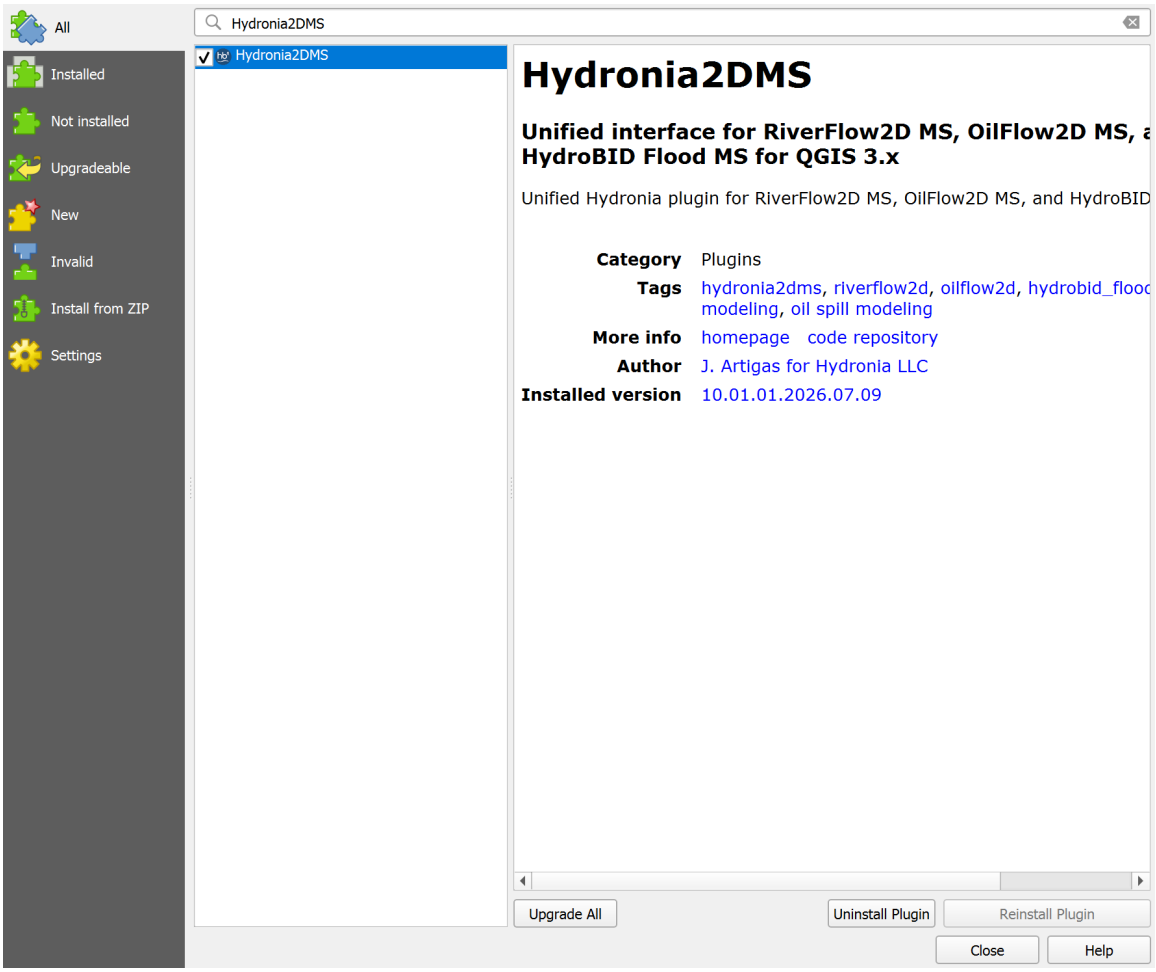


Figure 2.4 – QGIS Plugins window showing Hydronia2DMS installed for version 10 and later.

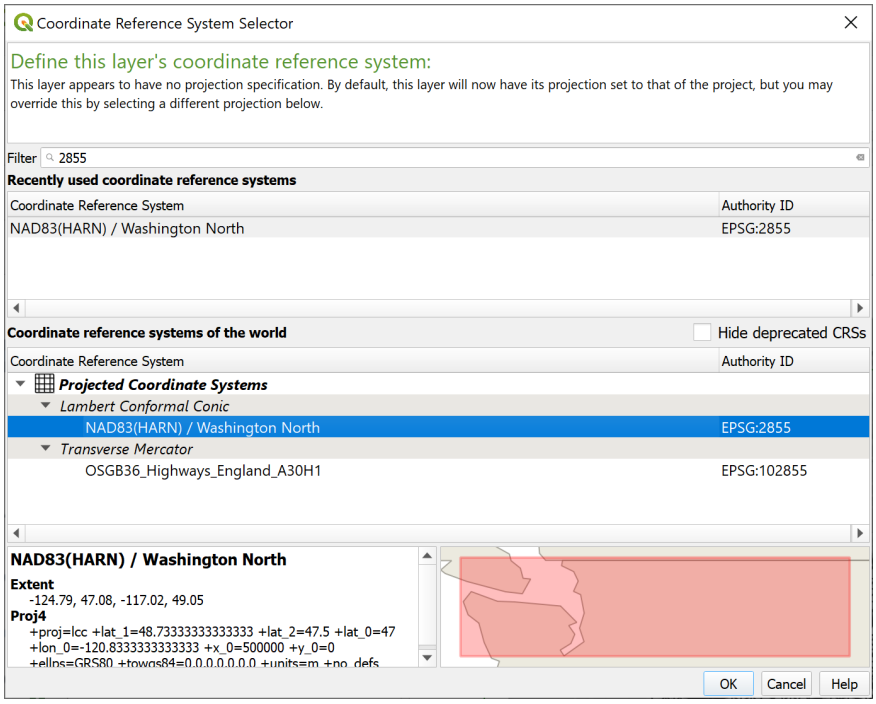
2.2 Start a new project



1. To create a new OilFlow2D project, click on the *New OilFlow2D Project* button in the toolbar. A dialog window appears where you select the optional component layers, the Coordinate Reference System (CRS), and the project directory.

2. Select *None* in the Layers drop down menu.

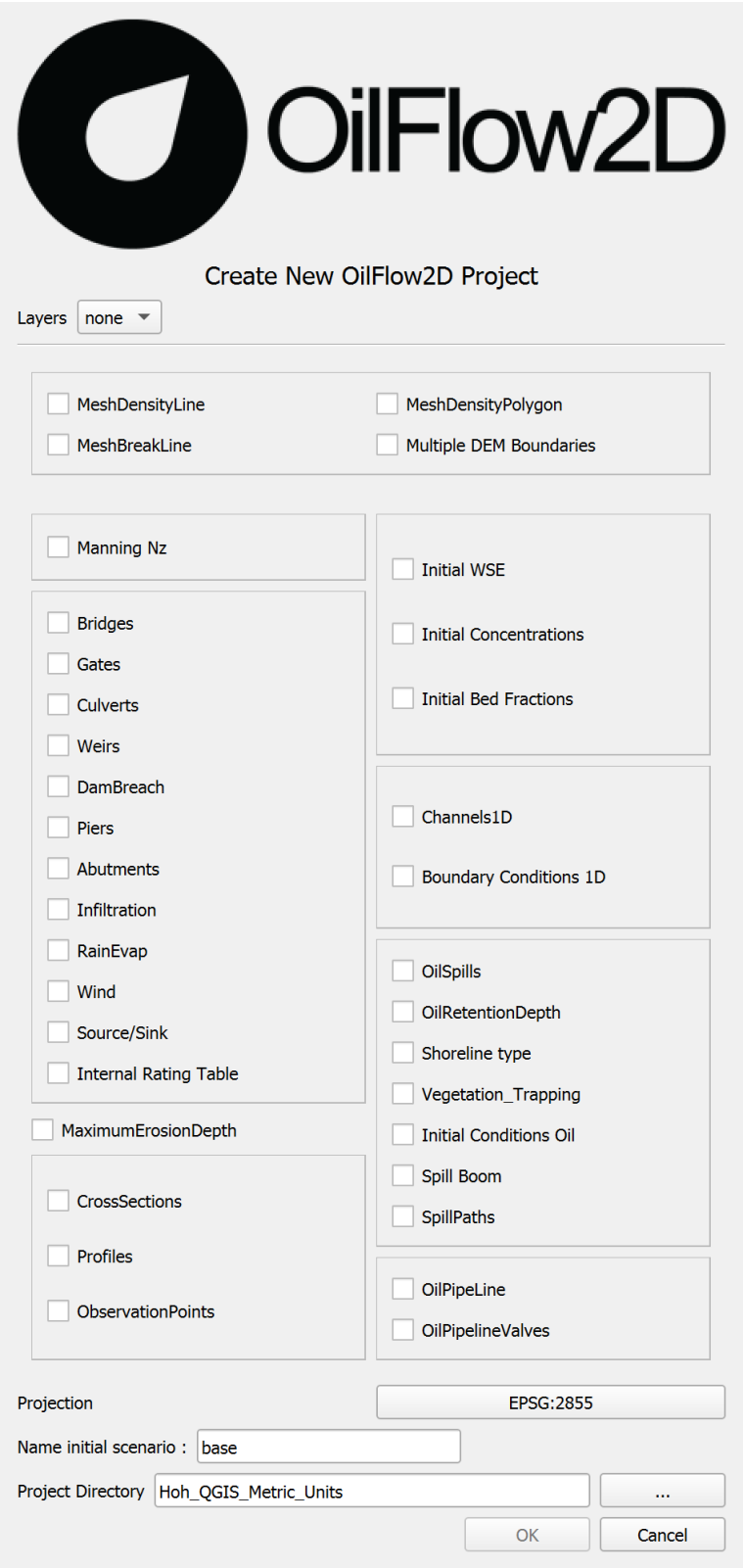
3. Select the *Projection* button. In the *Filter* textbox, type *2855* and select the *Coordinate Reference System* as shown:



Coordinate Reference System Selector dialog window.

4. Click OK.

5. After confirming the CRS selection, the Projection button displays EPSG:2855, corresponding to the NAD83(HARN)/Washington North Coordinate Reference System (CRS):

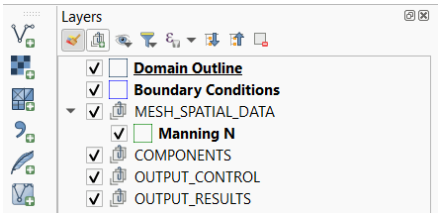


Create New OilFlow2D Project configured for Hoh_QGIS_Metric_Units.

6. Keep *Name initial scenario* set to base. Click the  button and choose the Hoh_QGIS_Metric_Units folder for the *Project Directory*. This is the folder where the model will write all results and output files.

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

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



Layers created for the project.

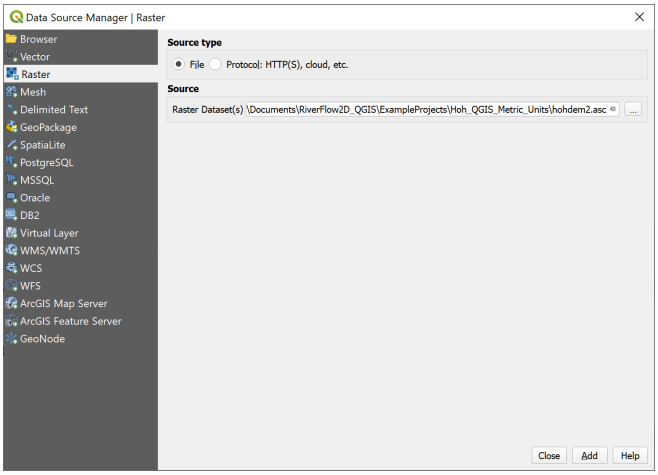
8. On the QGIS *Project* menu, click *Save*, to save the project in the same directory that you previously selected in the *Create New Project* dialog above.

2.3 Load elevation data

In this tutorial we will use a raster file that contains the terrain and river bed bottom elevation data in ASCII grid format.

1. To load an ASCII grid file, click the *Add Raster Layer* button  .

2. In the dialog search for the tutorial folder and select the 'hohdem2.tif' file as shown:

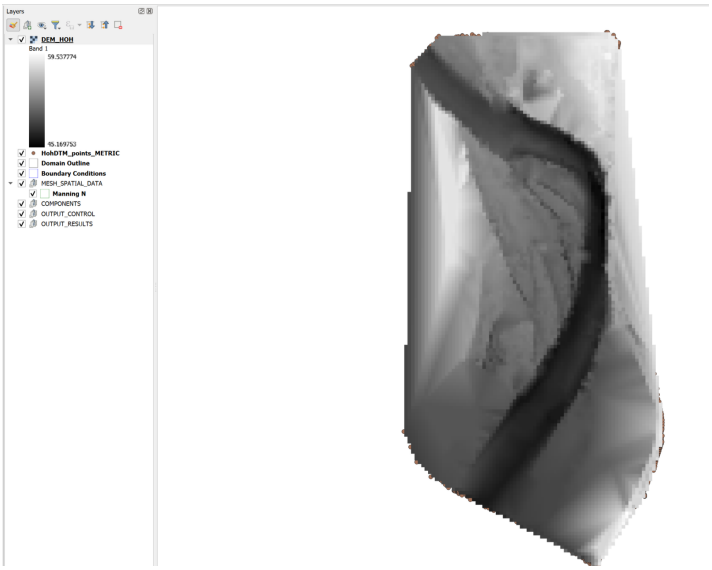


Dialog to create a layer from a raster file.

3. While on the dialog, click *Add* and click *Close*.

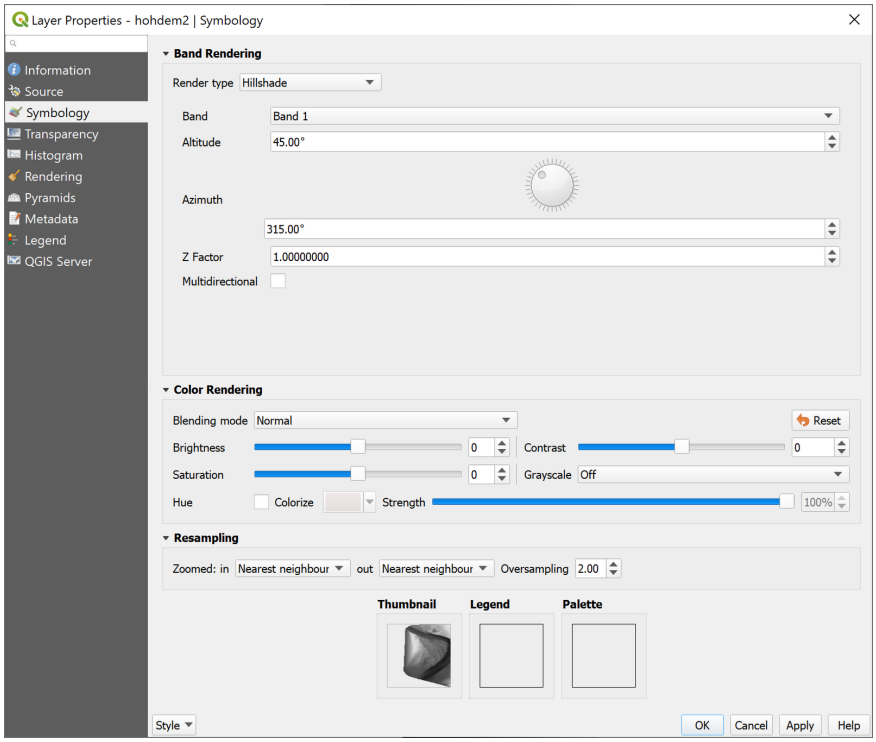
4. Use the *Zoom to Layer* button  to center the image.

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



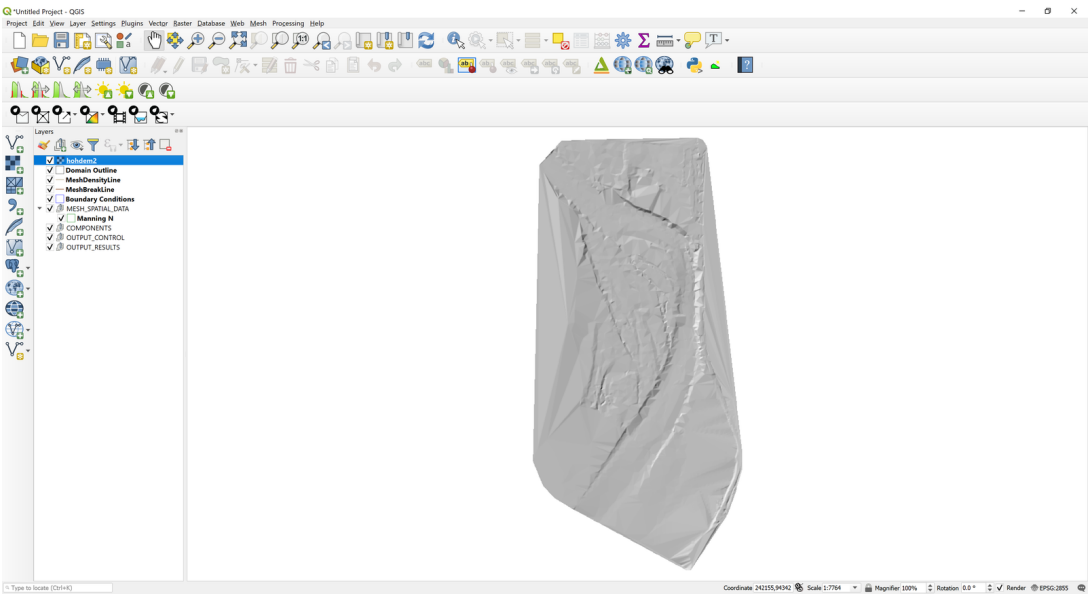
Digital elevation model in raster format.

Right-clicking on the label of the new raster layer and selecting *Properties* allows you to change the rendering style for a more informative palette such as *Hillshade* for instance.



Window to change the raster layer render style.

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



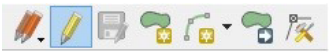
Digital elevation model with Hillshade render.

5. You should move the raster layer dragging it to the end of the list of layers to avoid that it would hide or interfere visually with the other layers.

2.4 Create the limits of the modeling area

We define the limits of the modeling area drawing a polygon on the *Domain Outline* layer. To create it do as follows:

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

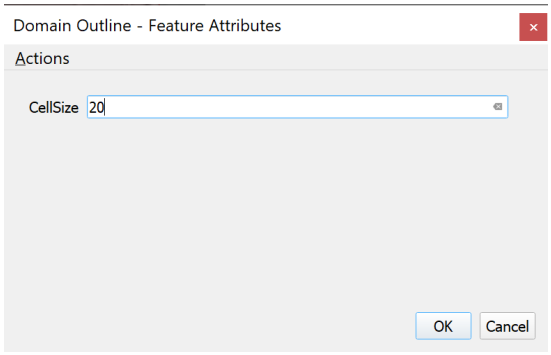


2. This activates the rest of the editing buttons. Now click the *Add Feature* tool which is the bean-looking polygon  .

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

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

3. To finalize and close the polygon, right-click on the map view area. A dialog window to input the cell size attribute of the newly created polygon will appear. The *CellSize* value for the reference size of the mesh cell is indicated. Enter a value of 20 m.



CellSize defined for the Domain Outline layer.

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

4. Save the polygon by clicking the *Save* button  .

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

   The *Domain Outline* is now complete.



Figure 2.5 – Domain Outline layer.

2.5 Generating the triangular-cell mesh

Now that the *Domain Outline* layer has been created, proceed to create the mesh by clicking on the *Generate Trimesh* button

The following figure shows the generated mesh. You will also see in the Layers panel the new layer: *Trimesh*

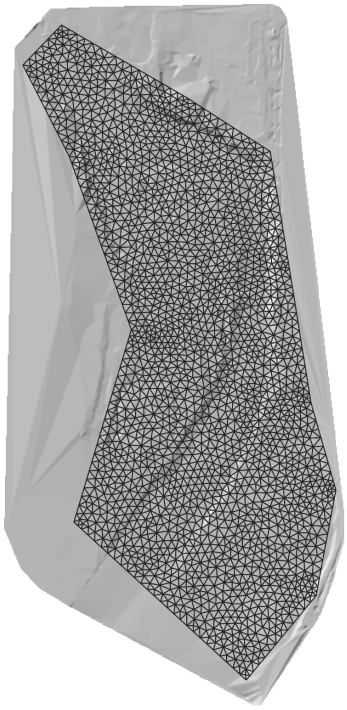


Figure 2.6 – Resulting mesh.

You can see the mesh generation statistics, and other messages produced by the mesh generation program while creating the mesh in the Log messages panel. This window is accessed from the *View* menu, then by clicking *Panels*.

Log Messages Panel

General | Plugins | Processing | Python warning | Gmsh

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

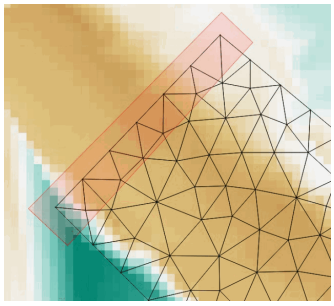
Figure 2.7 – Message panel of the registry with GMSH messages.

2.6 Setting up the boundary conditions

Inflow boundary conditions:

1. Select the *BoundaryConditions* layer in the Layers panel.

2. Click the *Toggle Editing* button  to add the polygons that will indicate the open boundary segments where inflow and outflow conditions are imposed. Draw a polygon at the upper end of the mesh as indicated in the figure:



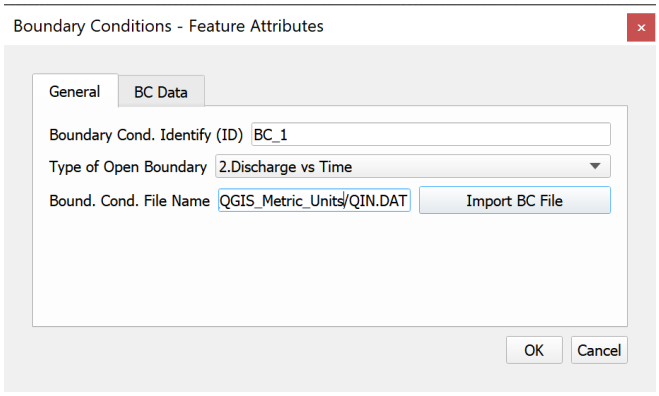
Polygon that covers the nodes defining the Inflow boundary condition segment.

3. To finish the polygon, right-click on desired location. A window to enter the attributes of the newly created polygon is displayed.

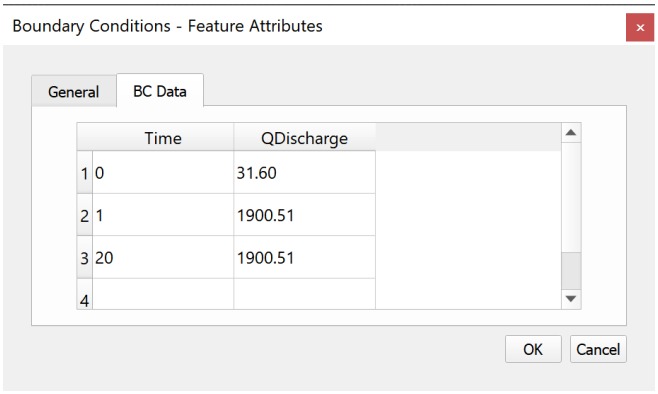
4. In the *Boundary Cond. ID* enter the desired name or leave the default.

5. From *Type of Open Boundary* list, select *2. Discharge vs. Time*

6. Click *Import BC File* button, and search for the 'QIN.DAT' hydrograph file as shown below:



Inflow boundary condition parameters.

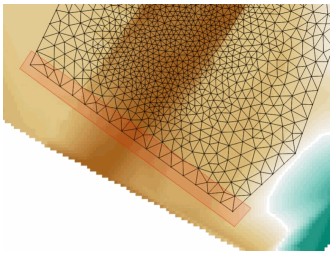


Hydrograph loaded from the 'QIN.DAT' file.

7. Click *OK* to close the dialog and then click *Save*  .

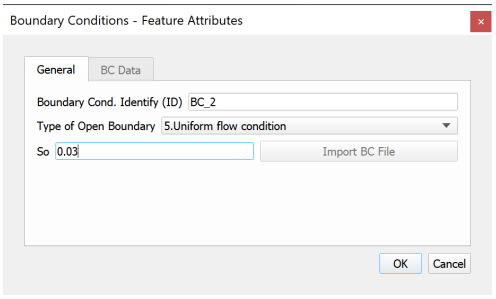
Outflow boundary conditions:

1. Draw the polygon defining the outflow boundary condition at the downstream end of the channel as shown.



Polygon that defines the outflow boundary condition segment.

2. Right click to close the polygon. A dialog window will appear to enter the parameters. Select the condition type *Uniform flow conditions* and enter the channel slope. Slope is entered in *So* as shown:

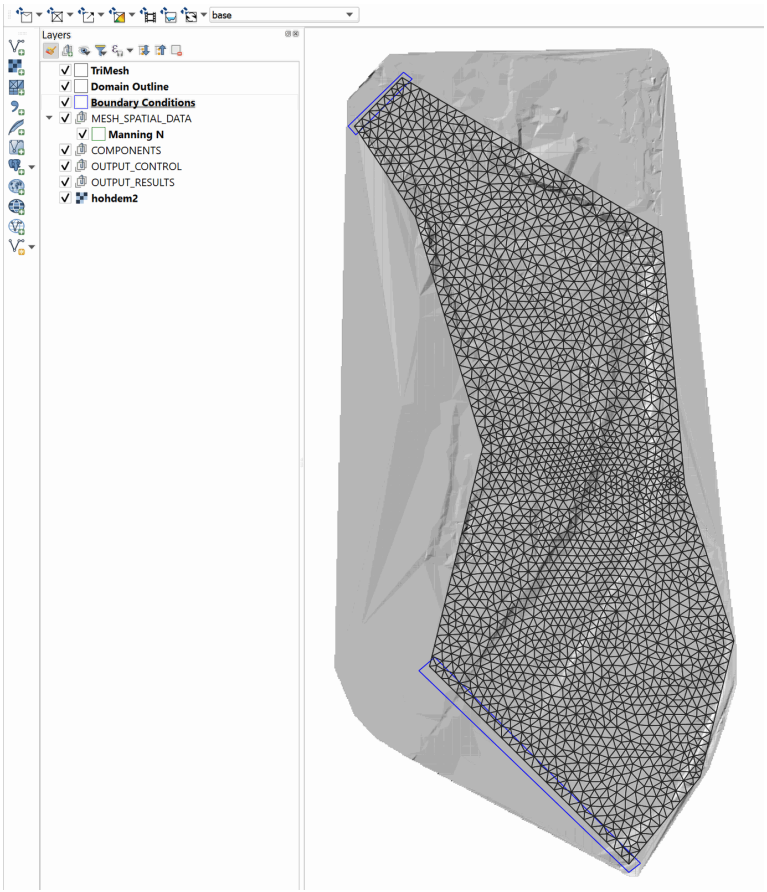


Parameters for the uniform flow outflow open boundary condition.

3. Save the changes made to the layer by clicking the *Save* button  .

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

   The figure below shows how the *BoundaryConditions* layer should look:



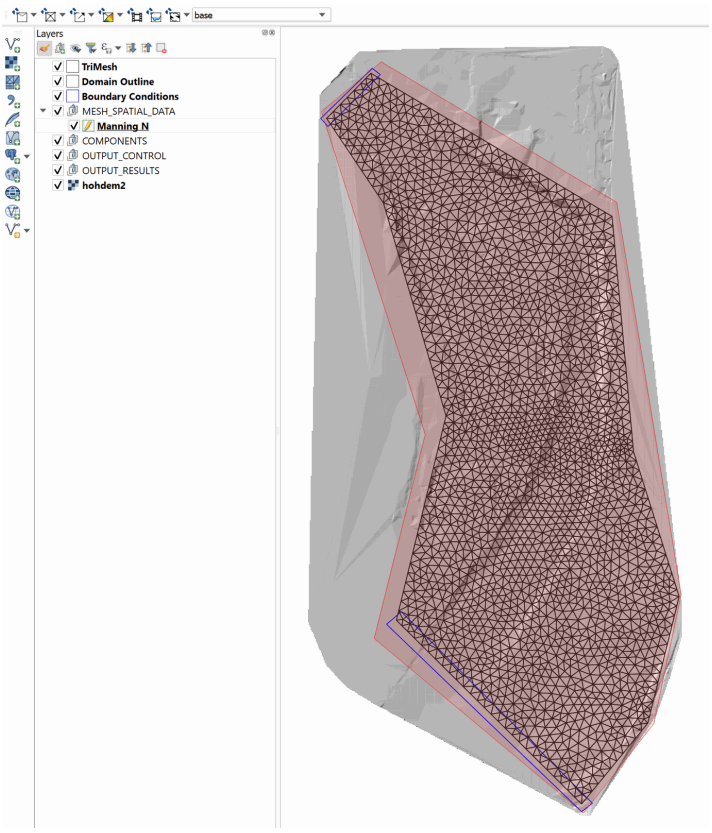
Polygons that define the inflow and outflow boundary conditions.

2.7 Assigning Manning's n

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

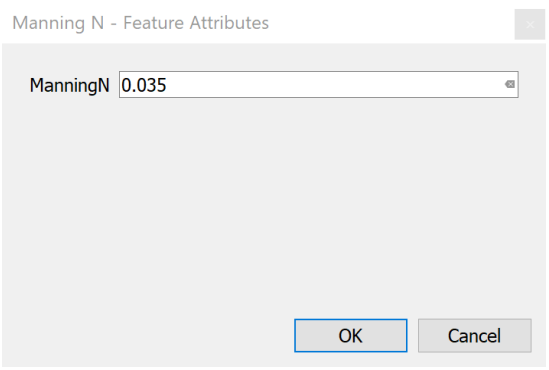
1. Select the *Manning N* layer and click the *Toggle Editing* button

2. Draw a polygon that covers the entire domain. The polygon may extend beyond the mesh area as shown:



Capa Manning N.

3. Close the polygon by right-clicking on the end vertex and enter a Manning's n equal to 0.035:



Diálogo para ingresar ManningN.

4. Click *Save*  , and then click the *Toggle Editing* button  to deactivate editing mode.

Save the QGIS project using the *Save* command in the *Project* menu. Name the project file 'Hoh.qgs'.

2.8 Exporting the files

Once the layers with the input data to the model have been created, we need to export data files required to run OilFlow2D.

1. Activate the DEM layer.

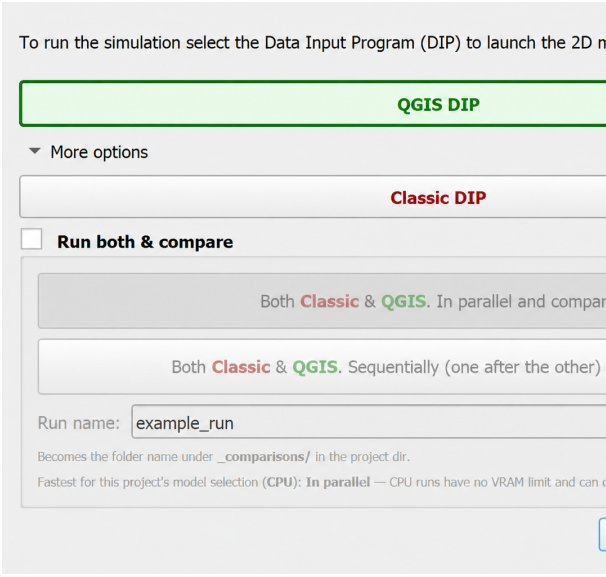
2. Open Plugins > OilFlow2D MS > Export OilFlow2D and click the exact visible action *Export Hydrodinamic OilFlow2D*.

3. In the *Export Files to OilFlow2D* dialog, keep *Scenario Name* set to base and choose DEM in the *DEM (Single Raster)* field.

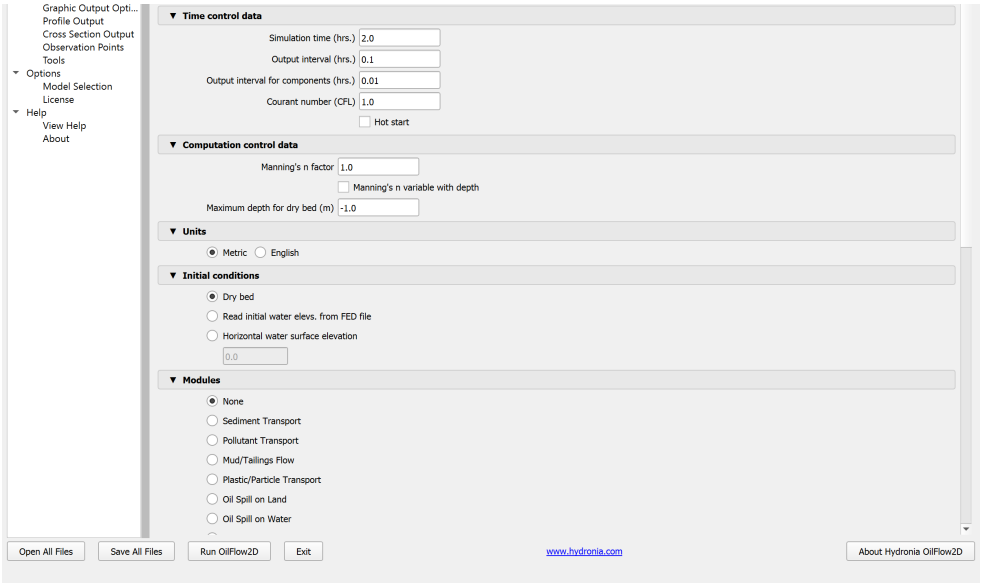
   The Scenario Name is also used for the exported filenames; do not change those filenames manually.

4. Click *OK*. A message at the top of the Map area shows the progress of the export process.

   After export completes, version 10 and later users see the Open DIP dialog. QGIS DIP is selected by default; click OK to open it inside QGIS. To use the historical standalone option, expand More options, select Classic DIP, and then click OK.

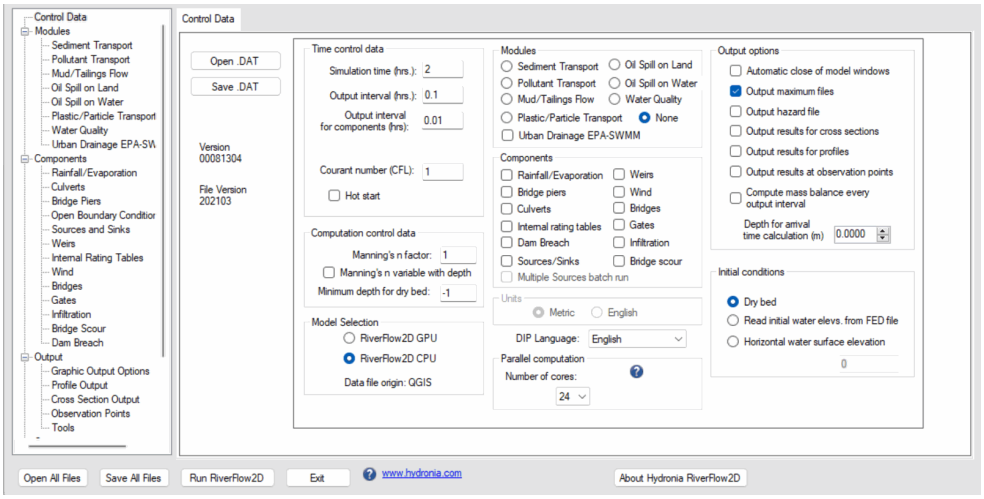


   With QGIS DIP selected, the current interface is shown below:



QGIS DIP interface for OilFlow2D.

QGIS DIP and Classic DIP use the same project settings, but their interfaces look different. Older standalone or non-maintenance versions may open Classic DIP automatically. The project-specific Classic DIP settings and Run steps below are retained.

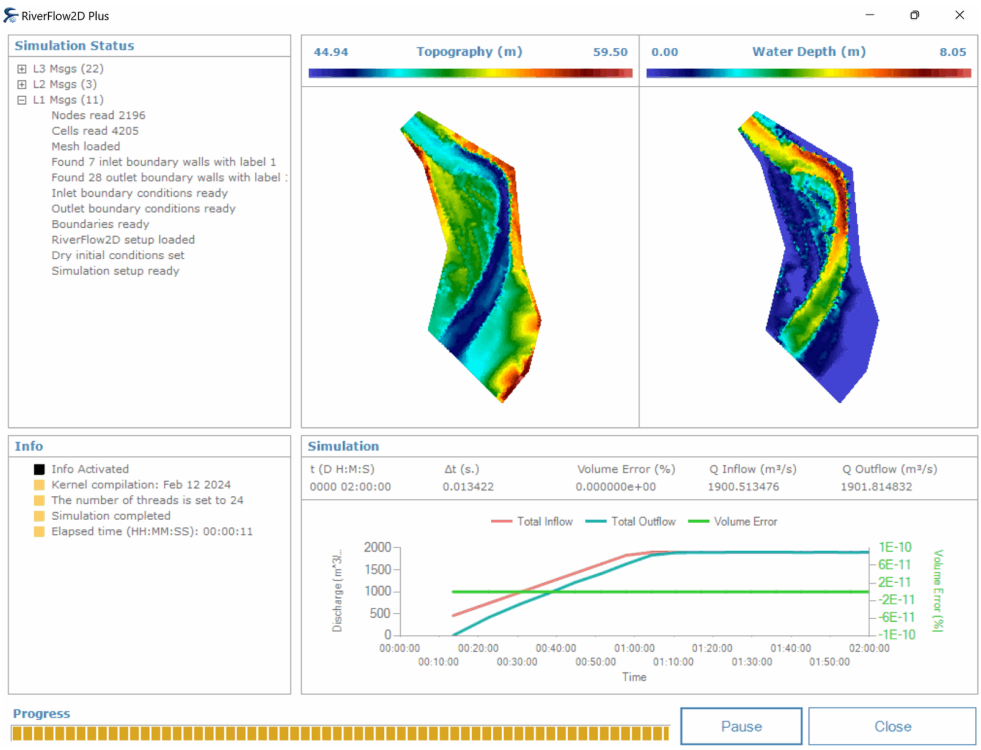


Hydronia Data Input Program window.

5. Click the *Run OilFlow2D* button to run the model.

6. A dialog box will ask if you want to save changes. No changes were made, so select [No].

   An image similar to the one shown below should appear:



Window displayed while the model runs.

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

This concludes the *Creating your first OilFlow2D application* tutorial.

3

Simulating overland oil spills with OilFlow2D

This section provides a step-by-step guide for setting up an overland oil spill using OilFlow2D. The tutorial illustrates the model application to simulate a real spill from a pipeline leak. It includes instructions to enter the terrain elevation data, create the mesh, prepare the layers with the spill location, and run the OilFlow2D model.

3.1 Tutorial data

The files required to follow this tutorial can be extracted from the 'ExampleProjects' zip file under the 'OilSpillOnLandTutorial' folder. This zip file is downloaded separately from your installation materials. The folder contains the files shown:

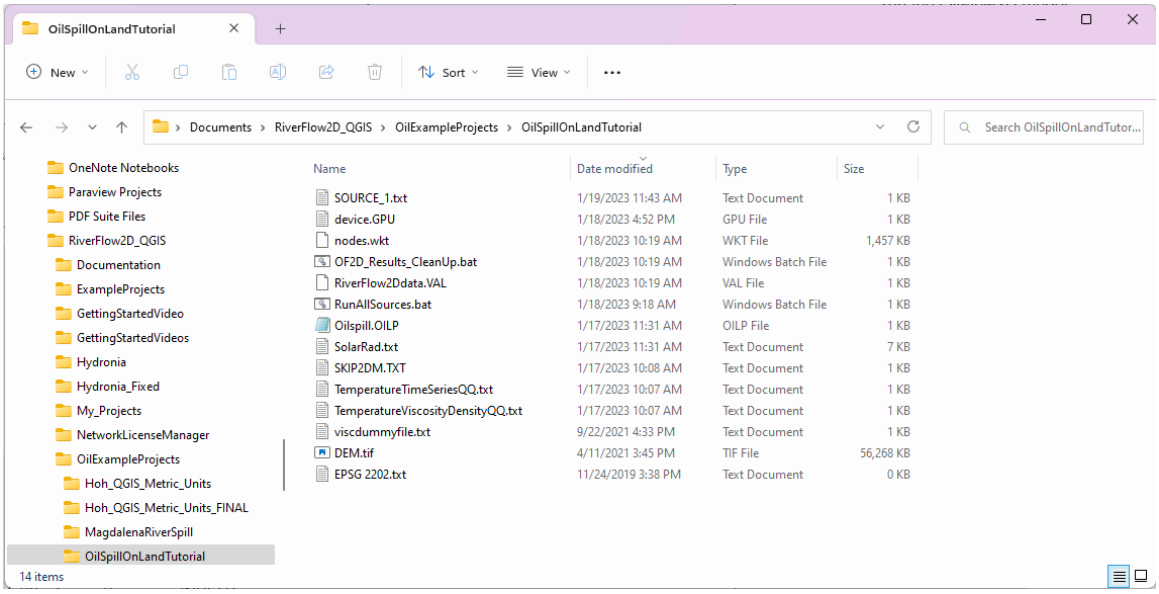


Figure 3.1 – Files with data required for the example.

3.2 Starting QGIS

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

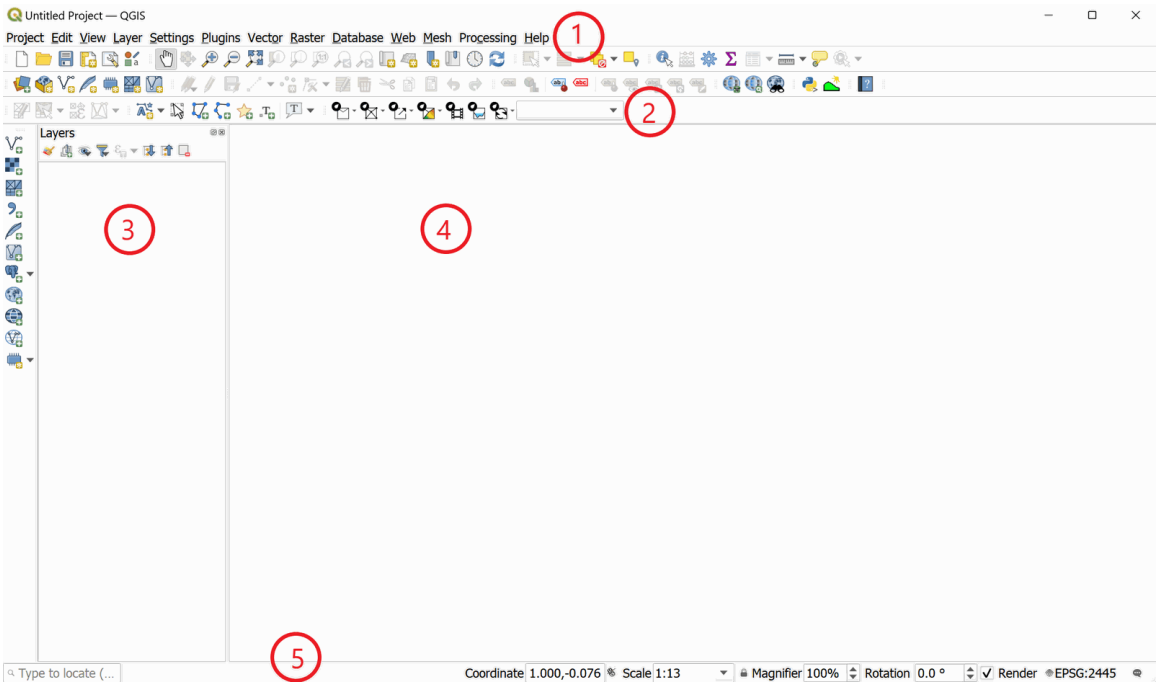


Figure 3.2 – QGIS interface for OilFlow2D indicating window areas. 1: Menu bar, 2: Tool bar, 3: Layer panel, 4: Maps view area, 5: Status bar.

Note the tool bar contains icons that give access to all the OilFlow2D components. Some of the

icons have a drop down list that allow getting to more tools.

3.3 Start a new OilFlow2D project



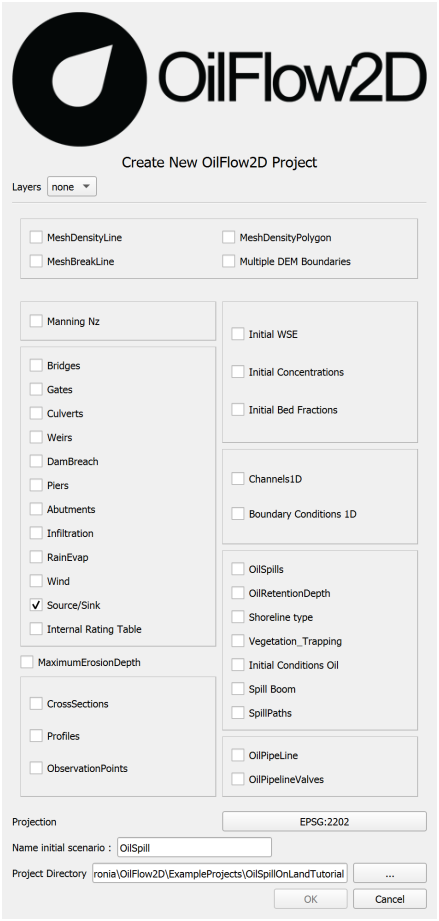
1. To create a new OilFlow2D project, click on the *New OilFlow2D Project* button in the toolbar. A dialog window appears where you select the layers that will be created, the Coordinate Reference System (CRS), and the directory path where the layers will be saved. This tutorial creates only the *Source/Sink* layer: select *None* in *Layers* and check *Source/Sink*.

2. Select the Projection: Coordinate Reference System (CRS) EPSG code: 2202 corresponding to *REGVEN / UTM zone 19N - Projected*

3. Select a working Project Directory:

   ExampleProjects\OilSpillOnLandTutorial

   This will be the folder where the model will write all results and output files. The dialog window should look like this:

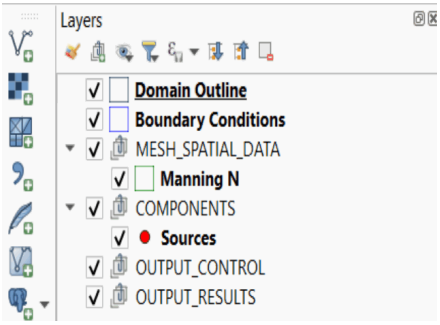


Create New OilFlow2D Project.

Verify that *Layers* is set to *None*, *Source/Sink* is checked, the CRS is EPSG:2202, and the project directory is the extracted OilSpillOnLandTutorial folder.

4. Provide a name for the *Scenario*: *OilSpill* for this tutorial.

5. After clicking OK, the templates of the layers are created, and displayed on the *Layers Panel*:



Layers created for the project.

OilFlow2D will use the unit system defined in the projection you selected. If the projection has coordinates in feet, units will be set to English. If the projection coordinates are in meters, units will be set to Metric/SI.

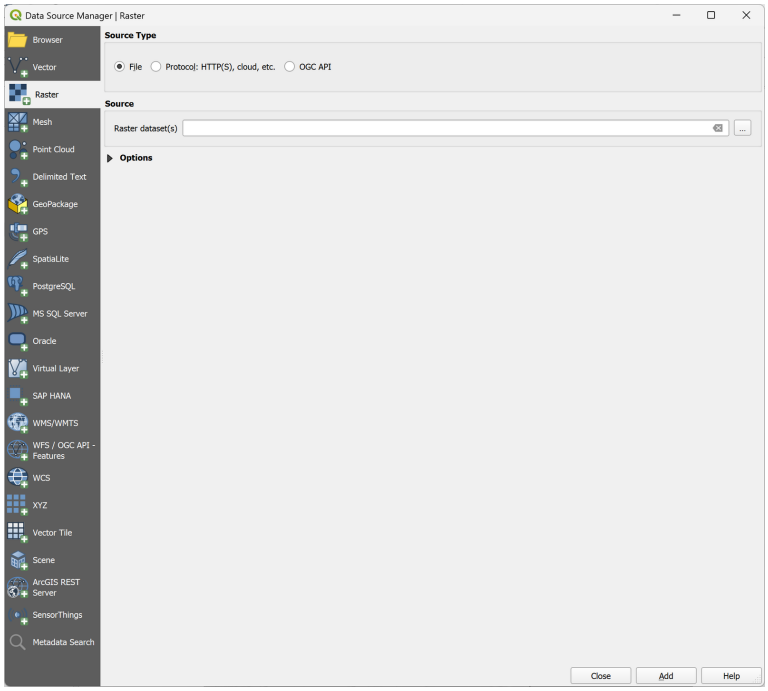
Copy all of the files from the OilSpillOnLandTutorial tutorial folder into the new scenario folder created. In this example, copy them into ExampleProjects\OilSpillOnLandTutorial\OilSpill.

3.4 Load elevation data

OilFlow2D uses elevation data in raster format.

1. To load an ASCII grid file, from the *Layer* menu, click *Add Layer*, and then click *Add Raster Layer…*. You may also click the *Add Raster Layer* button  .

   The following dialog will appear where you can select the raster file containing elevations. In this case, select DEM.tif from the extracted OilSpillOnLandTutorial folder.



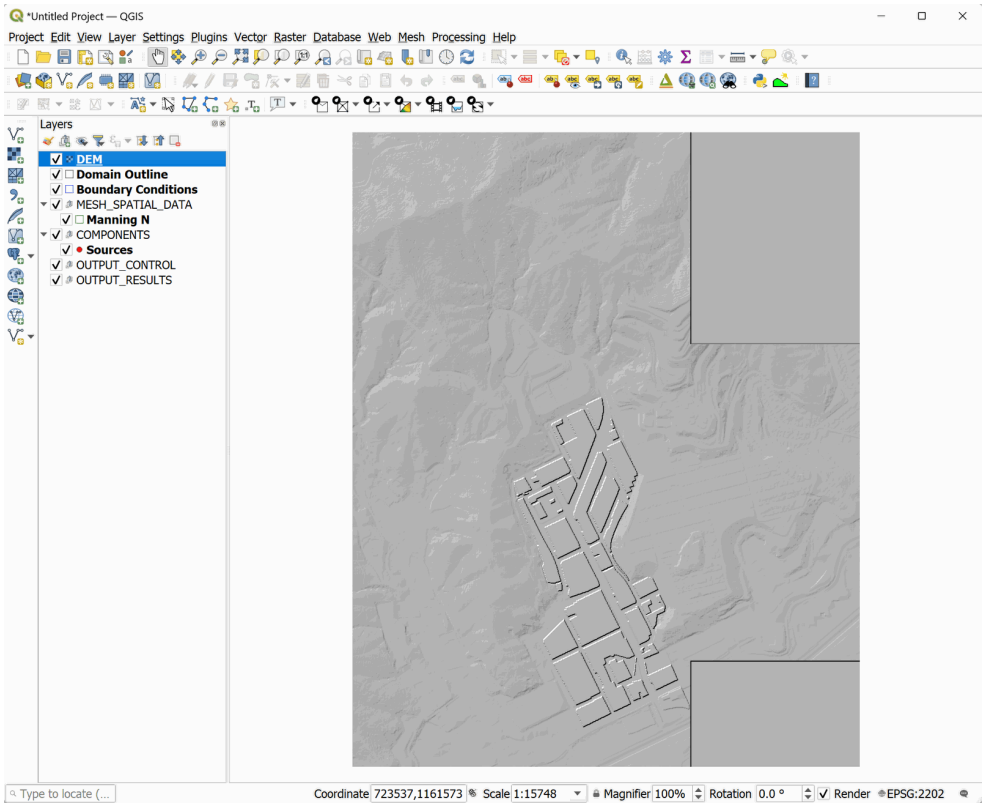
Raster data import dialog.

If you don't have raster elevation data format, you can import it in any other format, and use QGIS to convert it to raster.

2. Click Add, then click Close on the dialog and click on the Zoom Full button  .

3. The raster will be displayed on the screen, by default it is rendered in gray gradient and will need to be changed in the layer properties by *right-clicking* the DEM layer and selecting *Properties*. Under *Symbology*, select the drop down menu for *Render Type* and select *Hillshade*, then click OK.

The layer should look like the following figure:



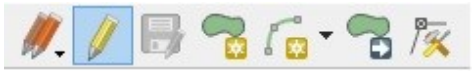
Digital elevation model in raster format.

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

3.5 Create the limits of the modeling area (Domain Outline)

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

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

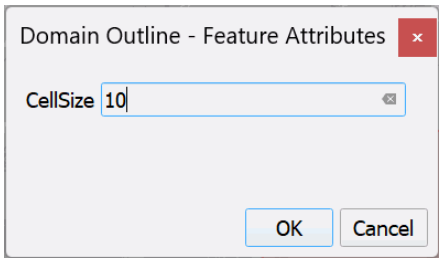
   Menu buttons for digitalization toolbar.

2. Clicking on the button activates the rest of the editing buttons. Now click the *Add Polygon Feature* tool which is the bean looking polygon  .

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

   Make sure that the polygon is contained within the limits of the raster layer since OilFlow2D will not extrapolate elevations to areas that are outside of the available data on the raster layer.

3. To finalize and close the polygon, right-click anywhere on the screen. A dialog window to input the cell size attribute of the newly created polygon. The value for the reference size of the mesh cell is indicated. Enter a value of 10 m.



Input of attributes for the Domain Outline layer.

If you want to make any correction to the polygon, use the *Node* tool  to move, add or remove nodes as necessary.

4. Save the polygon by clicking the *Save* button  .

and click on *Toggle Editing* icon  to deactivate the layer edit mode and complete the *Domain Outline*.



Figure 3.3 – Domain Outline layer.

3.6 Generating the triangular-cell mesh

Now that the *Domain Outline* layer has been created, proceed to create the mesh.

To generate the mesh, click on the icon  .

The following figure shows the generated mesh. You will also find the Layers panel one new layers: *Trimesh* :

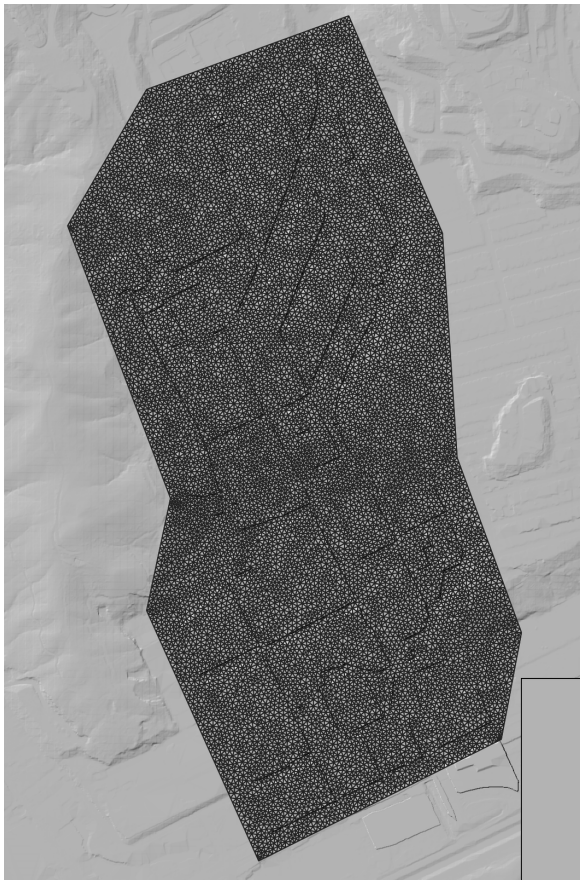


Figure 3.4 – Mesh generated.

3.7 Entering the spill data

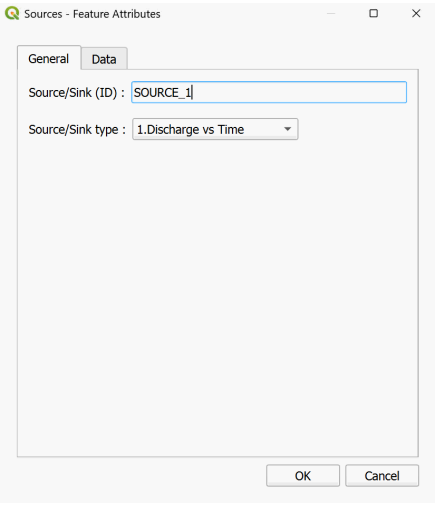
In this tutorial we assume that the spill occurs near the upstream end of the domain. We enter the spill data in the Source/Sink *Sources* layer.

1. Click the *Sources* layer to activate it and then click *Toggle Editing* (pencil) in the toolbar:
2. Now click the *Add Point Feature* tool  .
3. Click on a point at coordinates () to set the spill location.



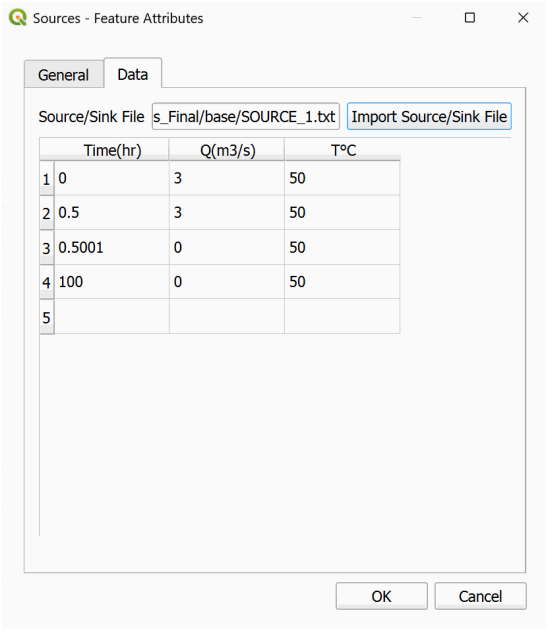
image

4. Then enter the spill name *Source_1*, and select the source/sink type: *Discharge vs Time* as shown:



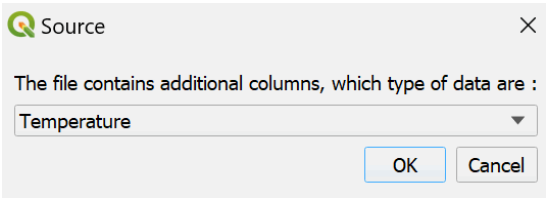
image

5. While you are on this dialog, activate the *Data* panel and use the *Import Source/Sink File* button to select the 'SOURCE_1.txt' file. This file contains the oil spill discharge (flowrate vs time).



image

6. Once you select the 'SOURCE_1.txt' file, a dialog will appear to confirm the type of additional column data we are importing. Leave the default *Temperature* and click OK.



image

7. To complete entering the spill data, click again the *Toggle Editing* icon (pencil) and Save.

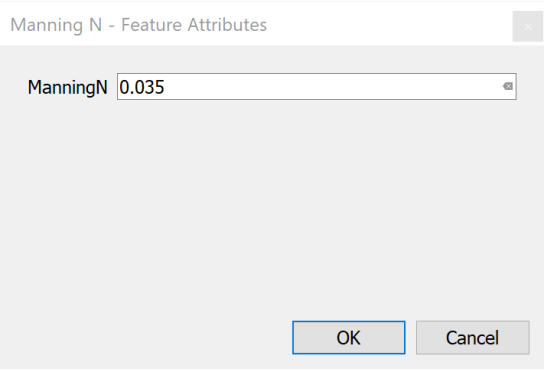
3.8 Assigning Manning's n roughness coefficients

Manning's coefficients are used in OilFlow2D to represent the terrain roughness. To assign Manning's n values, we will enter polygons with particular n's. There can be as many polygons as required to reproduce the spatial variability of this parameter. In this example, a single polygon will be used for the entire area.

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

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

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



Dialog to input Manning N Feature Attributes.

4. Click the Save icon  and then click the Editing Tool icon  to deactivate editing mode.

3.9 Setting the boundary conditions

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

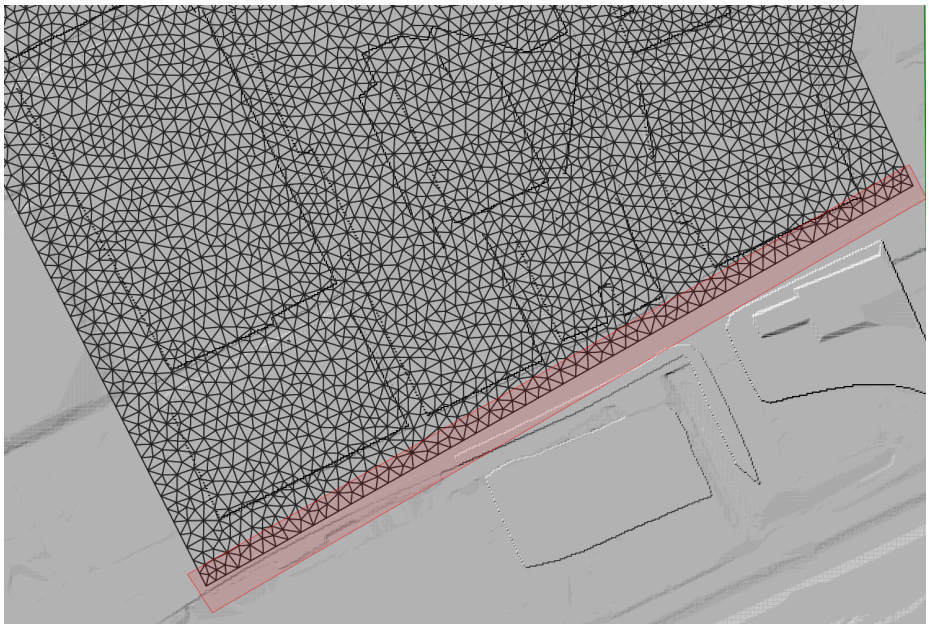
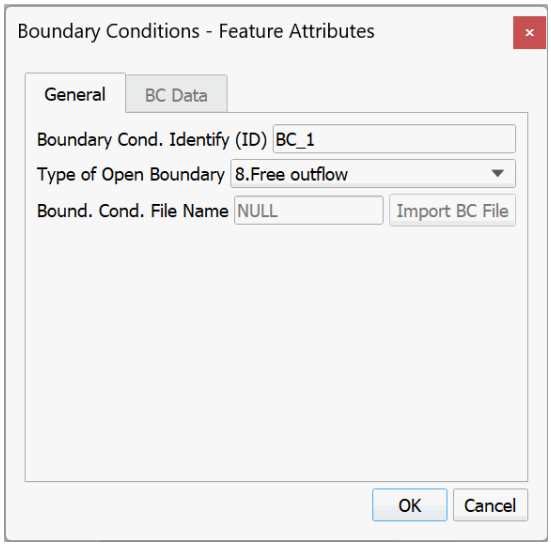


Figure 3.5 – Polygon that covers the nodes defining the Inflow boundary condition segment.

1. Select the *BoundaryConditions* layer in the Layers panel.
2. Click the Toggle Editing button  to add the polygons that are going to indicate the nodes on which the outflow conditions are established.
3. To finish the polygon, right-click on desired location. A window to enter the attributes of the newly created polygon is displayed.
4. The window contains a list to input the ID of this BC (*Boundary Cond. ID*). Click the dropdown menu for *Type of Open Boundary* select the boundary condition Type *8.Free outflow*.

   The window should appear as shown in Figure below:



   Outflow Boundary Condition parameters.

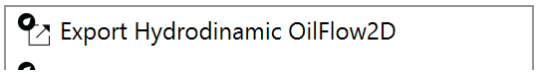
5. Click OK to close the dialog box.
6. Click the Save icon  and then click the Editing Tool icon  to deactivate editing mode.

3.10 Exporting the files to OilFlow2D

Once the layers with the input information to the model have been created, the next step is to run the tool that exports the data files in the proper format required by OilFlow2D. The process is as follows:

1. Make sure all of your required layers are active before performing this step.

2. Run the *Export Hydrodynamic OilFlow2D* plugin:



image

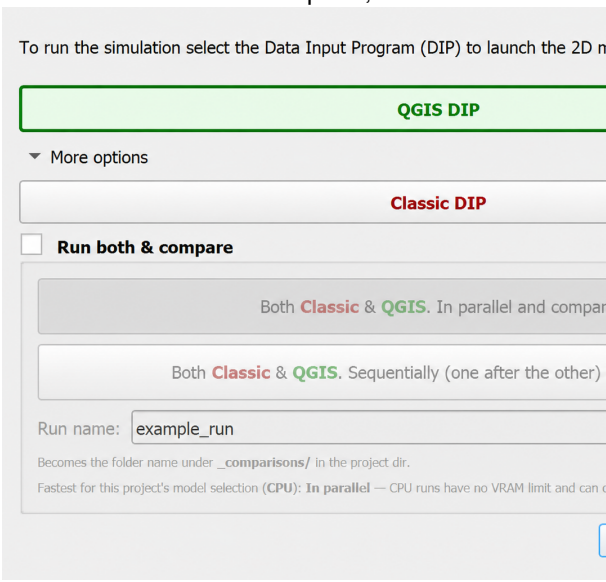
3. A dialog window is presented, where we will indicate the raster layer with the Digital Elevation Model (DEM), as this layer is not created by the plugin and its name may be arbitrary.

4. Make sure to indicate a file name (e.g. Oilspill) including the path where the generated files will be stored, and click OK.

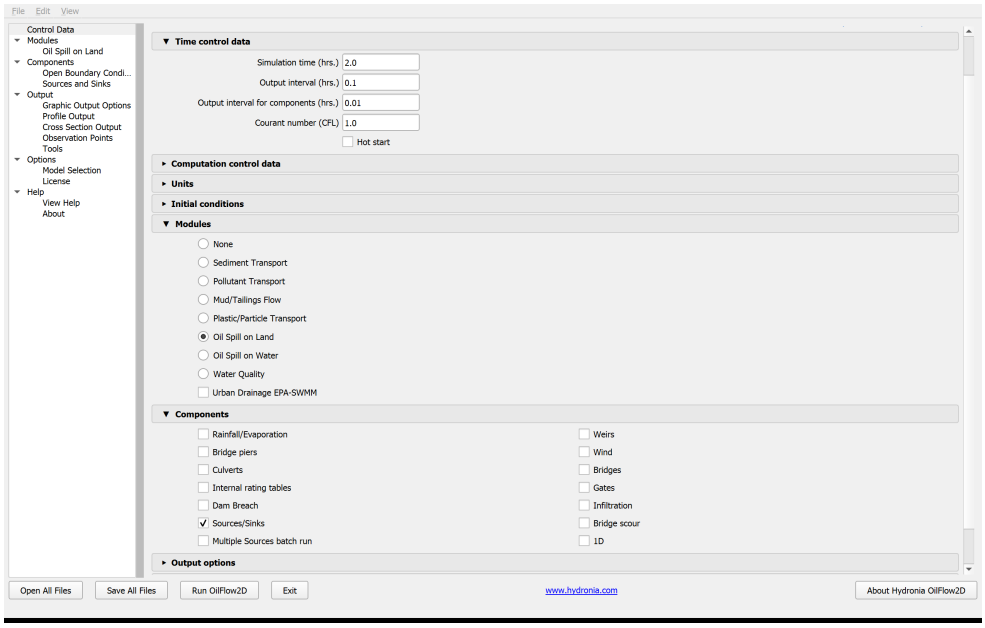


Window dialog of the Export to OilFlow2D tool with the example data.

While exporting the data, you will be able to see the task progress at the top of the message bar. After export completes, version 10 and later users see the Open DIP dialog. QGIS DIP is selected by default; click OK to open it inside QGIS. To use the historical standalone option, expand More options, select Classic DIP, and then click OK.



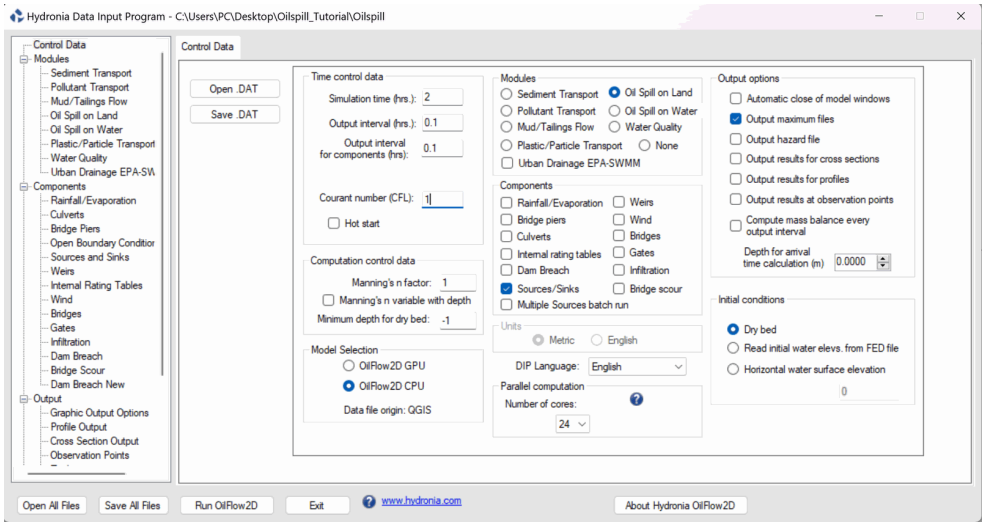
With QGIS DIP selected, the current interface is shown below:



QGIS DIP Control Data panel for the Oil Spill on Land tutorial.

QGIS DIP and Classic DIP use the same project settings, but their interfaces look different. Older standalone or non-maintenance versions may open Classic DIP automatically. The project-specific Classic DIP settings below are retained as a legacy values reference; in this example, Oilspill .DAT should already be set.

Then the window with the input parameters of OilFlow2D are presented, as shown in the image below:



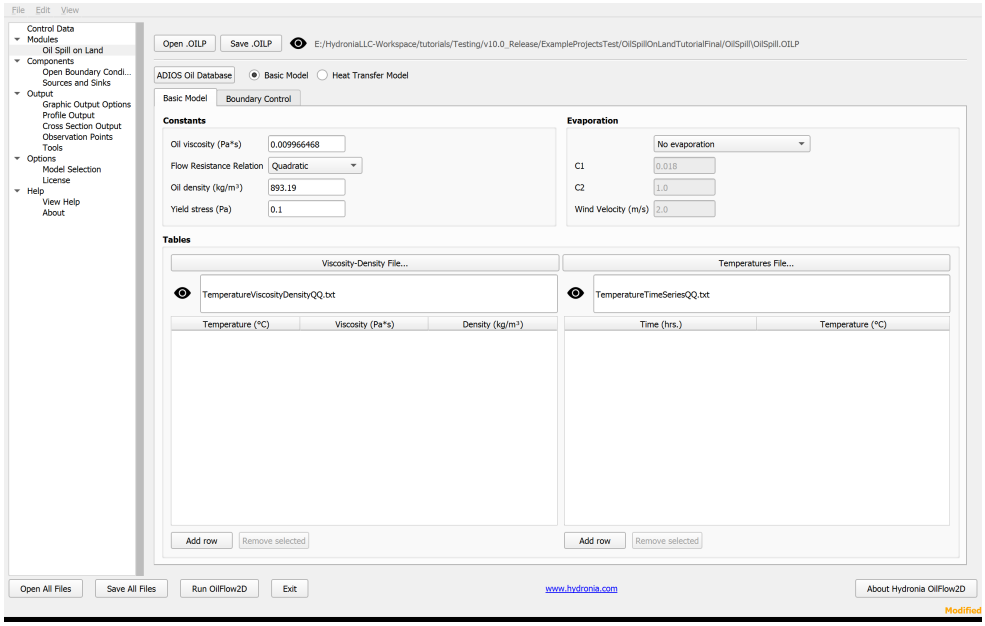
Hydronia Data Input Program window.

5. Ensure that the *Oil Spill on Land* Module is selected, the *Sources/Sinks* component is selected, that the Simulation time is set to 2 hours, and the *Output Interval for components* is set to 0.1.

6. Click [Save .DAT] and overwrite the 'Oilspill.dat' file.

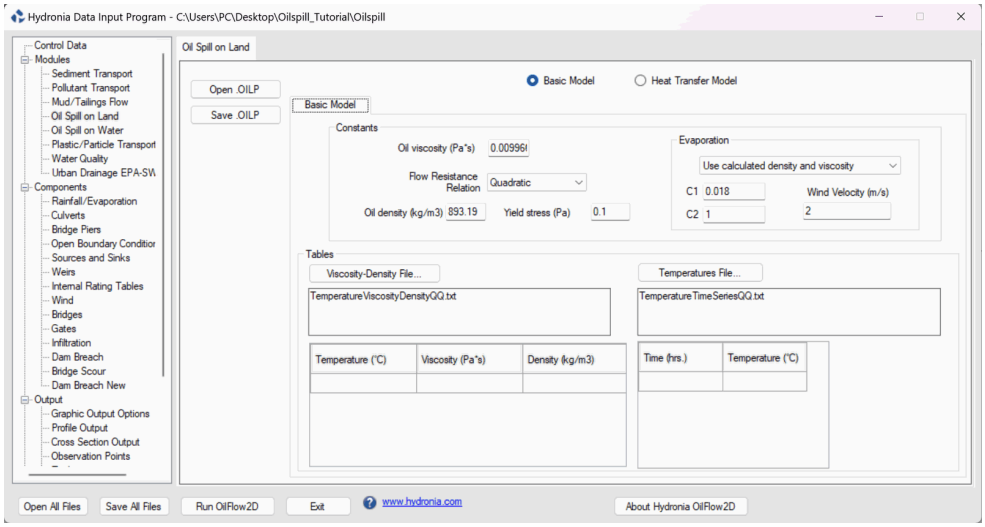
7. Now click on the *Oil Spill on Land* panel, make sure *Basic Model* radial button is selected, and enter the data the following data: Oil viscosity: 0.009966468 Pa.s, Flow Resistance Relation

Quadratic, Oil density: 893.19 kh/m^3, and Yield stress: 0.1 N/m^2 as indicated:



QGIS DIP Oil Spill on Land panel with Basic Model selected.

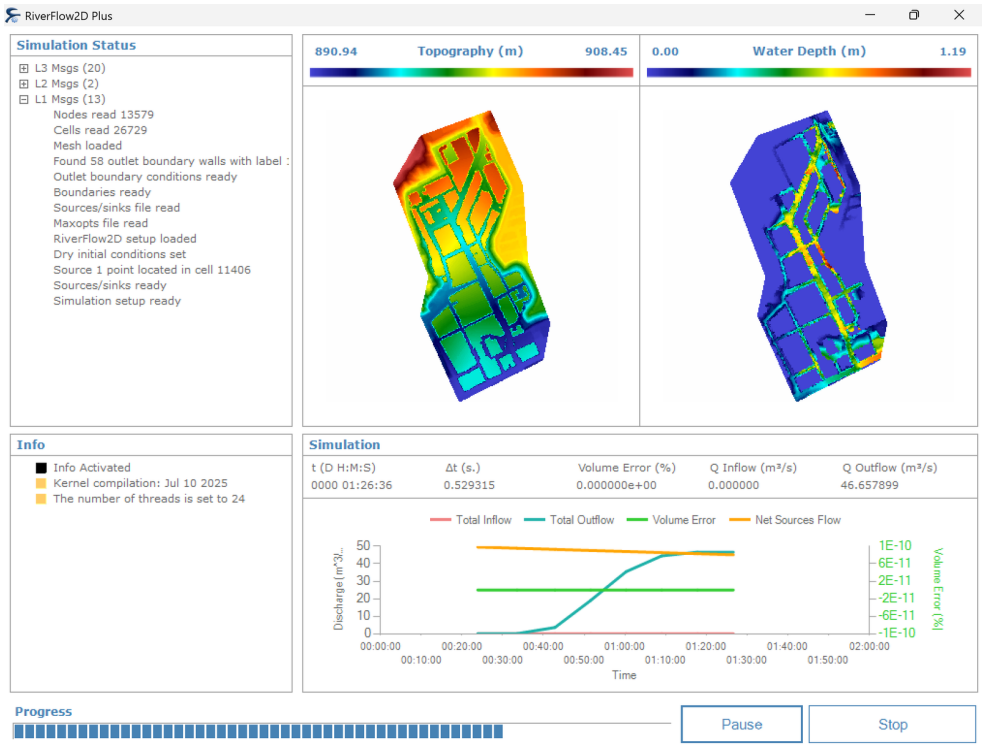
The following Classic DIP panel is retained as a legacy values reference:



Legacy Classic DIP Oil Spill on Land panel.

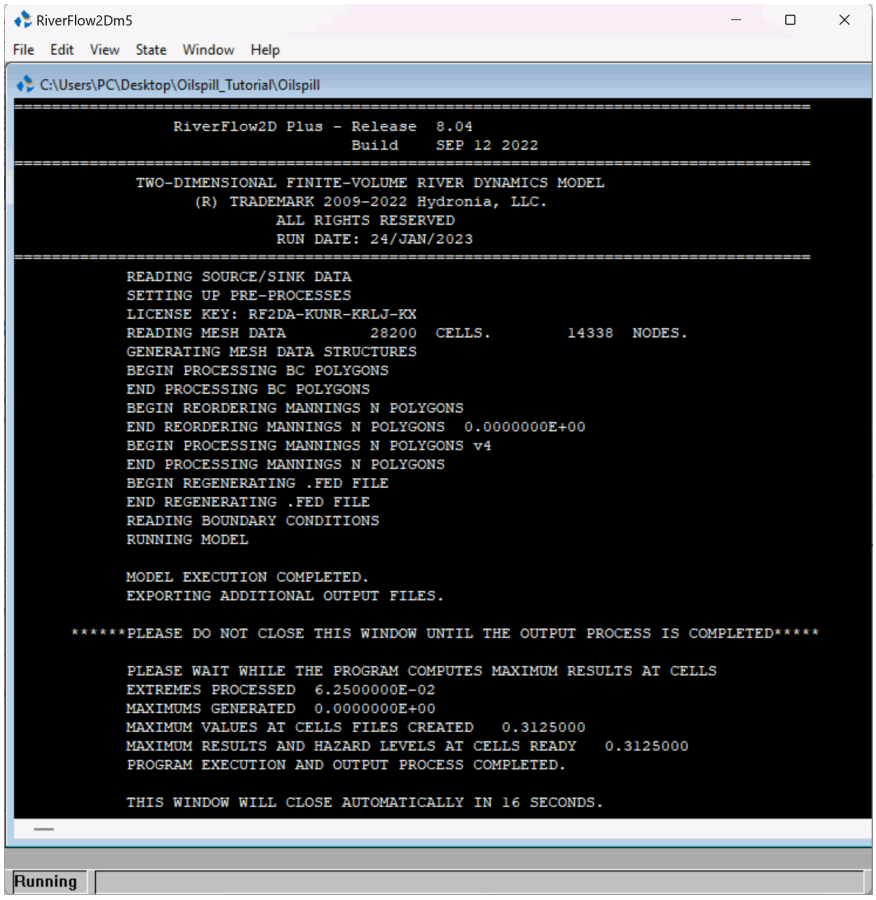
8. Click [Save .DAT] and overwrite the 'Oilspill.dat' file. You will receive warnings for empty files, you may ignore these.

9. Click the *Run OilFlow2D* button to run the model. An image similar to the one shown below should appear:



The graph generated by the OilFlow2D model.

10. When you have finalized reviewing the information included in this window, close it. The following post processing window will appear indicating the final post-processed task performed by the OilFlow2D model. The window will close automatically.

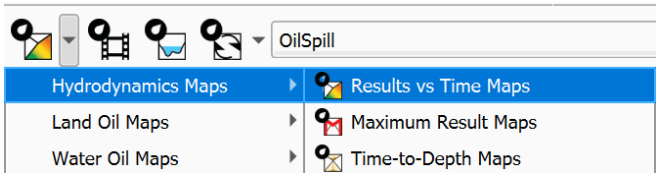


Final post-processes.

3.11 Creating OilFlow2D maps

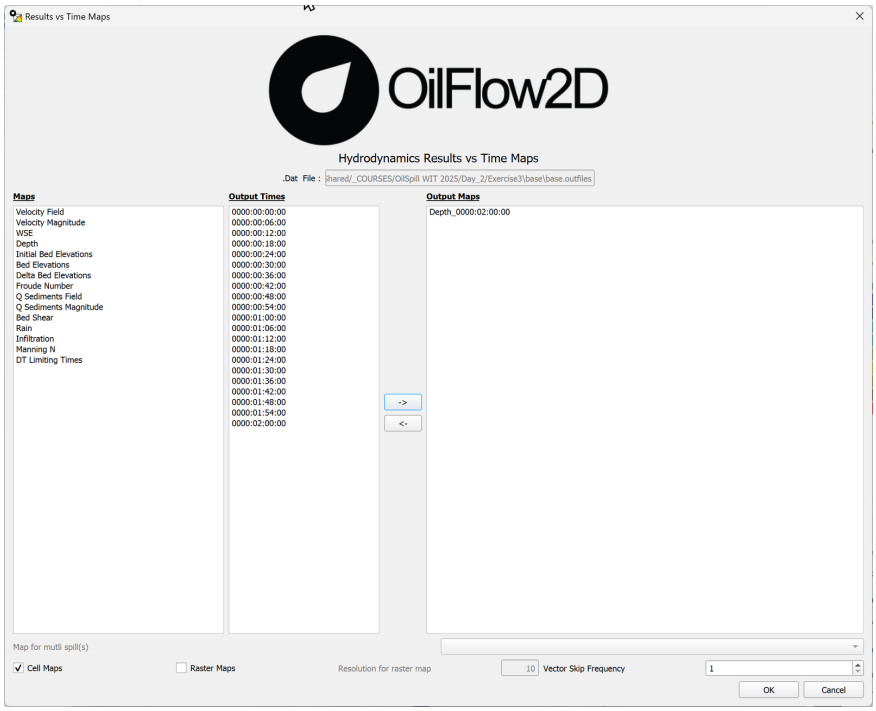
Once OilFlow2D has finished running, you may want to create maps using the model results. OilFlow2D QGIS plugin offers several tools to create maps and animations. In this section we will show how to create depth maps.

1. Click the *OilFlow2D Maps*  button dropdown menu and select *Results vs Time Maps*.



image

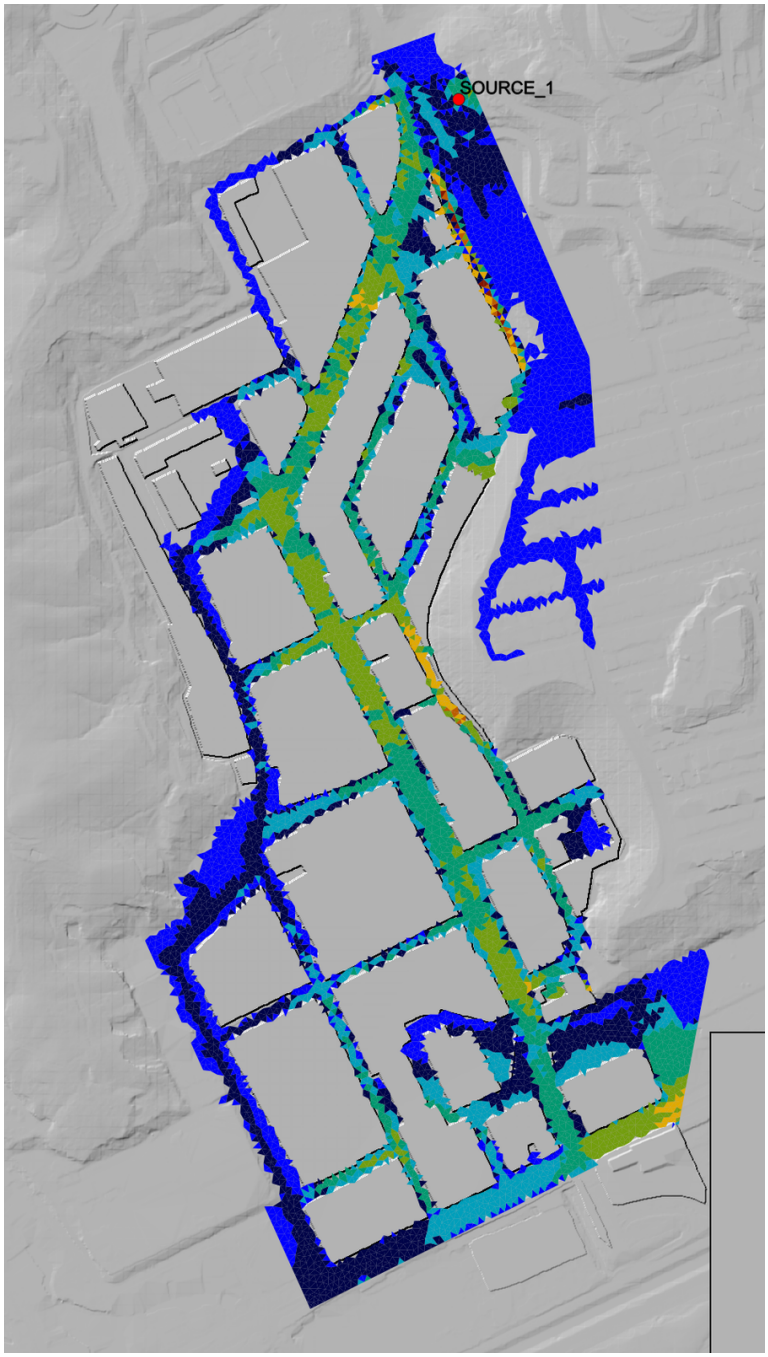
2. A dialog window is presented where we will indicate the output files list 'Oilspill.OUTFILES' and the desired parameter, that will be Depth for time 2 hours. Also indicate the Raster Map option as shown.



Window dialog of the Results vs Time Maps plugin.

3. Click *OK* to start the process. After a few seconds a new layer with the requested map will be created.

4. Right-click the newly generated layer and select *Properties*. In the *Properties* window, click on the *Symbology* tab and change the *Mode* to *Equal Count (Quantile)* option.



Oil depth map at 2 hours after the initial spill.

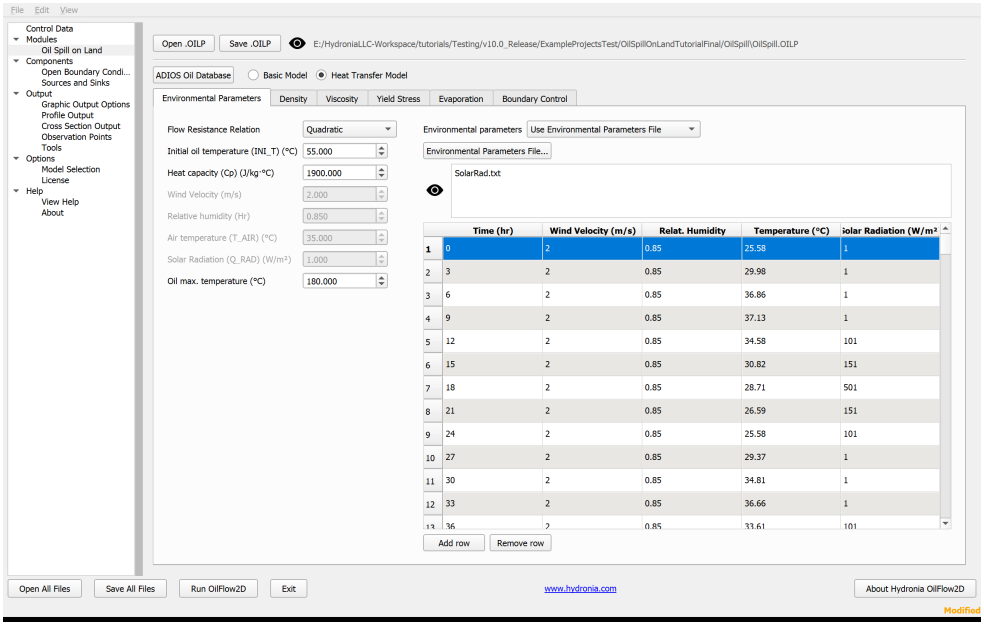
Save the project in QGIS to continue. On the QGIS *Project* menu, click *Save*, to save the project assigning the name Oilspill, in the same directory that you previously selected in the *Create New Project* dialog above.

3.12 Utilizing the Heat Transfer option for oil on land spills in OilFlow2D

In addition to modeling with density, viscosity, and yield stress, OilFlow2D can account for and simulate additional properties such as wind velocity, relative humidity, ambient temperature, and solar radiation effects over time.
Continue from the QGIS DIP or Classic DIP path selected in the export section; no second chooser is needed for this continuation.
The *Heat Transfer* model requires a temperature to be set on a third column for each row in the 'Source_1.txt' file. If the third column of data is missing, the model will not run.

1. In the selected DIP program with the project loaded, click on the *Oil Spill on Land* panel. Click on the *Heat Transfer Model* radial button and click *Open .OILP*. Select the *OilHT.OILP* file and click *Open*.

   This will load a template with the environmental parameters file and a yield stress table to use in the subsequent simulation.



   QGIS DIP Oil Spill on Land panel with Heat Transfer Model selected.

2. Click *Save .OILP* and select *Oilspill.OILP* to overwrite the file that the project is utilizing for the run.

3. Click on the *Control Data* panel and change the *Simulation time (hours.):* to *24*.

   We change the simulation time to account for the day/night cycle and observe how oil properties are affected by ambient temperature changes.

   **OPTIONAL* Under Model Selection select GPU if your computer has an nVidia GPU installed. This can significantly accelerate the simulation*

4. Click *Run OilFlow2D*, then click *Yes* to save the changes and overwrite the existing *Oilspill.dat* file.

Having run the Heat Transfer model, QGIS will be able to produce additional Results vs. time outputs. These will appear in *QGIS*, in the *Results vs. time* dropdown menu:

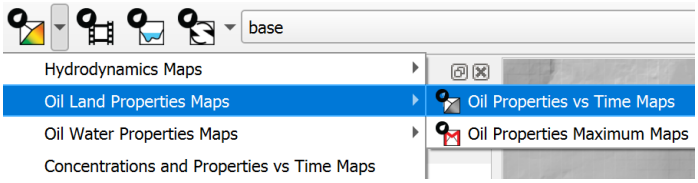


Figure 3.6 – Oil Properties vs. Time Maps menu button.

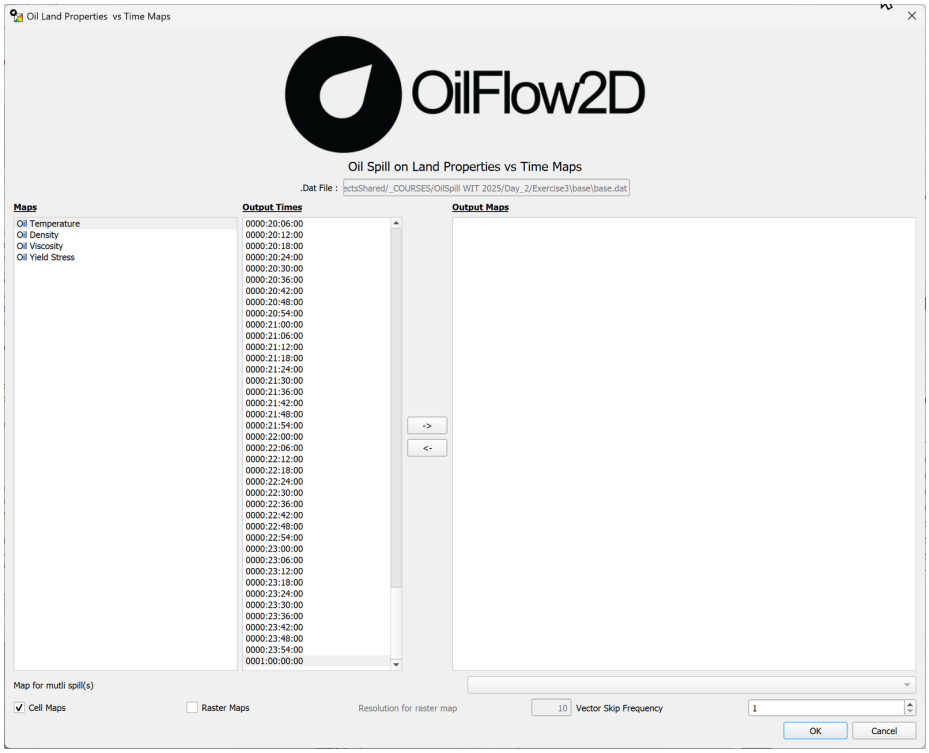


Figure 3.7 – Output map options for Oil Properties vs. Time Maps.

3.13 Animating oil spill on land results in OilFlow2D

Sometimes it is better to view an animation of the results for a different audience, or to view all of the results quickly. Animations can also be exported to a movie file for maximum portability. To create an animation, complete the steps as follows:

1. In QGIS, click on the *Oil Animation Button*  in the OilFlow2D toolbar.

2. A panel will open on the bottom left side of the QGIS interface.



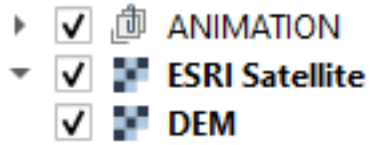
Output map options for Oil Properties vs. Time Maps.

3. Click on the *Select Layer* dropdown menu and select *Depth*. Click on the *Add Layer* button.

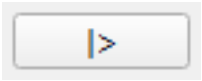
4. After the animation loads, it will be listed in the list window under the *Select layer* button.

5. in the *Layers* panel above, there will be a new group named *ANIMATION* which will contain the newly created *Depth* animation layer.

6. The layer needs to be moved above the DEM layer so that it is visible when being played. Click and drag the *ANIMATION* group and make sure to drop it above the layers that would need to be underneath it, such as the DEM and satellite imagery layers as in the following figure:



Layer order for Animations to be visible.

7. To play the animation, click the *Play*  button in the *OilFlow2D Animation* panel. Make sure all other output layers have been disabled.

The user can change the speed of the animation, export the animation as a shape file, export an image of a particular frame of an animation, and finally export a video via the buttons located at the bottom of the *OilFlow2d Animation* panel.
This concludes the tutorial *Simulating overland oil spills with OilFlow2D*.

4

Multiple Scenario Project Tool

When establishing a mathematical model of a system, it is very common to perform a variety of runs with different values of the most significant parameters to see their effects. In some cases, it is also useful to have the model run the scenario without a project and within a project. In previous versions of the plugin this had to be done in separate QGIS projects, which involved making copies of the project folders and data, which can bring problems with data handling and access. This version of the OilFlow2D plugin for QGIS allows the management of multiple scenarios within the same project. Variants of the same model can be created as subprojects, keeping the scenarios organized in one project.

4.1 New interface elements for managing multi-scenario projects in OilFlow2D

The OilFlow2D toolbar includes a scenario drop-down list. The first toolbar button opens the New OilFlow2D Project/S menu, which contains New OilFlow2D Project, New Scenario, and Delete Scenario. The OilFlow2D Tools menu contains Import Single to MultiScenario Project for converting an existing single-scenario project. The figures below show these current OilFlow2D controls in workflow order.

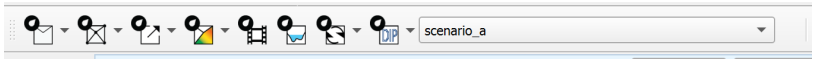


Figure 4.1 – OilFlow2D toolbar with the scenario list

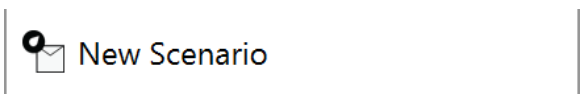


Figure 4.2 – OilFlow2D New Scenario menu item

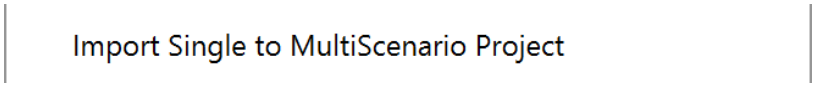
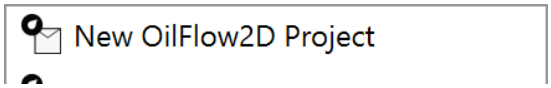


Figure 4.3 – OilFlow2D Import Single to MultiScenario Project menu item

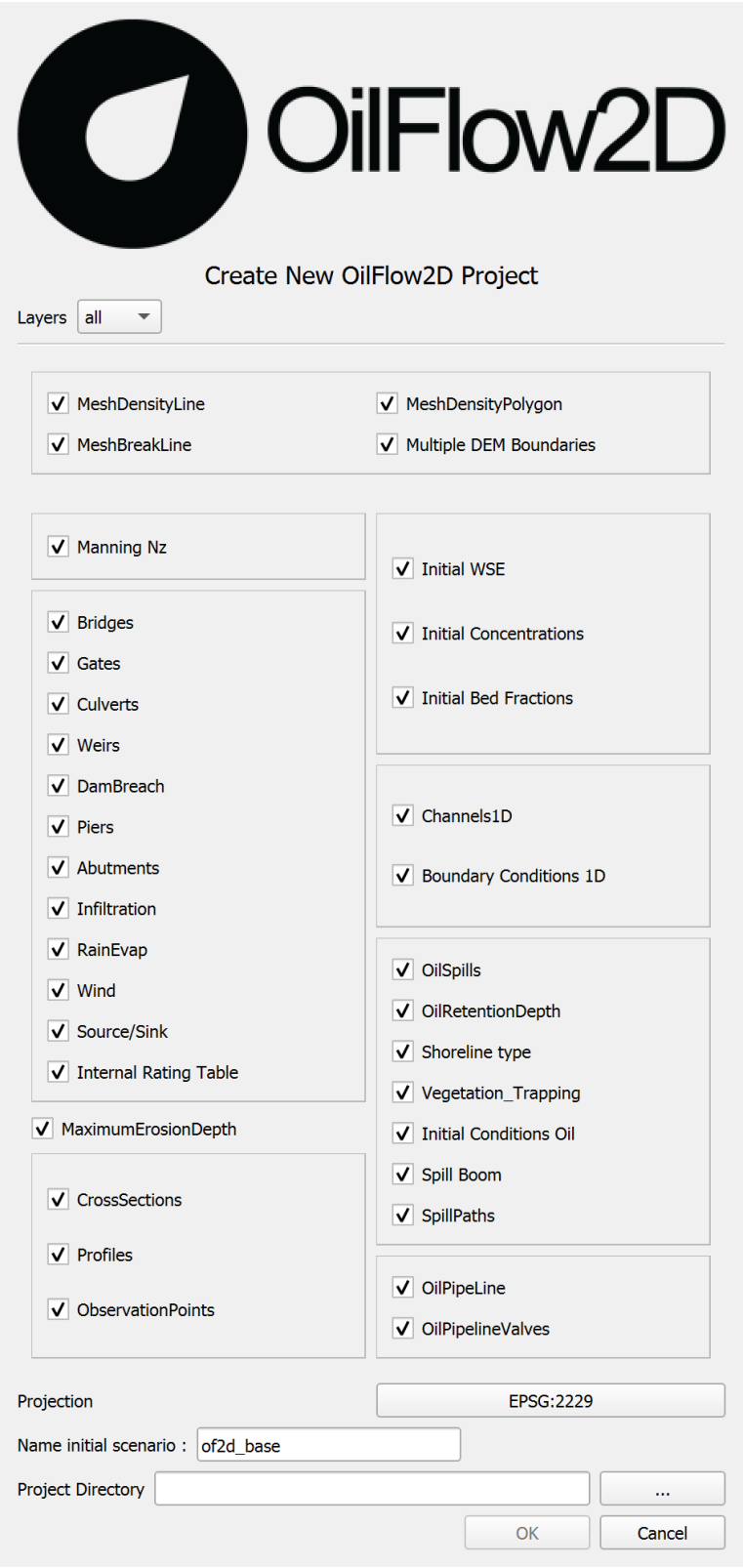
4.2 Creating a new OilFlow2D multiple-scenario project using QGIS

1. Open Plugins > OilFlow2D MS > New OilFlow2D Project/Scene and select New OilFlow2D Project.



OilFlow2D menu to create a new project

2. In the Create New OilFlow2D Project dialog, choose the project directory and review the layers to include. The Name initial scenario : field is initially of2d_base in the current interface. Use a short scenario name without spaces, for example base, before confirming the project.



OilFlow2D Create New OilFlow2D Project dialog

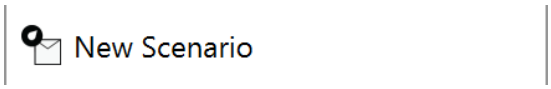
The selected project directory contains a subfolder with the initial scenario name. That scenario folder contains the shape data folder and the other OilFlow2D layer data. Each additional scenario

receives its own scenario folder and shape data folder.

4.3 Create a new scenario

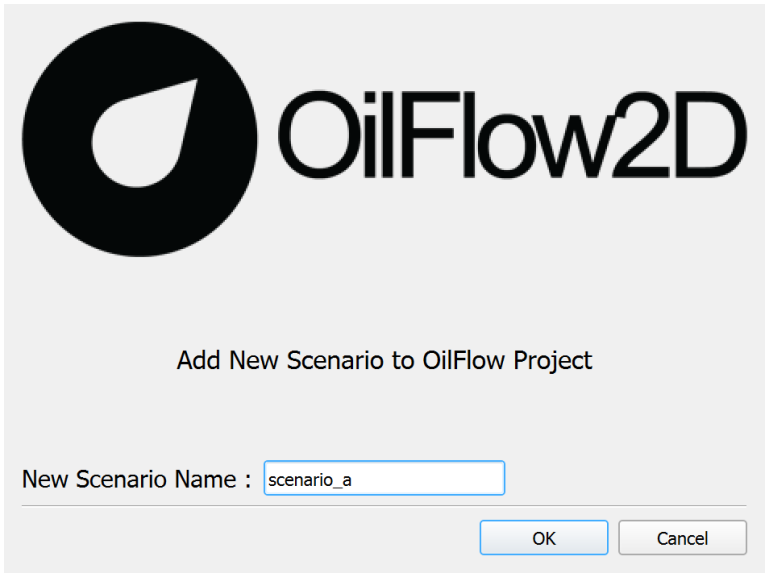
After creating a multi-scenario project, or after converting an existing project as described below, create an additional scenario from the current scenario.

1. Open Plugins > OilFlow2D MS > New OilFlow2D Project/Scene and select the visible New Scenario action.



OilFlow2D menu for creating a new scenario

2. In the Add New Scenario to OilFlow Project dialog, enter a short name without spaces in New Scenario Name :. The configured example uses scenario_a, as shown below, and then confirms with OK.

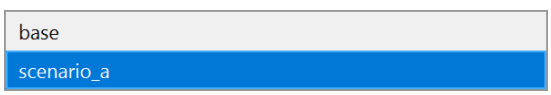


Add New Scenario to OilFlow Project dialog configured with scenario_a

The new scenario is based on the input layers of the current scenario. OilFlow2D creates the new scenario folder, copies the required input-layer files, updates the layer paths, and adds the new name to the scenario list. Post-processing products such as maps and animations are not copied.

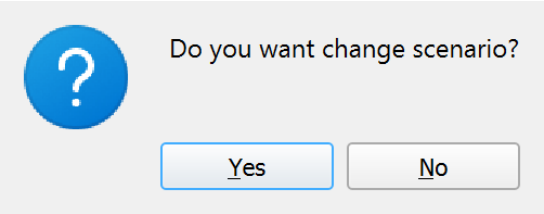
4.4 Switching scenarios

1. Open the scenario list in the OilFlow2D toolbar and select the desired scenario. The example below shows base and scenario_a in the list.



OilFlow2D scenario list with base and scenario_a

2. When changing from the current scenario, OilFlow2D displays the OilFlow2D confirmation dialog with the message Do you want change scenario?. Choose Yes to switch or No to remain in the current scenario.

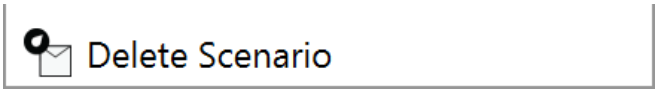


OilFlow2D confirmation dialog for switching scenarios

When a scenario is switched, the state of the layers in the current scenario is saved automatically.

4.5 Deleting scenarios

1. Open Plugins > OilFlow2D MS > New OilFlow2D Project/Scene and select the visible Delete Scenario action.



OilFlow2D Delete Scenario menu item

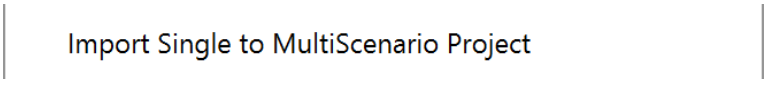
2. Complete the deletion through the visible OilFlow2D dialog and prompts.

Do not delete a scenario folder manually from the project directory. The project can become unusable; use the OilFlow2D action instead.

4.6 Import an OilFlow2D project into Multi-scenario mode

The multi-scenario tool can convert an existing single-scenario OilFlow2D project.

1. Open Plugins > OilFlow2D MS > OilFlow2D Tools and select the exact visible action Import Single to MultiSce



OilFlow2D Tools menu with Import Single to MultiScenario Project

2. In the visible import window, choose the source .DAT file and confirm the import.

OilFlow2D creates a scenario subfolder for the imported project, copies the project data into that scenario folder, and updates the QGIS project metadata and layer sources to the new locations. The scenario list is then updated with the imported scenario.
The original project files remain in the project folder. Keep the QGIS project file (.qgz) and remove the original data only when it is no longer needed.
This concludes the tutorial on using the Multiple Scenario Project tool within the QGIS OilFlow2D toolbar.

5

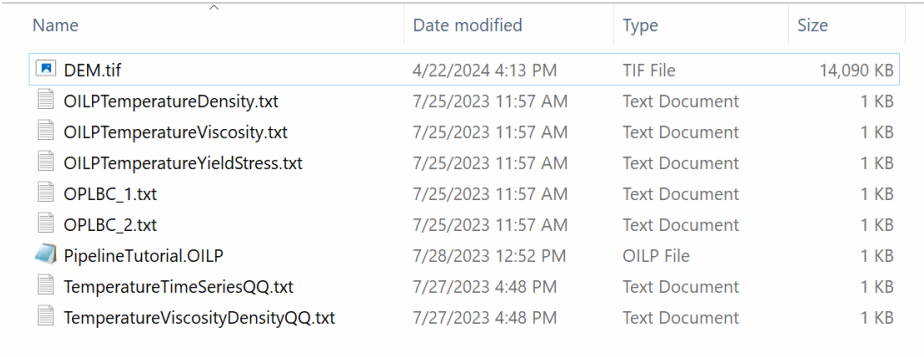
Simulating Oil Pipeline Breaks

This tutorial will explain how to use the OilFlow2D model to simulate oil spills from a pipeline utilizing the Oil Pipeline Module. The steps to accomplish this are as follows:

1. Starting QGIS and Creating a new *OilFlow2D* project.
2. Adding the required layers.
3. Create the Pipeline by adding the *Oilpipeline* layer.
4. Running the *Oil Pipeline Break Module* within the OilFlow2D plugin.
5. Generating the triangular mesh and exporting the data files to the OilFlow2D model.
6. Configuring parameters in the Hydronia Data Input Program.
7. Creating maps of all sources from results data.

5.1 Tutorial data

The files required to follow this tutorial can be extracted from the 'ExampleProjects' zip file under the 'SimulatingPipelines' folder. This zip file is downloaded separately from your installation materials. The folder contains the files shown:

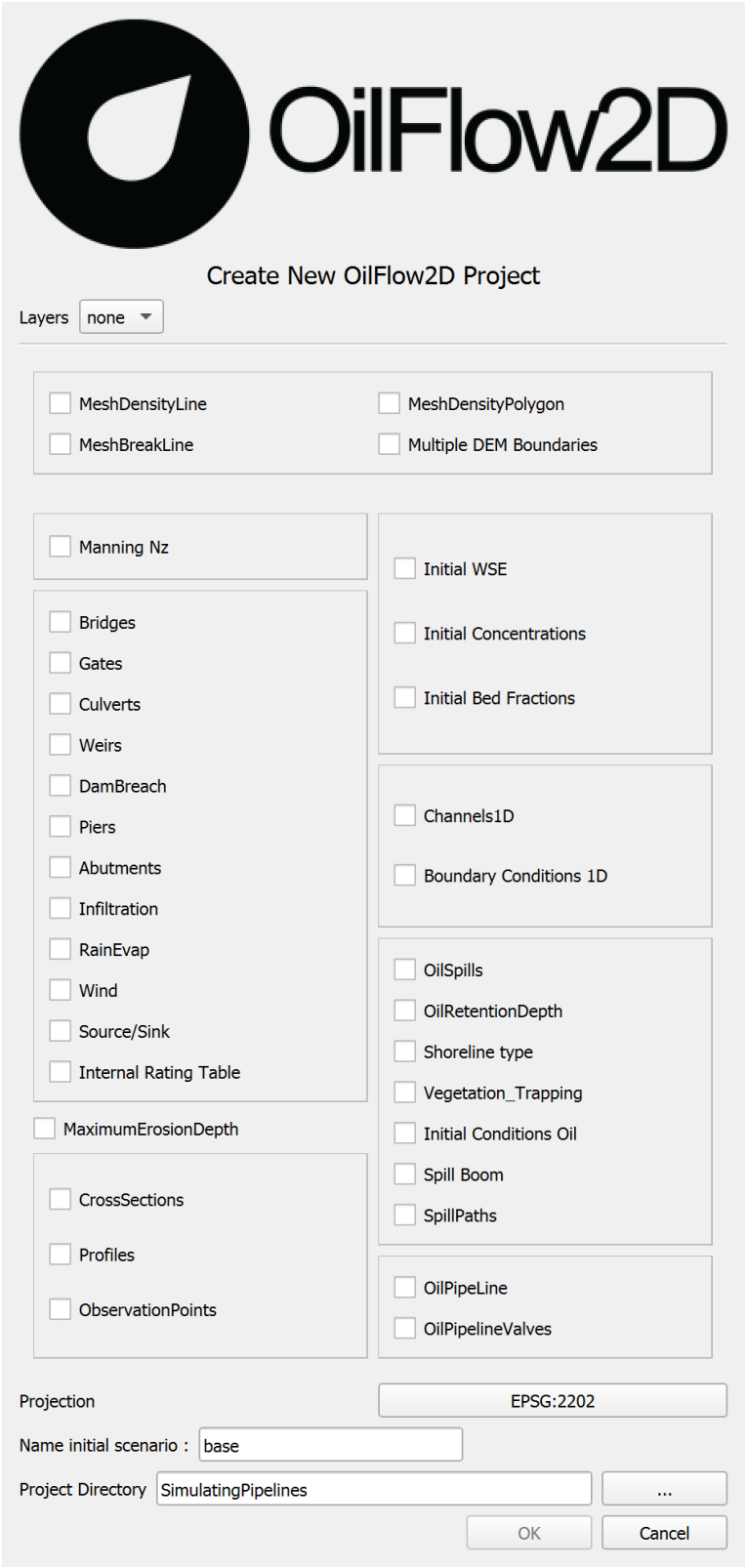


Name	Date modified	Type	Size
DEM.tif	4/22/2024 4:13 PM	TIF File	14,090 KB
OILPTemperatureDensity.txt	7/25/2023 11:57 AM	Text Document	1 KB
OILPTemperatureViscosity.txt	7/25/2023 11:57 AM	Text Document	1 KB
OILPTemperatureYieldStress.txt	7/25/2023 11:57 AM	Text Document	1 KB
OPLBC_1.txt	7/25/2023 11:57 AM	Text Document	1 KB
OPLBC_2.txt	7/25/2023 11:57 AM	Text Document	1 KB
PipelineTutorial.OILP	7/28/2023 12:52 PM	OILP File	1 KB
TemperatureTimeSeriesQQ.txt	7/27/2023 4:48 PM	Text Document	1 KB
TemperatureViscosityDensityQQ.txt	7/27/2023 4:48 PM	Text Document	1 KB

Figure 5.1 – Files with data required in this tutorial.

5.2 Starting QGIS and Creating a new *OilFlow2D* project

1. To create a new OilFlow2D project, open QGIS and then click on the *New OilFlow2D Project* button  in the toolbar. A dialog window appears where you select the optional component layers, the Coordinate Reference System (CRS), the initial scenario name, and the project directory. For this example, select none in *Layers*, choose EPSG:2202, keep *Name initial scenario* set to base, and use SimulatingPipelines as the *Project Directory*.



New OilFlow2D Project window configured for SimulatingPipelines.

Copy all of the files from the SimulatingPipelines tutorial folder into the new scenario folder created. In this example it would be copied into \SimulatingPipelines\base.

2. Save the QGIS project by clicking on the Save Project  icon in the toolbar.

3. Make sure to save the QGIS project above the scenario folder. In this example it would be saved under \SimulatingPipelines\.

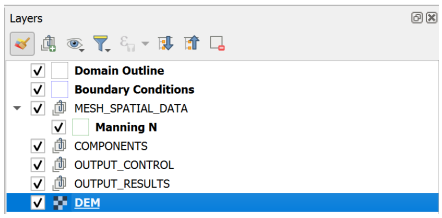
5.3 Setting up the required layers

The Digital Elevation Model (DEM) or Digital Surface Model (DSM) needs to be added to the project, then the *Domain Outline*, *Manning N* will need to be drawn and given input values.

1. In this tutorial we will load a DSM clicking the *Add Raster Layer* button  . You may also use the QGIS shortcut *Control+Shift+R*.

   The DSM file DEM.tif is located in the extracted SimulatingPipelines tutorial folder.

2. Drag the newly added DSM layer to the bottom of the list in the *Layers* panel:

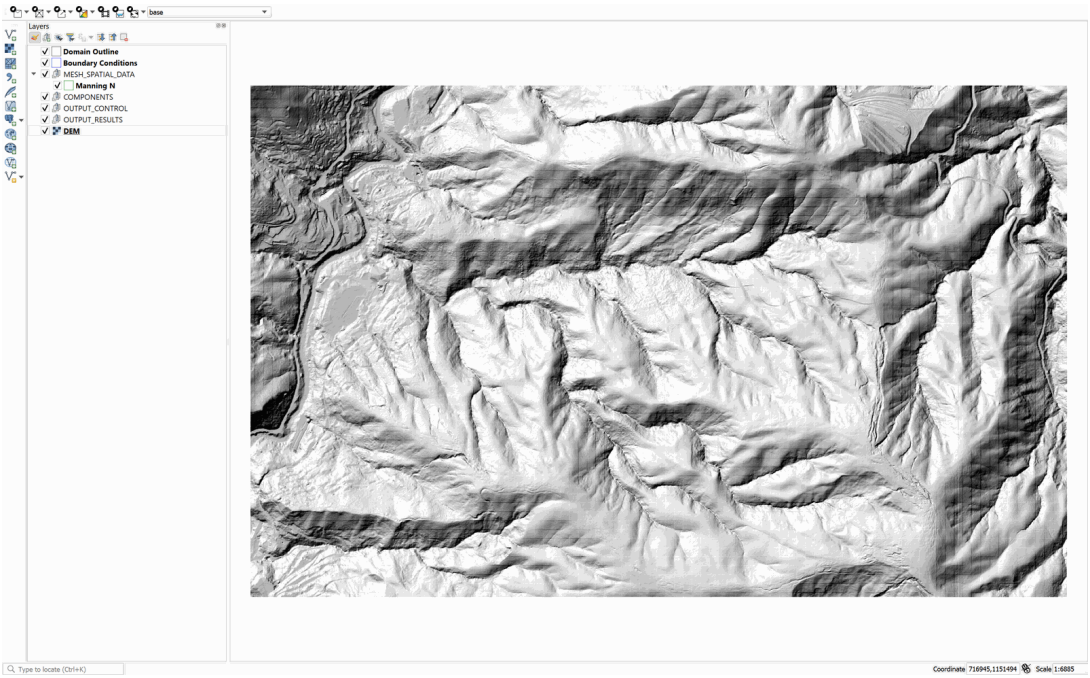


   Layer order for DEM.

   Right-clicking on the label of the new raster layer and selecting *Properties*, in the *Symbology* panel you can change the *Render type* for a more informative palette such as *Hillshade* for instance.

3. Right-click the DEM layer and select *Zoom to layer(s)*.

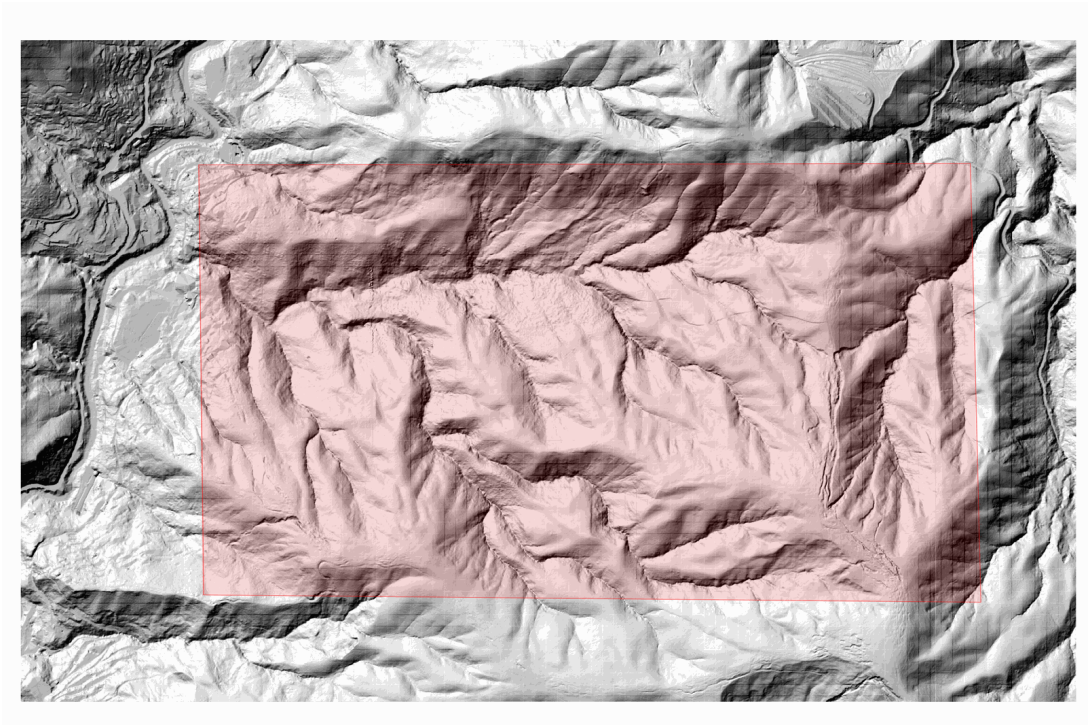
   This is what you should have on your screen so far:



QGIS Layout after adding the DEM.

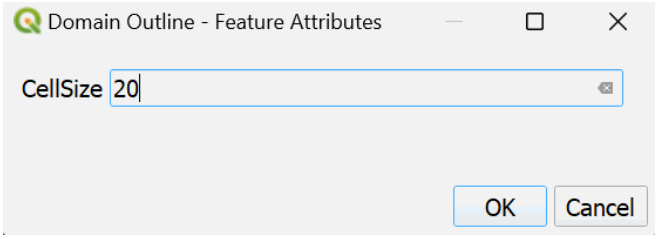
4. The limits of the modeling area are defined using a polygon on the *Domain Outline* layer. To create it do as follows: Select the *Domain Outline* layer in the *Layers* panel and click on *Toggle Editing* icon  in the toolbar section.

5. Click the *Add Polygon Feature* tool  . Proceed to delineate the outline of the polygon by clicking the vertices with the left mouse button as shown:



Domain Outline area.

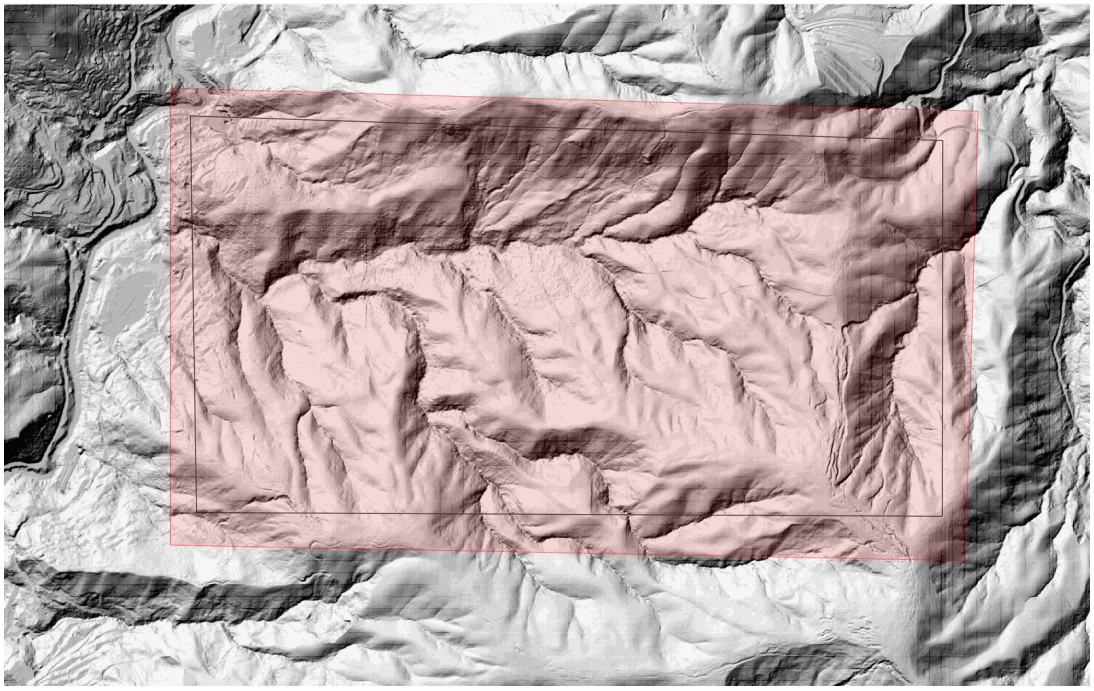
6. Right-click to close the polygon. A dialog will appear asking for the *Cellsize* value for our mesh. Input 20 as our value.



Domain Outline Feature cellsize input dialog box.

7. Click the Save icon  and then click the Toggle Editing icon  to deactivate editing mode.
8. Now that we have created the *Domain Outline*, deactivate the *Boundary Conditions* layer by unchecking the box on the left side, as it will not be used for this exercise.
9. Select the *Manning N* by clicking on the layer located under the *MESH_SPATIAL_DATA* group.
10. Click on *Toggle Editing* icon  in the toolbar section.

11. Click the *Add Polygon Feature* tool  . Proceed to delineate the outline of the polygon by clicking the vertices with the left mouse button, keeping the polygon outside of the *Domain Outline* polygon as shown:



Domain Outline Feature cellsize input dialog box.

12. Right-click to close the polygon. A dialog window will appear to input the *Manning N* coefficient. Enter the value 0.03.

13. Deactivate the *Boundary Conditions* layer in the *Layers* panel by clicking the checkbox on the left side. It will not be used in this tutorial.

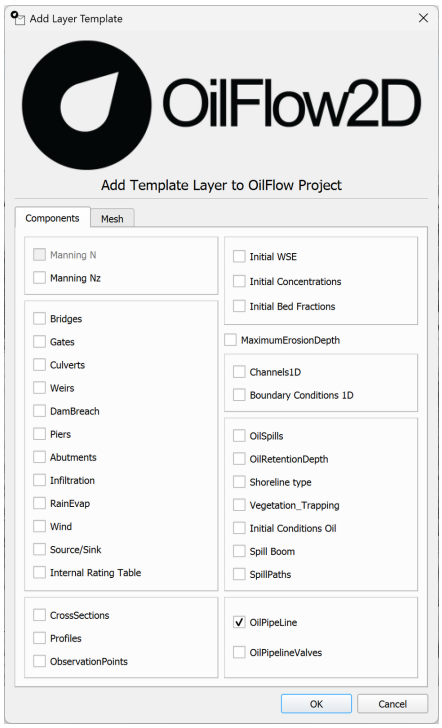
14. Click the Save icon  and then click the Toggle Editing icon  to deactivate editing mode.

15. Save the QGIS project by clicking on the Save Project  icon in the toolbar.

5.4 Create the Pipeline by adding the *Oilpipeline* layer

A polyline will be used to delineate the location and length of the pipeline for this tutorial. First, a the *Oilpipeline* layer must be created:

1. Click on the *OilFlow2D Tools*  dropdown menu and select *New Template Layer*.



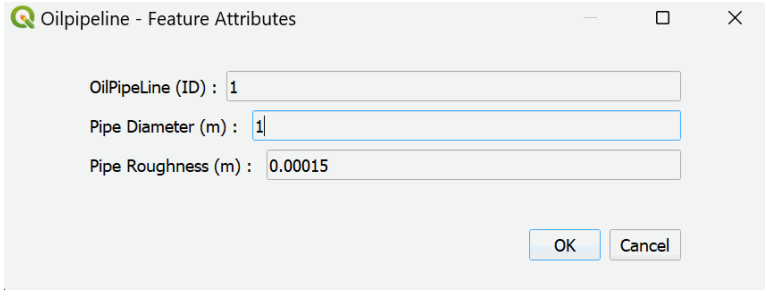
Add Layer Template window.

2. Click [OK] in the *Add Layer Template* window.

3. Select the *OilPipeLine* layer in the *Layers* panel and click the *Toggle Editing* icon  in the toolbar section.

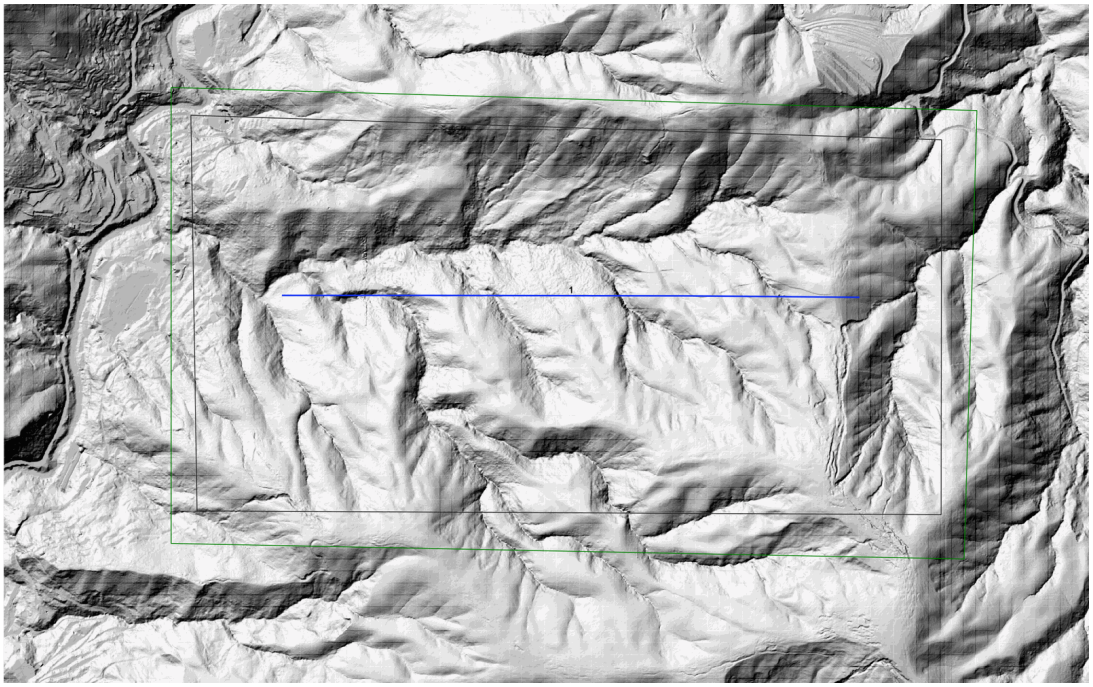
4. Click the *Add Line Feature*  button and then start to draw the pipeline on the map by left-clicking to create the vertices.

5. Right-click to end drawing the pipeline. A dialog window will appear requesting to add additional data about the pipeline. Keep default values and make the pipe diameter 3 meters as shown:



Oilpipeline layer input dialog box.

The finished pipeline should look similar to the following:



Finished Oilpipelines polyline.

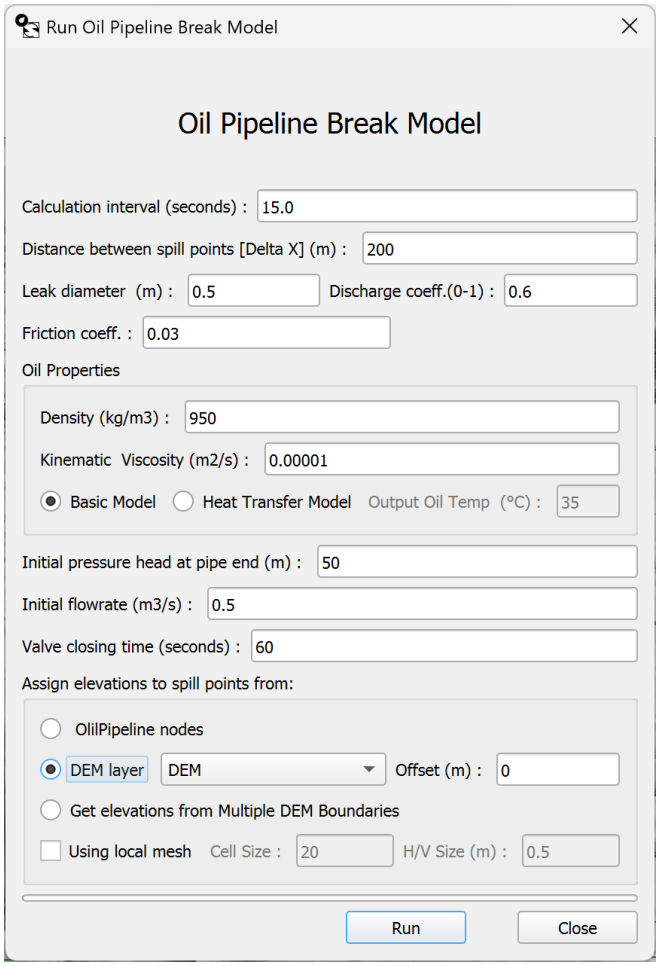
6. Save the QGIS project by clicking on the Save Project  icon in the toolbar.

5.5 Running the *Oil Pipeline Module* within the OilFlow2D plugin

The *Oil Pipeline Break Model* tool will be used in this tutorial to create the sources of the spill along the length of the pipeline. It requires us to specify the length between each break, the total inflow rate, the oil properties, and will calculate the spill outflow rates for each break based on these parameters.

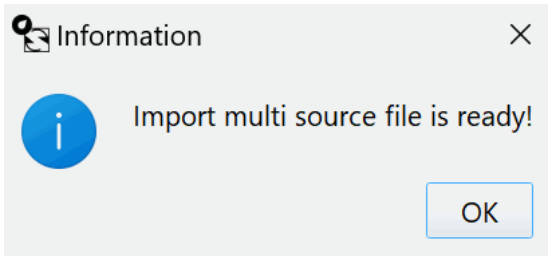
1. Click on the *OilFlow2D Tools*  dropdown menu and select *Run Oil Pipeline Break Model*.

2. Enter the data as shown in the following figure:



Oil Pipeline Break parameters.

3. Click [Run]. The plugin will generate a new *Sources* layer, populate it with the spill points and they will appear along the pipeline in the map area:

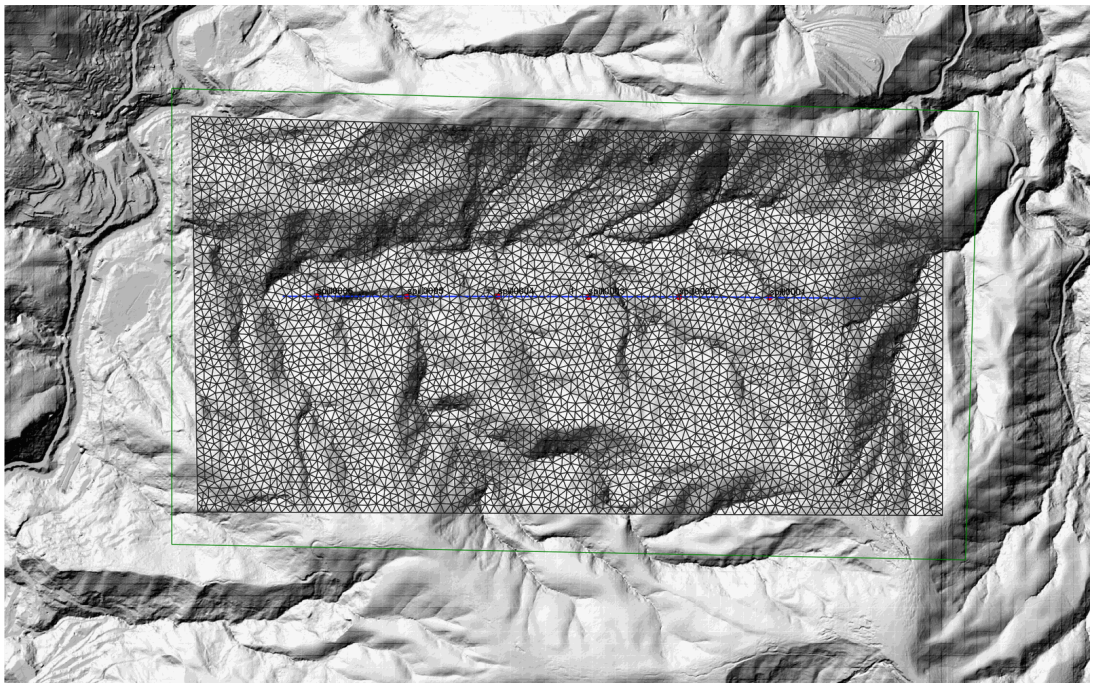


Spill sources created after running the Oil Pipeline Break Model.

4. Save your QGIS project by clicking on the Save Project icon in the toolbar.

5.6 Generating the triangular mesh and exporting to the OilFlow2D model

1. Create the mesh with a cell size of 20 meters by clicking on the *Generate Trimesh*  button.



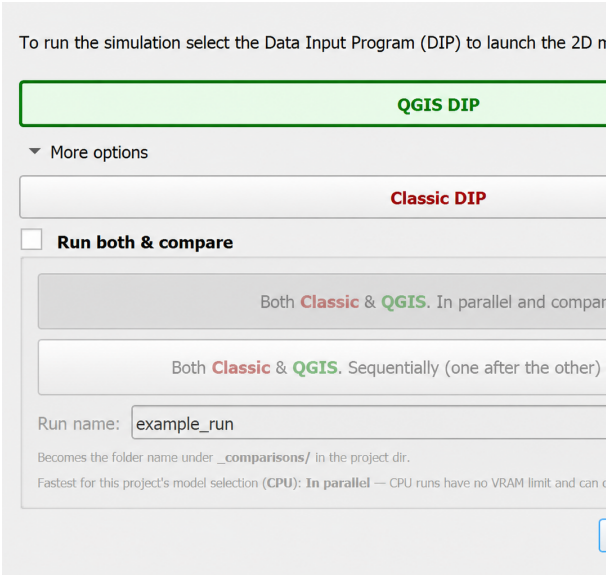
Spill sources created after running the Oil Pipeline Break Model.

2. Save your QGIS project by clicking on the Save Project icon in the toolbar.

3. Click on the *Export files for OilFlow2D*  dropdown and select *Export Hydrodynamic OilFlow2D*.

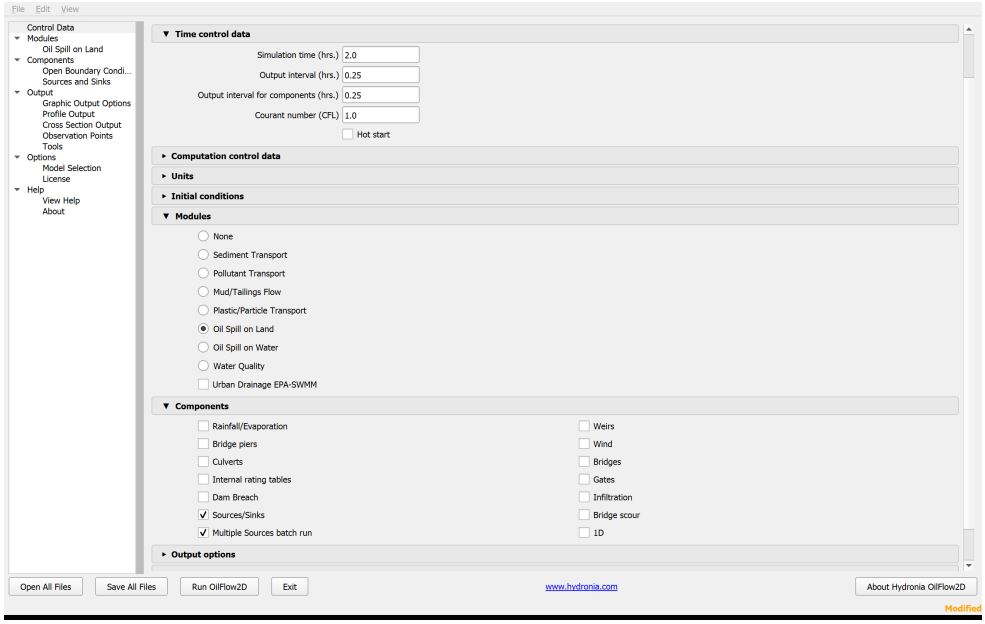
4. Leave all default values and click [OK] to export.

   After export completes, version 10 and later users see the Open DIP dialog. QGIS DIP is selected by default; click OK to open it inside QGIS. To use the historical standalone option, ex-



pand More options, select Classic DIP, and then click OK.

With QGIS DIP selected, the current interface is shown below:



QGIS DIP Control Data panel for Simulating Pipelines.

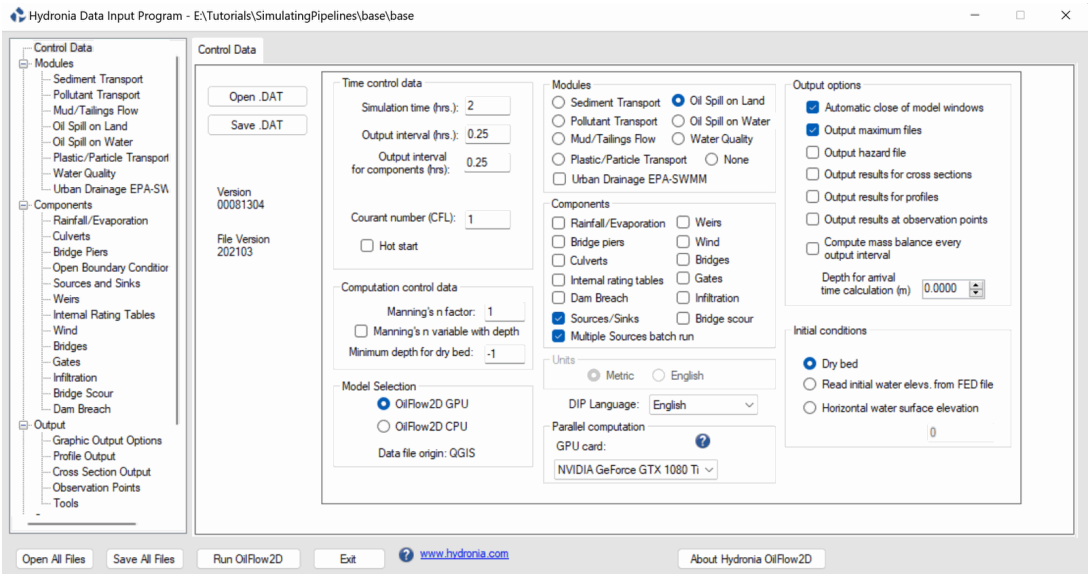
QGIS DIP and Classic DIP use the same project settings, but their interfaces look different. Older standalone or non-maintenance versions may open Classic DIP automatically. The project-specific Classic DIP settings and Run steps below are retained.

5.7 Configuring parameters in the Hydronia Data Input Program

The tutorial will simulate pipeline breaks at each of the spill points for 2 hours with 15 minute intervals. Please do the following:

1. Configure the *Control Data* parameters with the parameters in the figure below:

   The following Classic DIP Control Data panel is retained as a legacy values reference:



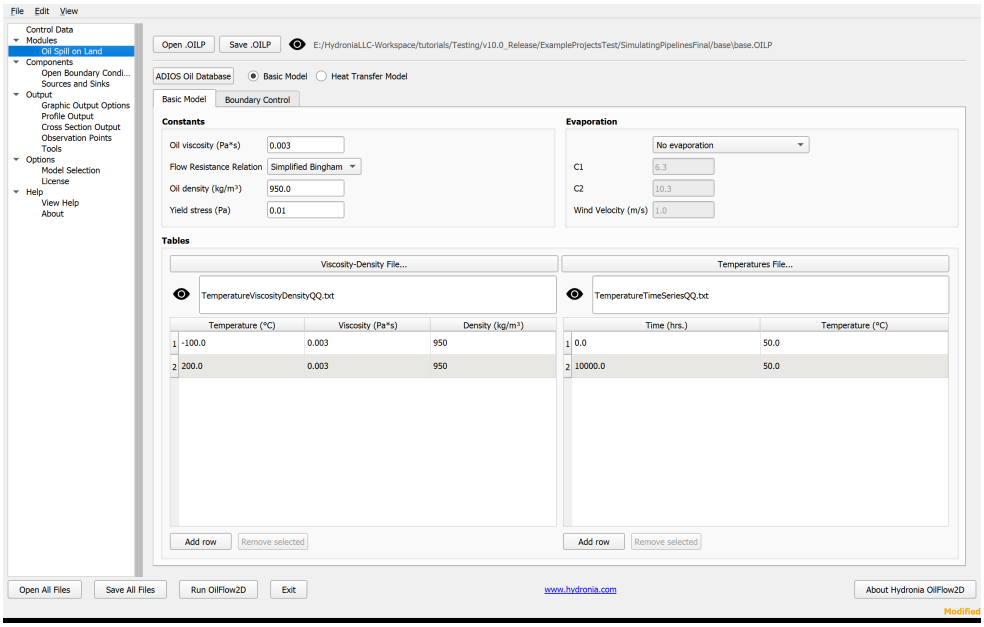
   Legacy Classic DIP Control Data parameters.

2. Click on the *Oil Spill on Land* module in the left pane of the DIP.

3. Click on the [Open .OILP] button.

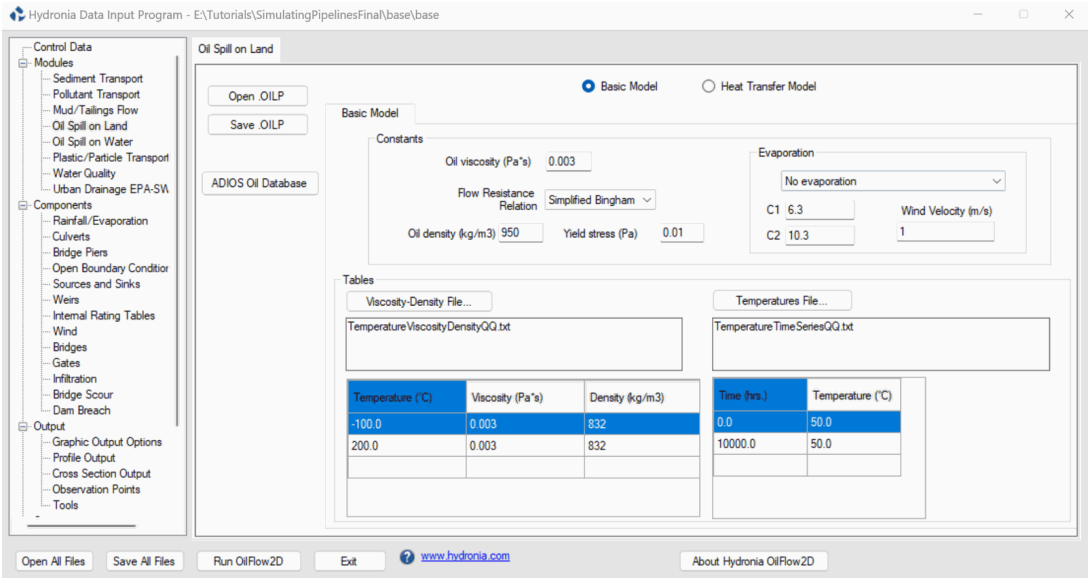
4. Browse to the 'SimulatingPipelines' folder and select the 'PipelineTutorial.OILP' then click [OPEN].

   The *Oil Spill on Land* window should look like the figure below:



   QGIS DIP Oil Spill on Land panel for Simulating Pipelines.

   The following Classic DIP Oil Spill on Land panel is retained as a legacy values reference:



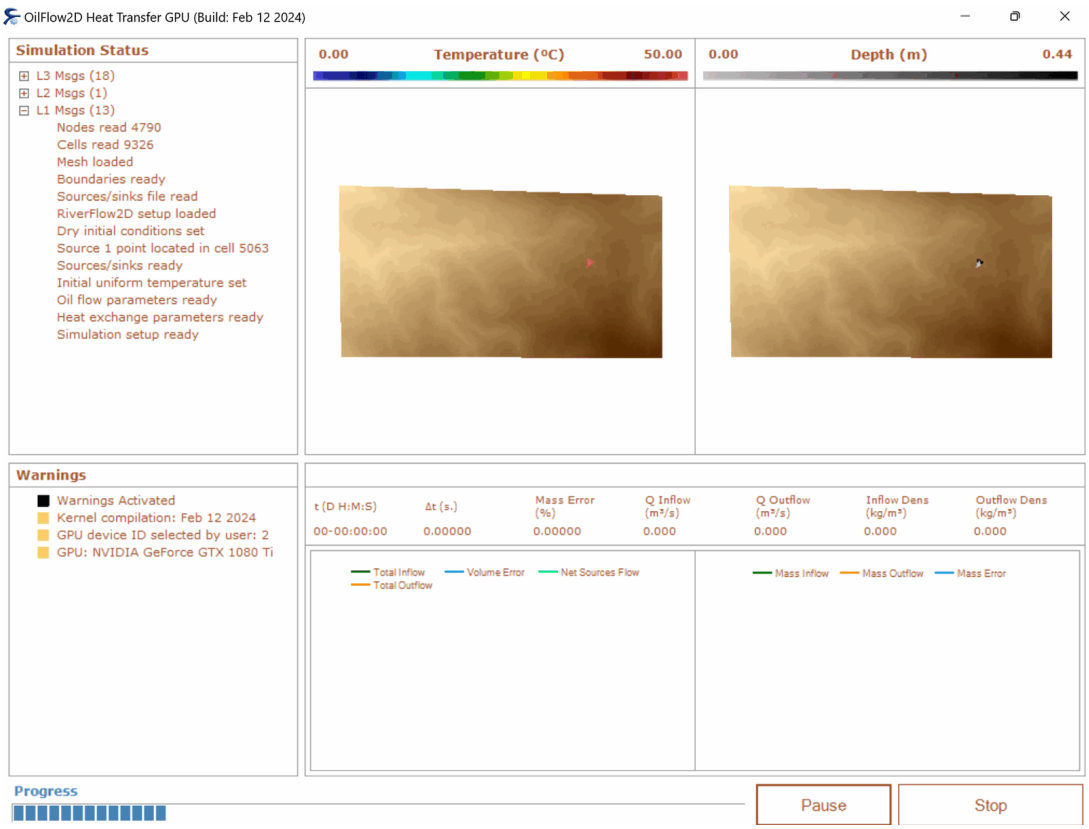
Legacy Classic DIP Oil Spill on Land parameters.

5. Click the [Save .OILP] button. Keep the default name (The name of the scenario in your case). Click [Save].

6. Return to the *Control Data* pane and Click [Run OilFlow2D].

7. When asked to save the DAT file, Click [Yes] and click [Save], making sure it is in the default project directory with the name provided.

Multiple model windows will appear and close automatically during the simulation process, they will look like the following figure:



OilFlow2D model window.

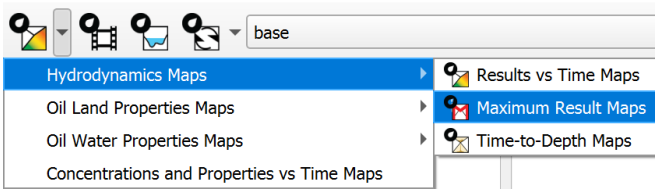
When this process ends, the windows will all be closed automatically. The model has finished simulating all sources.

5.8 Creating maps of all sources from results data

The OilFlow2D plugin for QGIS can create a single layer with the results of the multiple pipeline breaks. In this example a map of the combined spill affected area will be created. This can be performed with the following steps:

1. In the *OilFlow2D Maps* dropdown button in the OilFlow2D toolbar, click on *Hydrodynamics Maps* and then *Maximum Result Maps*.

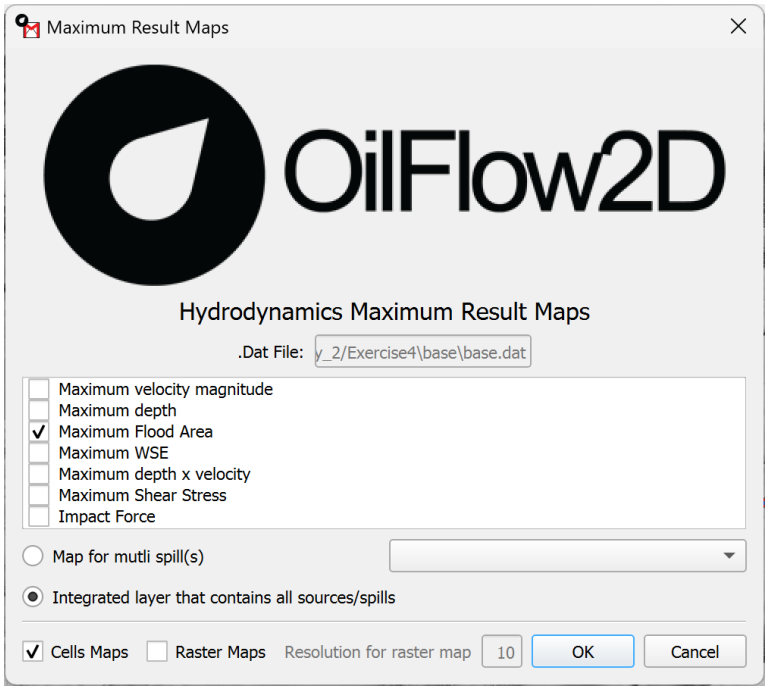
2. Click the [. . .] button to specify the *Maximum Value File*. A dialog will appear.



image

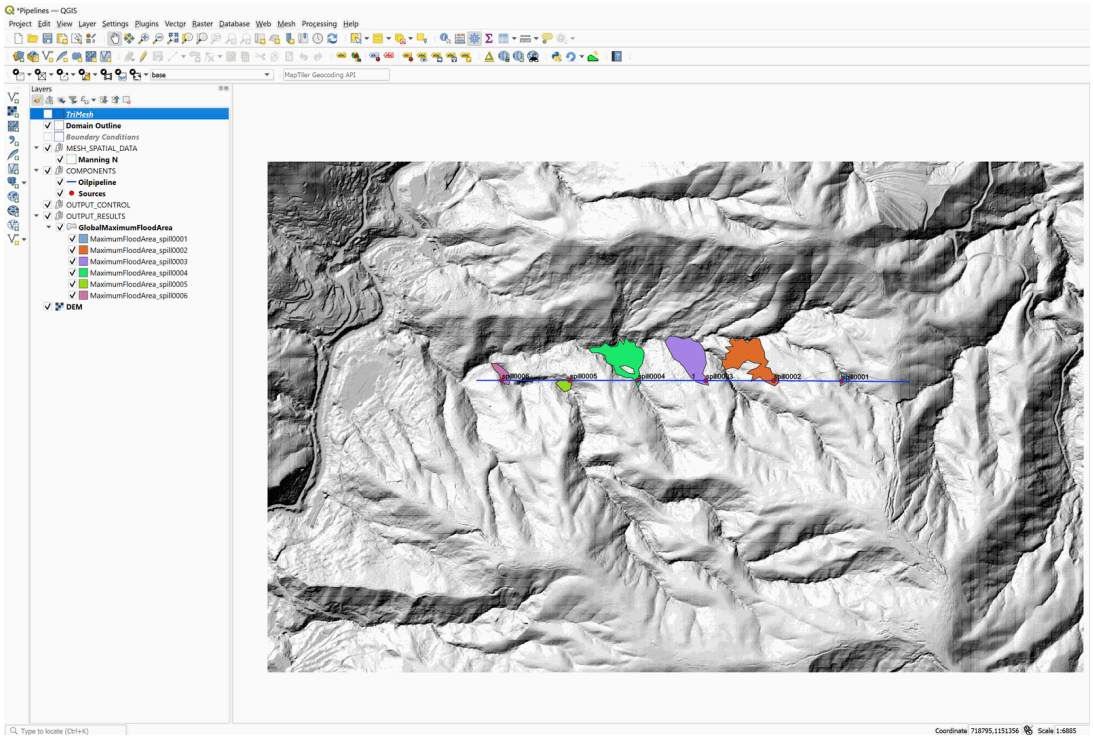
3. Browse into the newly created results folder, named ‘spill0001’. Select the file with your project name _cells_max.textout. In this example the file is named ‘Pipelines_cells_max.textout’. Click [Open].

4. A list will appear in the *Maximum Result Maps* window. Click the checkbox for *Maximum Spill Area*, and select the radio button for *Integrated layer that contains all sources/spills*, then click [OK].



Maximum Result Maps window.

After classifying the spill-area polygons by the layer attribute that corresponds to the spill point, the final result should look like the following:



Final integrated map of all spill areas.

This concludes the Simulating Oil Pipeline Tutorial.

6

Simulating overland oil spills from a tank break

This tutorial explains how to use the OilFlow2D model to simulate overland oil spills from a tank break. The steps to accomplish this are as follows:

1. Start QGIS and create a new *OilFlow2D* project.
2. Adding the terrain elevation data.
3. Create the limits of the modeling area (*Domain Outline*).
4. Setting the boundary conditions.
5. Assigning Manning's n roughness coefficients.
6. Using the Weirs layer to create the tank.
7. Filling the tank with oil utilizing the InitialWSE layer.
8. Generating the triangular mesh and exporting the data files to the OilFlow2D model.
9. Configuring parameters in the Hydronia Data Input Program.
10. Creating OilFlow2D maps and animations from results data.

The files required to follow this tutorial can be extracted from the 'ExampleProjects' zip file under the 'SimulatingTankbreaks' folder. This zip file is downloaded separately from your installation materials. The folder contains the files shown:

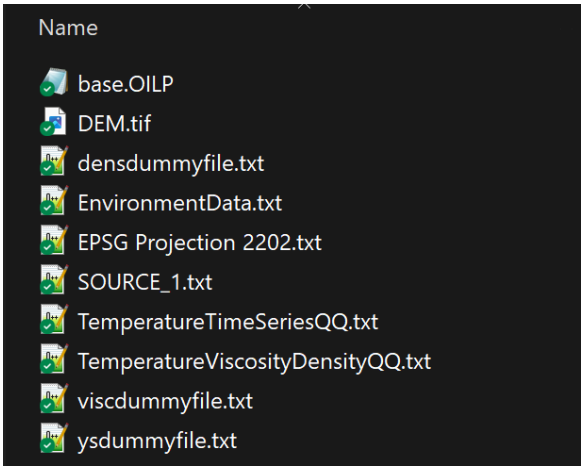


Figure 6.1 – Files with data required in this tutorial.

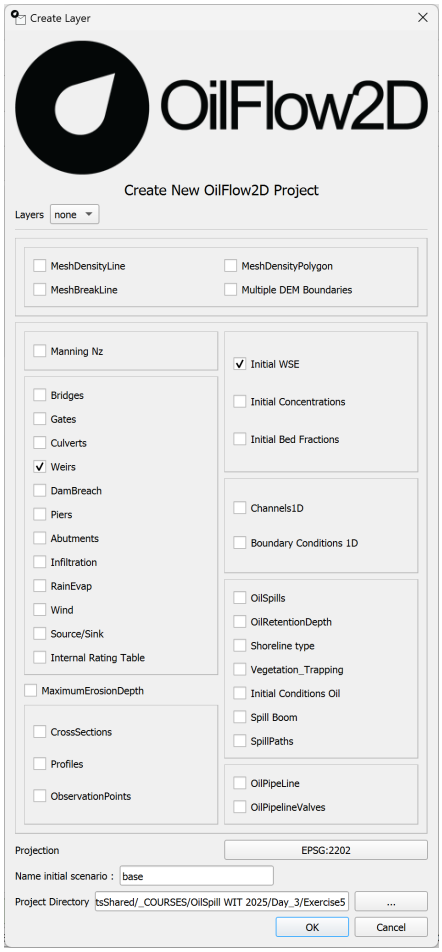
6.1 Start a new OilFlow2D project

1. To create a new OilFlow2D project, click on the *New OilFlow2D Project* button in the toolbar. A dialog window appears where you select the layers that will be created, the Coordinate Reference System (CRS), and the directory path where the layers will be saved. Select *None*, then enable *Domain Outline*, *Manning N*, *BoundaryConditions*, *InitialWSE*, and *Weirs*. This tutorial uses the *InitialWSE* layer to define the oil level and the *Weirs* layer to outline the tank.

2. Select the Projection: Coordinate Reference System (CRS) EPSG code: 2202 corresponding to *REGVEN / UTM zone 19N - Projected*

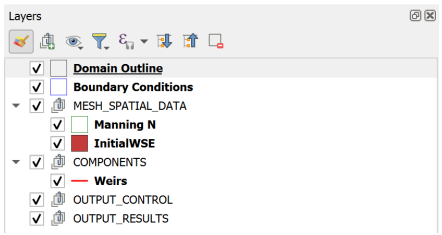
3. Use the same working Project Directory:

   ExampleProjects\SimulatingTankbreaks



Create New OilFlow2D Project.

4. After clicking OK, the templates of the layers are created, and displayed on the *Layers Panel*:



Layers created for the project.

OilFlow2D will use the unit system defined in the projection you selected. If the projection has coordinates in feet, units will be set to English. If the projection coordinates are in meters, units will be set to Metric/SI.

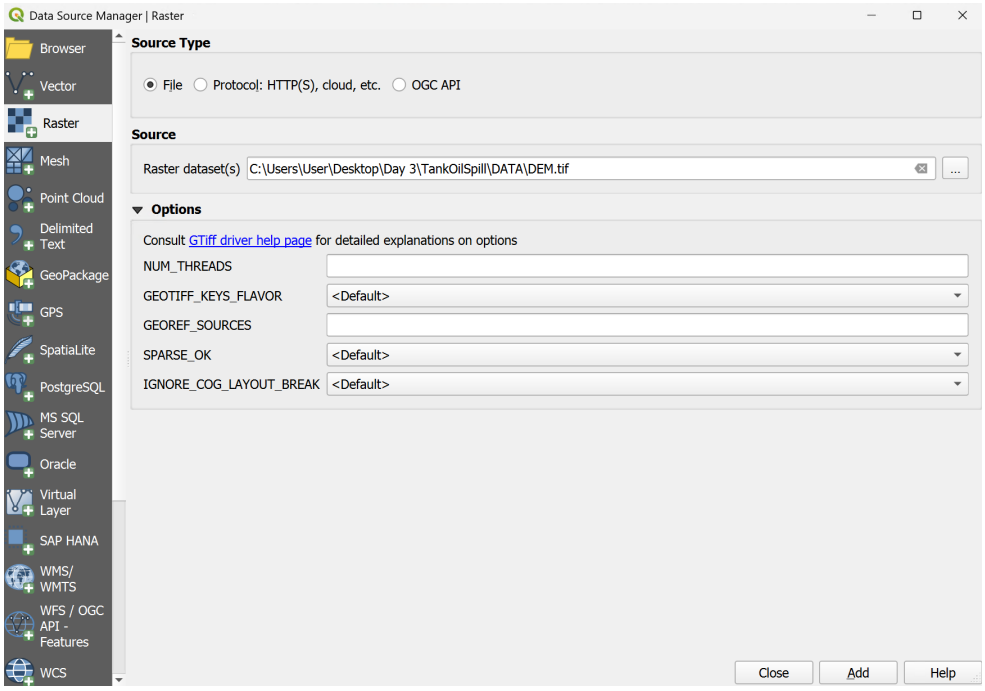
6.2 Load elevation data

OilFlow2D uses elevation data in raster format.

1. To load the raster elevation file, from the *Layer* menu, click *Add Layer*, and then click *Add*

Raster Layer…. You may also click the *Add Raster Layer* button  .

The following dialog will appear where you can select the raster file containing elevations. In this case, select DEM.tif from the extracted SimulatingTankbreaks folder.

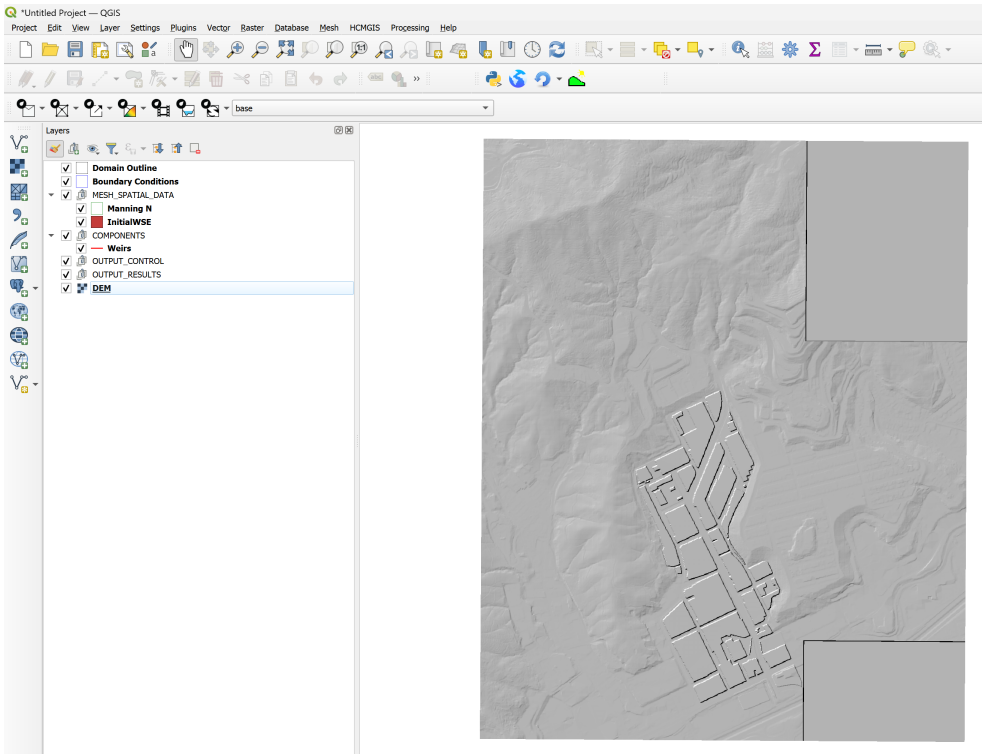


Raster data import dialog.

2. Click Add, then click Close on the dialog and click on the Zoom Full button  or by right-clicking the DEM layer and selecting *Zoom to layer*.

3. The raster will be displayed on the screen, by default it is rendered in gray gradient and will need to be changed in the layer properties by *right-clicking* the DEM layer and selecting *Properties*. Under *Symbology*, select the drop down menu for *Render Type* and select *Hillshade*, then click OK.

The layer should look like the following figure:



Digital elevation model in raster format.

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

4. Save the QGIS project by clicking on the Save Project icon in the toolbar. Make sure it is saved in the same directory that you previously selected in the *Create New Project* dialog above.

6.3 Create the limits of the modeling area (Domain Outline)

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

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



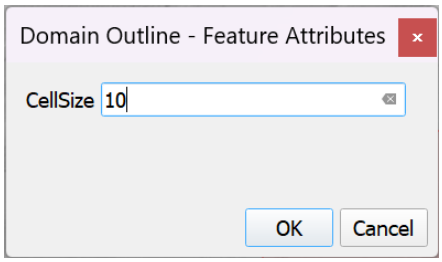
Menu buttons for digitalization toolbar.

2. Clicking on the button activates the rest of the editing buttons. Now click the *Add Polygon Feature* tool which is the bean looking polygon .

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

Make sure that the polygon is contained within the limits of the raster layer since OilFlow2D will not extrapolate elevations to areas that are outside of the available data on the raster layer.

3. To finalize and close the polygon, right-click anywhere on the screen. A dialog window to input the cell size attribute of the newly created polygon. The value for the reference size of the mesh cell is indicated. Enter a value of 10 m.



Input of attributes for the Domain Outline layer.

If you want to make any correction to the polygon, use the *Node* tool  to move, add or remove nodes as necessary.

4. Save the polygon by clicking the *Save* button  .

and click on *Toggle Editing* icon  to deactivate the layer edit mode and complete the *Domain Outline*.



Figure 6.2 – Domain Outline layer.

6.4 Setting the boundary conditions

To define the boundary conditions:

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

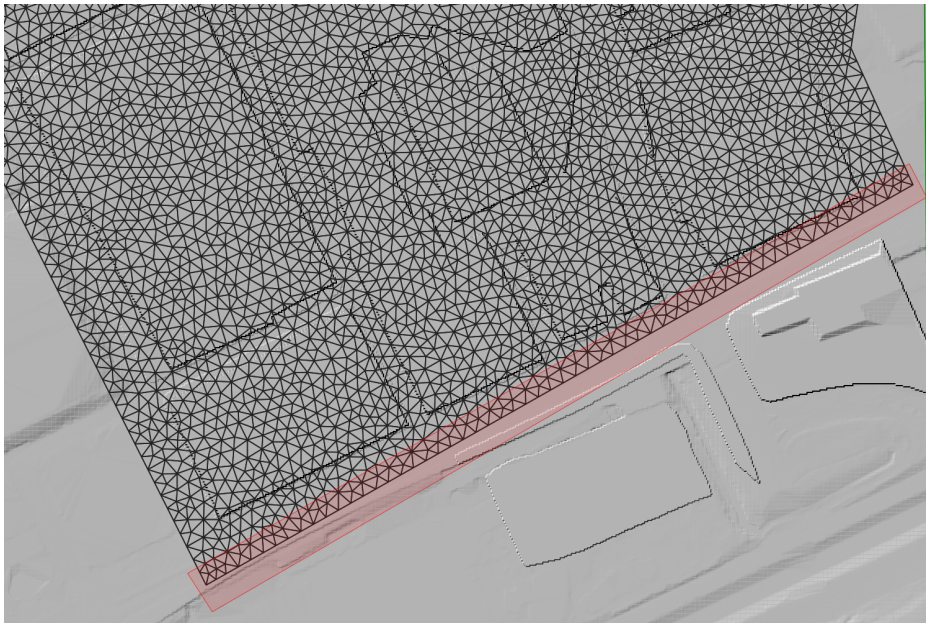


Figure 6.3 – Boundary-condition polygon at the lower end of the mesh.

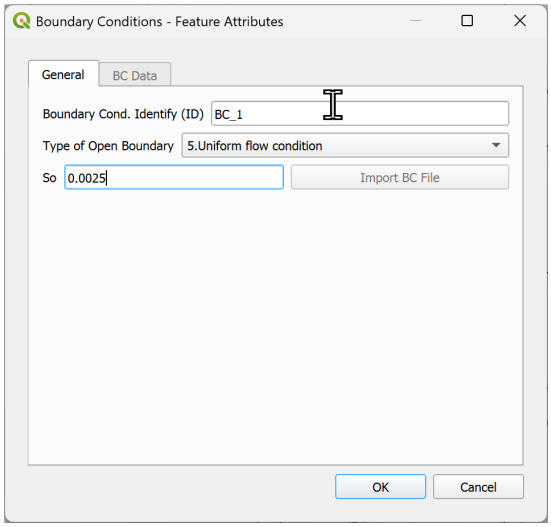
1. Select the *BoundaryConditions* layer in the Layers panel.

2. Click the Toggle Editing button  to add the polygons that are going to indicate the nodes on which the outflow conditions are established.

3. To finish the polygon, right-click on desired location. A window to enter the attributes of the newly created polygon is displayed.

4. The window contains a list to input the ID of this BC (*Boundary Cond. ID*). Click the dropdown menu for *Type of Open Boundary* select the boundary condition Type *5.Uniform flow condition*. Enter the value 0.0025 in the *So* field.

   The window should appear as shown in Figure below:



Uniform flow Boundary Condition parameters.

5. Click OK to close the dialog box.

6. Click the Save icon  and then click the Editing Tool icon  to deactivate editing mode.

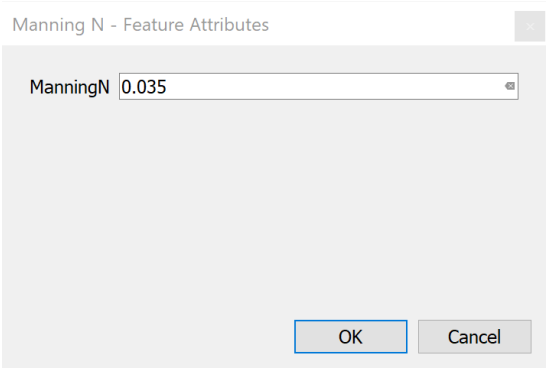
6.5 Assigning Manning's n roughness coefficients

Manning's coefficients are used in OilFlow2D to represent the terrain roughness. To assign Manning's n values, we will enter polygons with particular n's. There can be as many polygons as required to reproduce the spatial variability of this parameter. In this example, a single polygon will be used for the entire area.

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

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

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



Dialog to input Manning N Feature Attributes.

4. Click the Save icon  and then click the Editing Tool icon  to deactivate editing mode.

6.6 Using the Weirs layer to create the tank

In this tutorial we assume that the spill occurs in a tank. We will use the Weirs layer to draw the tank in the northern part of the domain.

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

2. Now click the *Add Line Feature* tool  .

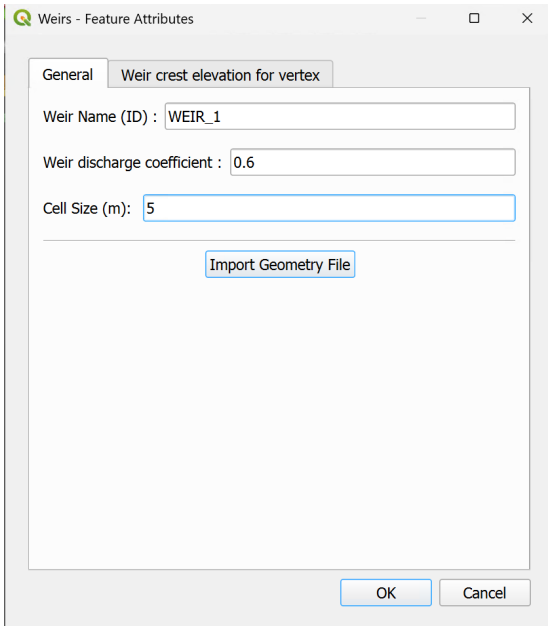
3. Left-click to create the vertices of the tank, starting from the bottom left corner and moving in the direction shown in the figure.

4. Right-click to close the polyline, ensuring that there is a gap between the last and first vertices as shown in the figure below:



Weir polyline, drawn counter-clockwise.

5. Then enter the *Cell Size* attribute, in this case we will use 5 m:

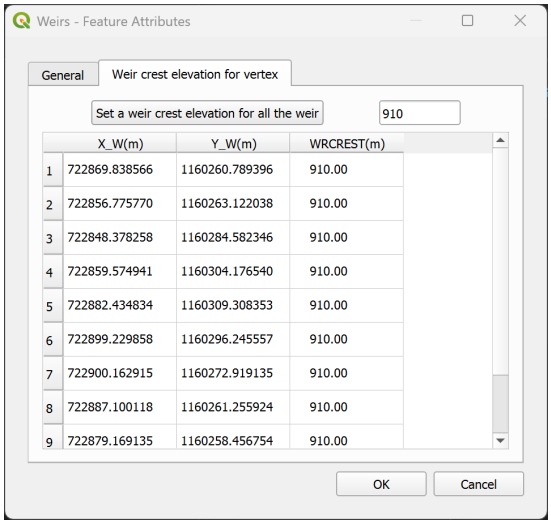


Weir feature attributes dialog, General tab.

6. Click on the tab *Weir crest elevation for vertex*, enter the number as shown in the following figure before clicking *Set a weir crest elevation for all the weir*

   This will populate the WCREST column with the value entered.

To determine the appropriate weir crest elevation, first use the *Identify Features* tool on the DEM to find the ground elevation where the tank will be placed. The weir crest elevation should be that ground elevation plus the desired height of the tank wall (e.g., ground elevation + 10m).



Weir feature attributes dialog, Weir crest elevation for vertex tab.

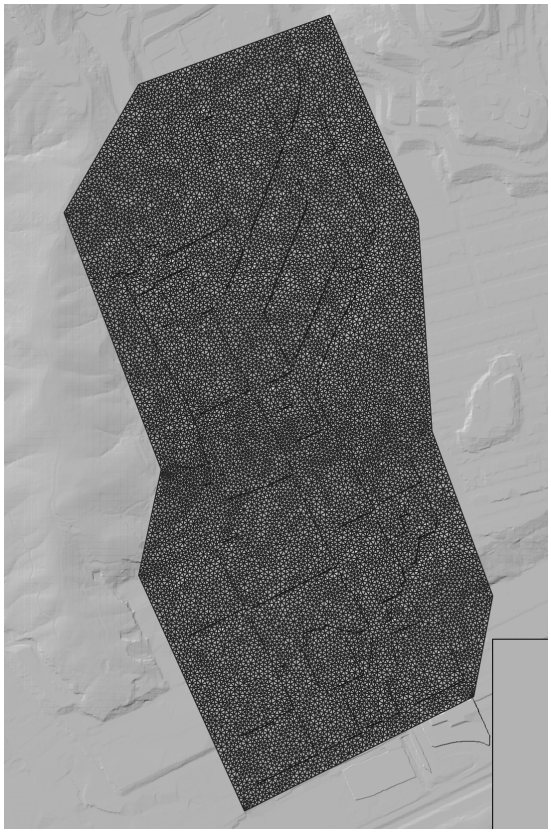
7. To complete entering the Weir, click Save  and then the *Toggle Editing* icon (pencil)  .

6.7 Generating the triangular-cell mesh

Now that the *Domain Outline* layer has been created, proceed to create the mesh.

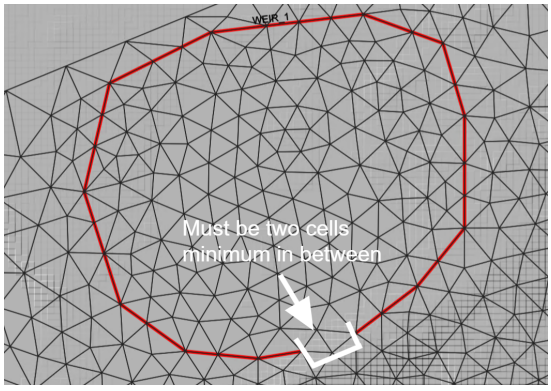
1. To generate the mesh, click on the icon  .

   The following figure shows the generated mesh. You will also find the Layers panel one new layers: *Trimesh* :



Mesh generated.

Inspect the weir feature to ensure that it has a minimum of two cells in between the first and last vertices.



Weir gap with generated mesh.

6.8 Filling the tank with oil utilizing the *InitialWSE* layer

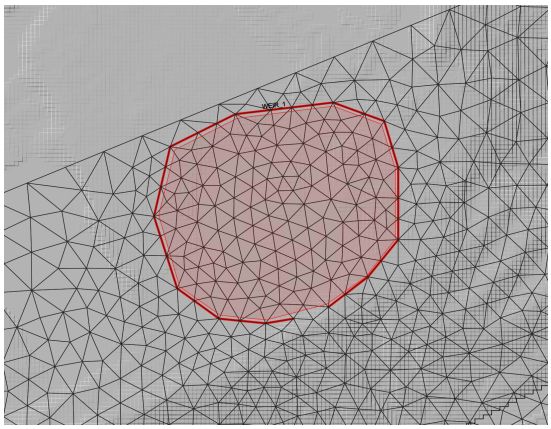
This layer is used to define the initial water surface elevation within the created polygon, in this tutorial it will be used to fill the tank with oil.

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

2. Now click the *Add Polygon Feature* tool  .

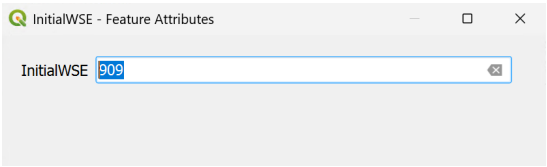
3. Left-click inside the *Weirs* polygon, staying close to each vertices to create the internal polygon of the tank.

4. Right-click to close the polygon.



Initial water surface elevation polygon.

5. A dialog window is presented where you must input the initial water surface elevation, in this case we will use 909 m. *This value may need to be different depending on where you place the tank within the domain.*



InitialWSE feature attributes dialog.

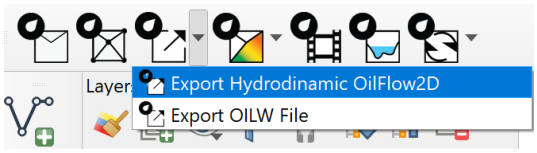
6. To complete entering the InitialWSE, click Save  and then the *Toggle Editing* icon (pencil)  .

6.9 Exporting the files to OilFlow2D

Once the layers with the input information to the model have been created, the next step is to run the tool that exports the data files in the proper format required by OilFlow2D. The process is as follows:

1. Make sure all of your required layers are active before performing this step.

2. Run the *Export Hydrodynamic OilFlow2D* plugin:



Export Hydrodynamic OilFlow2D command.

3. A dialog window is presented, where we will indicate the raster layer with the Digital Elevation Model (DEM), as this layer is not created by the plugin and its name may be arbitrary.

4. Make sure to indicate a file name (e.g. Oilspill) including the path where the generated files will be stored, and click OK.



Window dialog of the Export to OilFlow2D tool with the example data.

While exporting the data, you will be able to see the task progress at the top of the message bar. Once the process of creating the files is completed, the Hydronia Data Input Program (DIP) is opened automatically and a dialog window is presented with the model project to run. In this case: 'base.DAT' should already be set.

After export completes, version 10 and later users see the Open DIP dialog. Keep QGIS DIP selected and click **OK**. To use the historical standalone option, expand **More options**, select **Classic DIP**, and click **OK**.

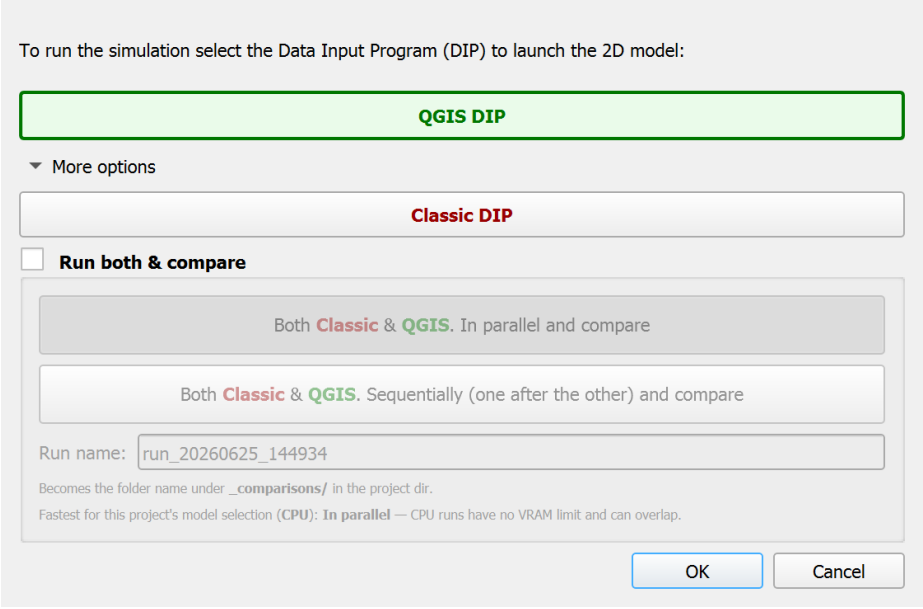


Figure 6.4 – Open DIP dialog showing QGIS DIP selected by default and More options expanded with Classic DIP.

The OilFlow2D input settings are then available in QGIS DIP:

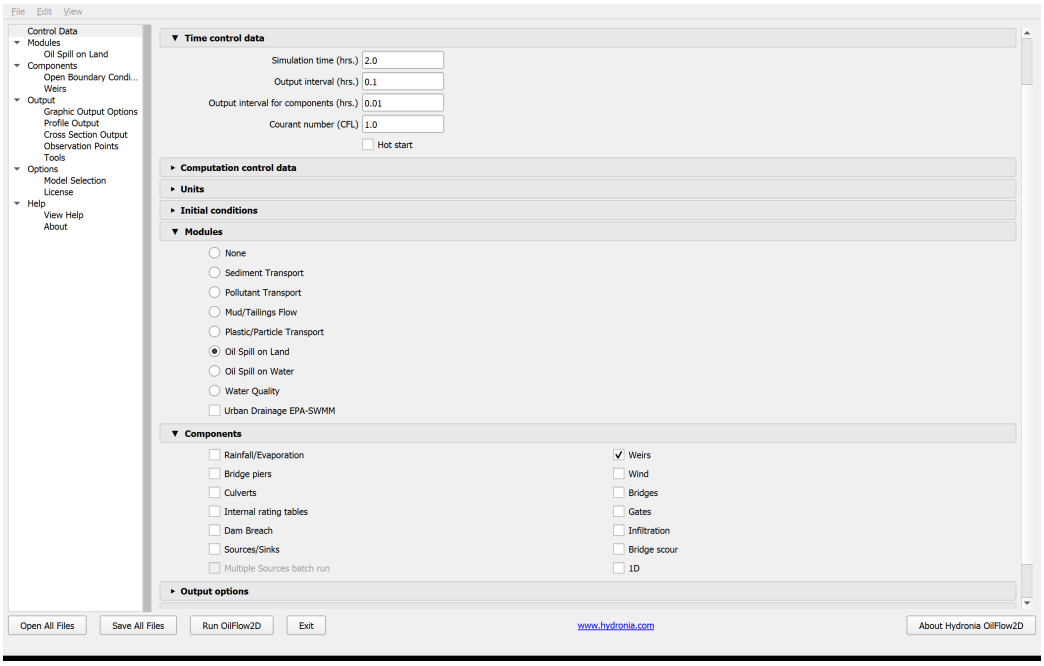


Figure 6.5 – QGIS DIP Control Data panel for Simulating Tank Breaks.

The legacy configuration figure below is retained as a values reference.

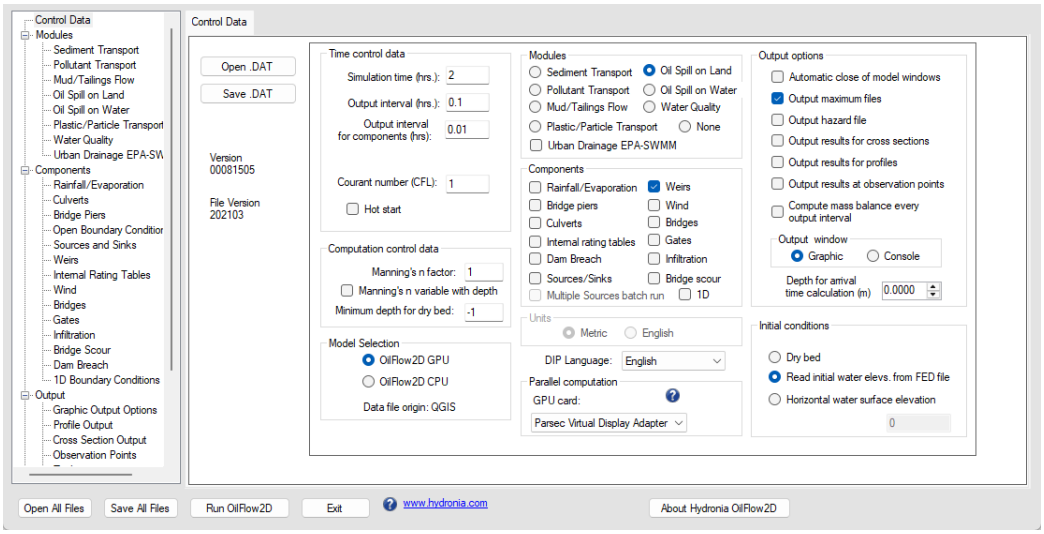
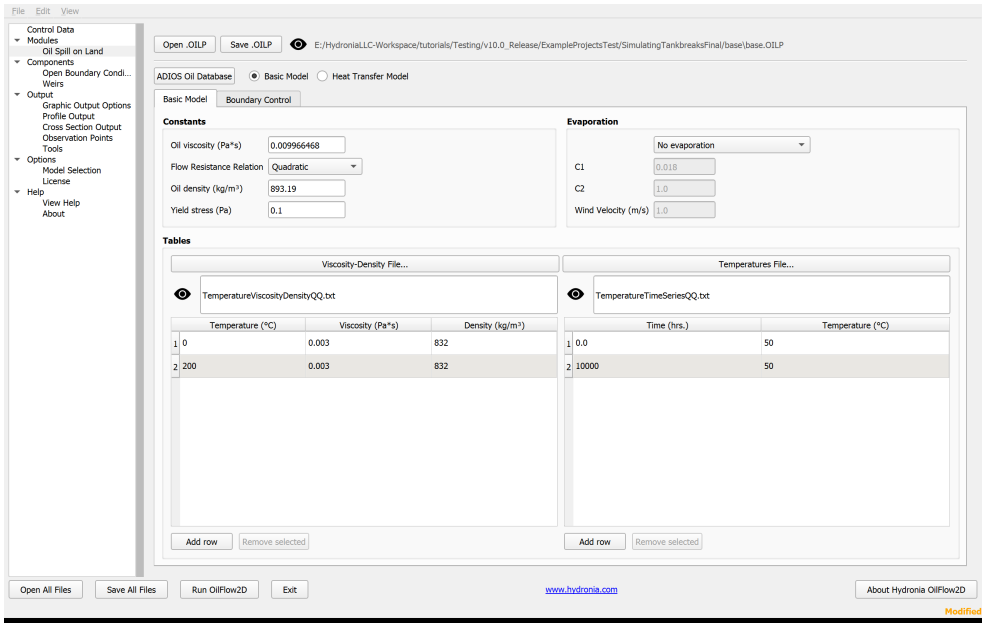


Figure 6.6 – Legacy configuration reference for the Hydronia Data Input Program.

5. Ensure that the *Oil Spill on Land* Module is selected, the Simulation time is set to 2 hours, and the *Output Interval* is set to 0.1.

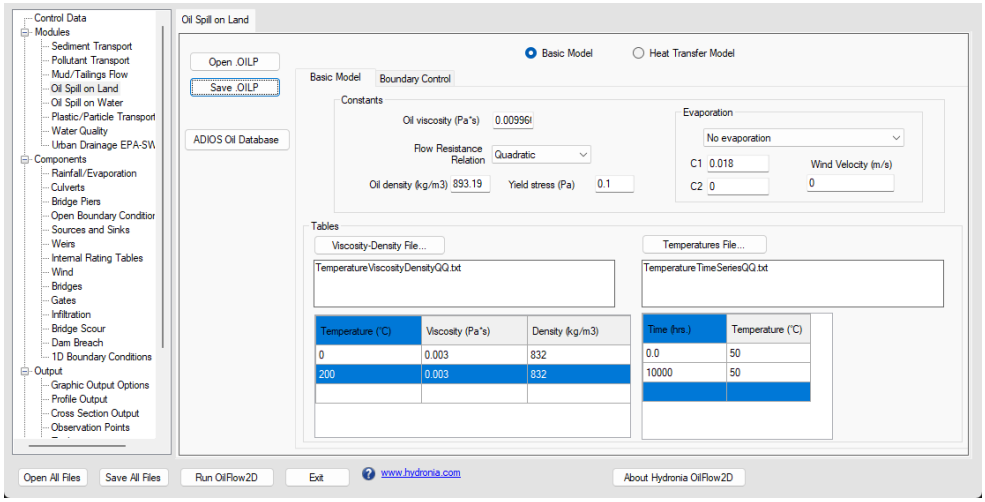
6. Click **Save .DAT** and overwrite the 'base.dat' file.

7. Now click on the *Oil Spill on Land* panel, make sure *Basic Model* radial button is selected, and enter the data the following data: Oil viscosity: 0.009966468 Pa.s, Flow Resistance Relation Quadratic, Oil density: 893.19 kg/m^3, and Yield stress: 0.1 N/m^2. In the Tables section, make sure to fill in the values as seen in the figure below:



QGIS DIP Oil Spill on Land panel for Simulating Tank Breaks.

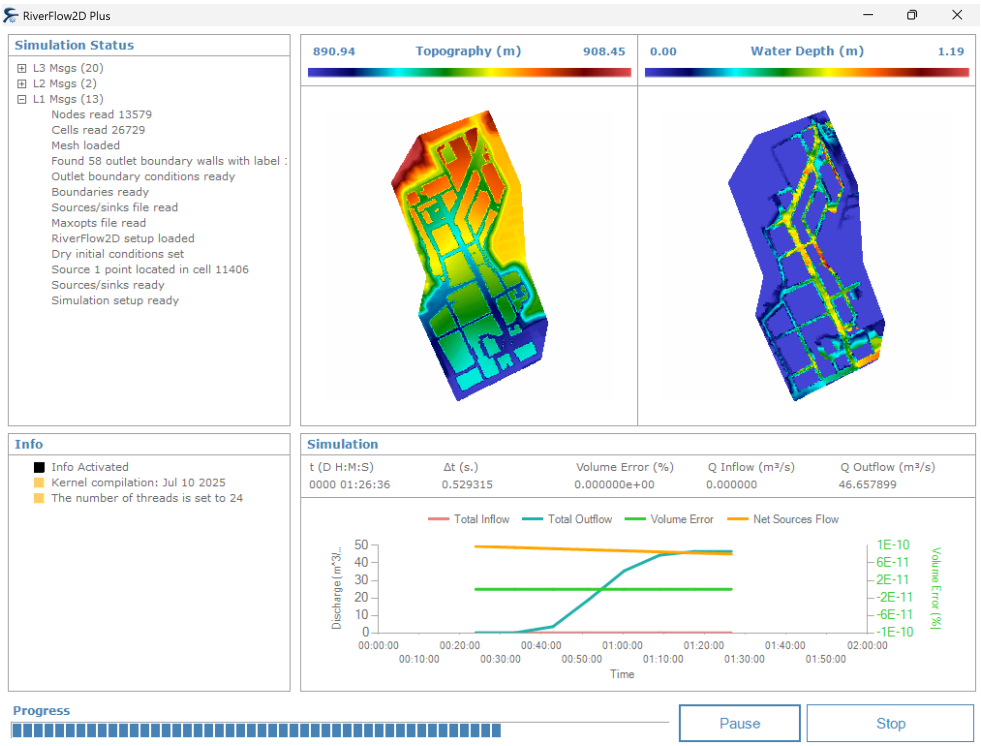
The following Classic DIP Oil Spill on Land panel is retained as a legacy values reference:



Legacy Classic DIP Oil Spill on Land panel.

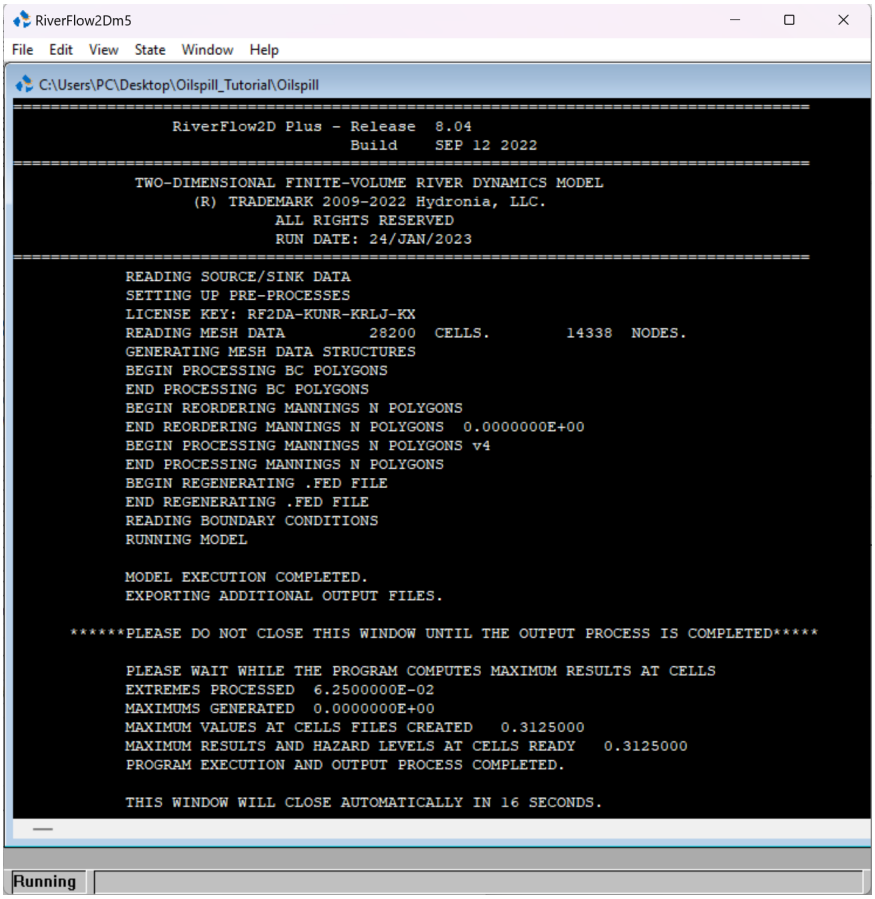
8. Click **Save .DAT**. This will create the TemperatureViscosityDensityQQ.txt and Temperature-TimeSeriesQQ.txt files in your scenario folder.

9. Click the *Run OilFlow2D* button to run the model. An image similar to the one shown below should appear:



The graph generated by the OilFlow2D model.

10. When you have finalized reviewing the information included in this window, close it. The following post processing window will appear indicating the final post-processed task performed by the OilFlow2D model. The window will close automatically.

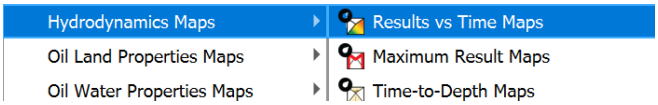


Final post-processes.

6.10 Creating OilFlow2D maps

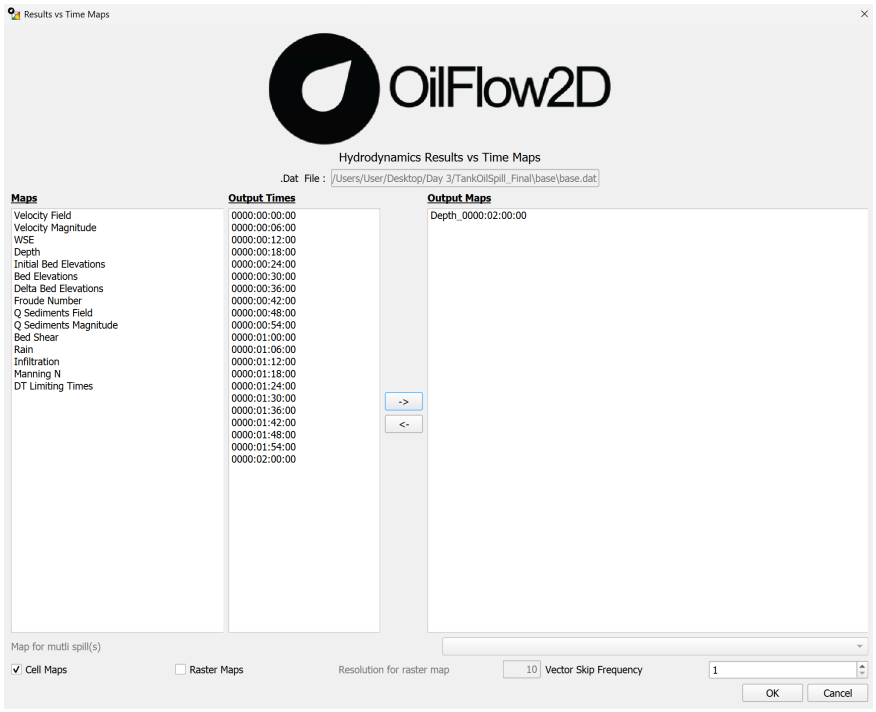
Once OilFlow2D has finished running, you may want to create maps using the model results. OilFlow2D QGIS plugin offers several tools to create maps and animations. In this section we will show how to create depth maps.

1. Click the *OilFlow2D Maps*  button dropdown menu and select *Results vs Time Maps*.



OilFlow2D Maps menu.

2. A dialog window is presented where we will indicate the output files list 'base.OUTFILES' and the desired parameter, that will be Depth for time 2 hours. Also indicate the Raster Map option as shown.



Window dialog of the Results vs Time Maps plugin.

3. Click *OK* to start the process. After a few seconds a new layer with the requested map will be created.

4. Right-click the newly generated layer and select *Properties*. In the *Properties* window, click on the *Symbology* tab and change the *Mode* to *Equal Count (Quantile)* option.



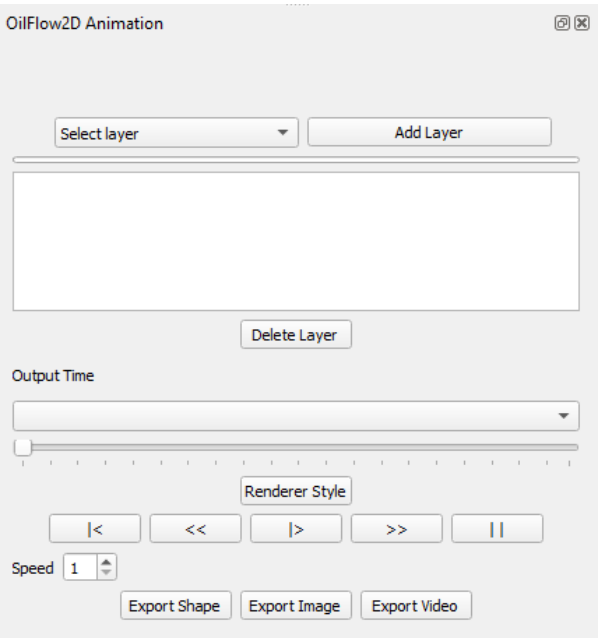
Oil depth map at 2 hours after the initial spill.

Save the project in QGIS to continue. On the QGIS *Project* menu, click *Save*, to save the project assigning the name SimulatingTankBreaks, in the same directory that you previously selected in the *Create New Project* dialog above. The saved project file will be SimulatingTankBreaks.qgz.

6.11 Animating oil spill on land results in OilFlow2D

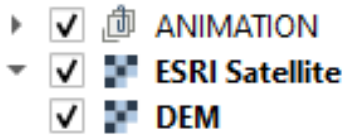
Sometimes it is better to view an animation of the results for a different audience, or to view all of the results quickly. Animations can also be exported to a movie file for maximum portability. To create an animation, complete the steps as follows:

1. In QGIS, click on the *Oil Animation Button*  in the OilFlow2D toolbar.
2. A panel will open on the bottom left side of the QGIS interface.

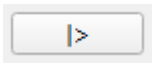


Output map options for Oil Properties vs. Time Maps.

3. Click on the *Select Layer* dropdown menu and select *Depth*. Click on the *Add Layer* button.
4. After the animation loads, it will be listed in the list window under the *Select layer* button.
5. in the *Layers* panel above, there will be a new group named *ANIMATION* which will contain the newly created *Depth* animation layer.
6. The layer needs to be moved above the DEM layer so that it is visible when being played. Click and drag the *ANIMATION* group and make sure to drop it above the layers that would need to be underneath it, such as the DEM and satellite imagery layers as in the following figure:



Layer order for Animations to be visible.

7. To play the animation, click the *Play*  button in the *OilFlow2D Animation* panel. Make sure all other output layers have been disabled.

The user can change the speed of the animation, export the animation as a shape file, export an image of a particular frame of an animation, and finally export a video via the buttons located at the bottom of the *OilFlow2d Animation* panel.
This concludes the tutorial: *Simulating overland oil spills from a tank break*.

7

Oil spills in Rivers. Part I: Hydrodynamics

This tutorial guides in the application of OilFlow2D to simulate oil spills in rivers. The first part explains how to setup and run the hydrodynamic stage, where the model is used to generate the velocity field in the river. The second part, included in a separate chapter, explains how to enter the spill locations, and other oil data to simulate oil trajectories and fate.

7.1 Tutorial data

Before starting the project with QGIS. It is recommended to create a folder that will contain all the necessary data such as the boundary condition files, the elevation data files, either in ASCII or in raster format, etc.

The files required to follow this tutorial can be extracted from the 'ExampleProjects' zip file under the 'OilSpillsInARiverPart1' folder. This zip file is downloaded separately from your installation materials.

The folder contains the files shown:

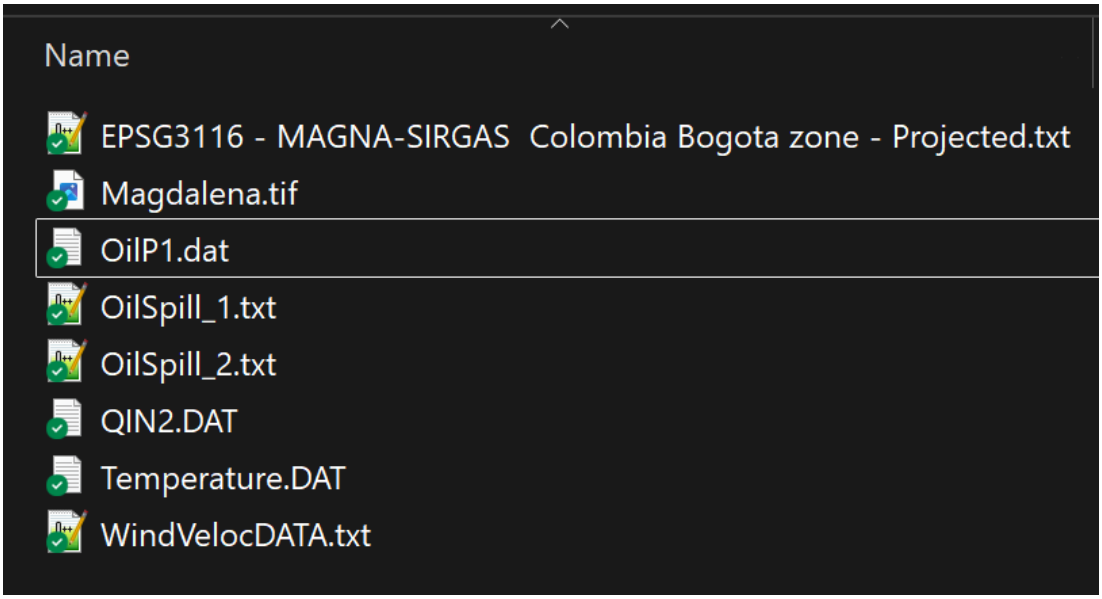


Figure 7.1 – Files with data required in this tutorial.

7.2 Starting QGIS

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

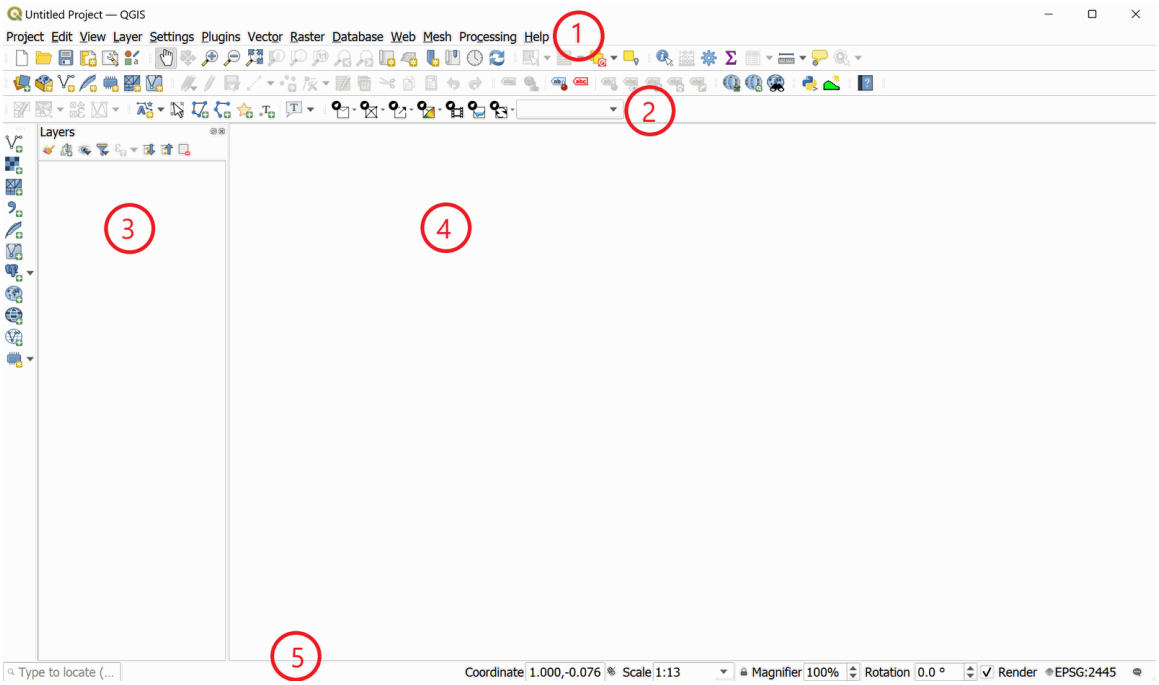


Figure 7.2 – QGIS interface for OilFlow2D indicating window areas. 1: Menu bar, 2: Tool bar, 3: Layer panel, 4: Maps view area, 5: Status bar.

Note the tool bar contains icons that give access to all the OilFlow2D components. Some of the

icons have a drop down list that allow getting to more tools.

7.3 Start a new OilFlow2D project

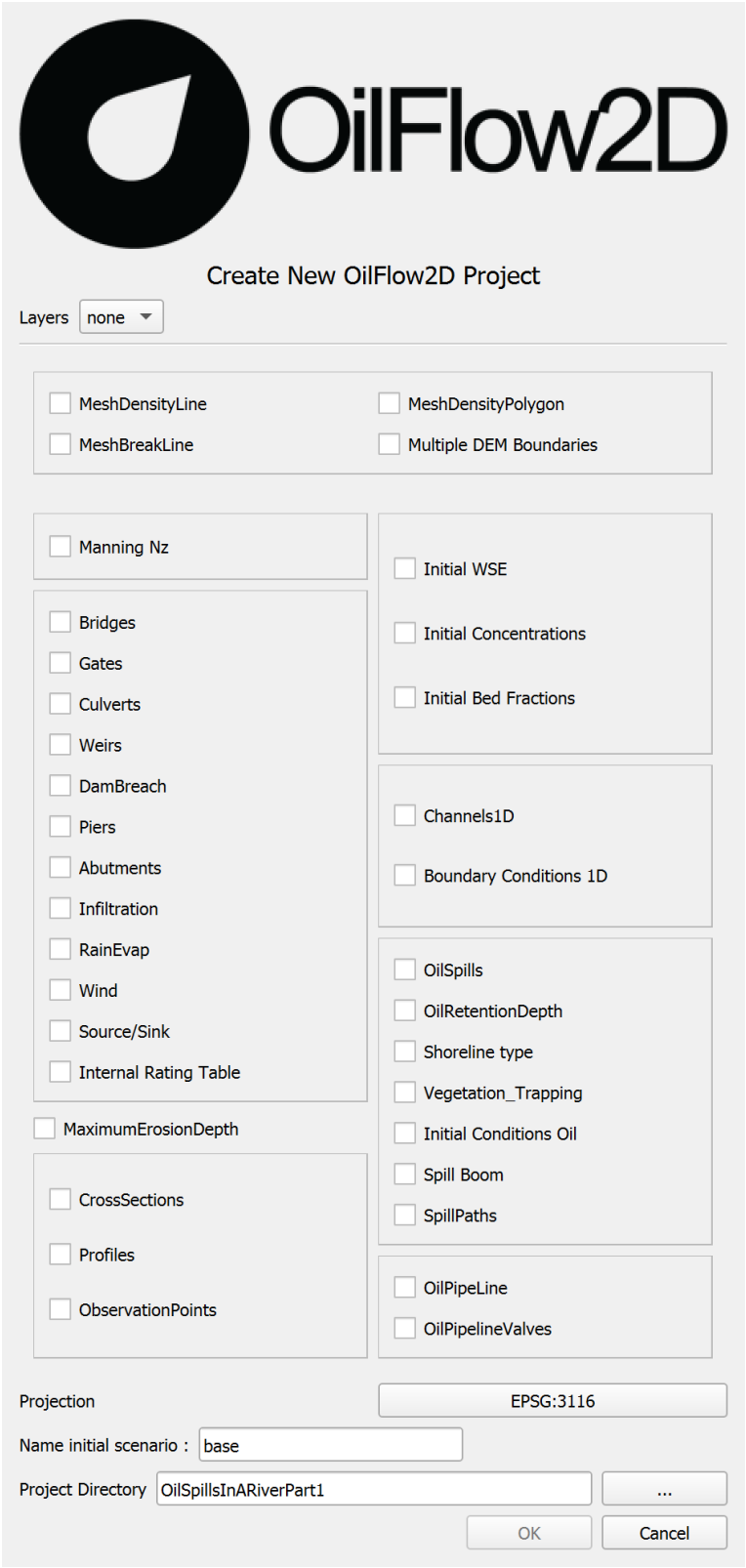
1. In the OilFlow2D toolbar, click on the *New OilFlow2D Project* button  to start a new OilFlow2D project. A dialog window appears where you select the optional component layers, the Coordinate Reference System (CRS), the initial scenario name, and the project directory.

2. Select *None* in the Layers drop down menu.

3. Select the *Projection* button. Click on the [EPSG] projection button and select: Coordinate Reference System (CRS) EPSG 3116 corresponding to *MAGNA-SIRGAS Colombia Bogota zone - Projected (meters)*, and click OK.

4. Keep *Name initial scenario* set to base. Click the […] button and choose the OilSpillsInARiverPart1 folder for the *Project Directory*. This is the folder where the model will write all results and output files.

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



Create New OilFlow2D Project configured for OilSpillsInARiverPart1.

OilFlow2D will use the unit system as that defined in the projection you selected. If the projection has coordinates in feet, units will be set to English. If the projection coordinates are in

meters, units will be set to Metric or SI.

6. On the QGIS *Project* menu, click *Save*, to save the project assigning the name OilwPart1, in the same directory that you previously selected in the *Create New Project* dialog above.

7.4 Load elevation data

OilFlow2D uses elevation data in raster format. To load the raster, from the *Layer* menu, click *Add Layer*, and then click *Add Raster Layer…*. You may also click the *Add Raster Layer* button  . Click the ... browse button and select *Magdalena.tif* from \ExampleProjects\OilSpillsInARiverPart1\DATA. Verify that the selected path appears under *Raster dataset(s)*.

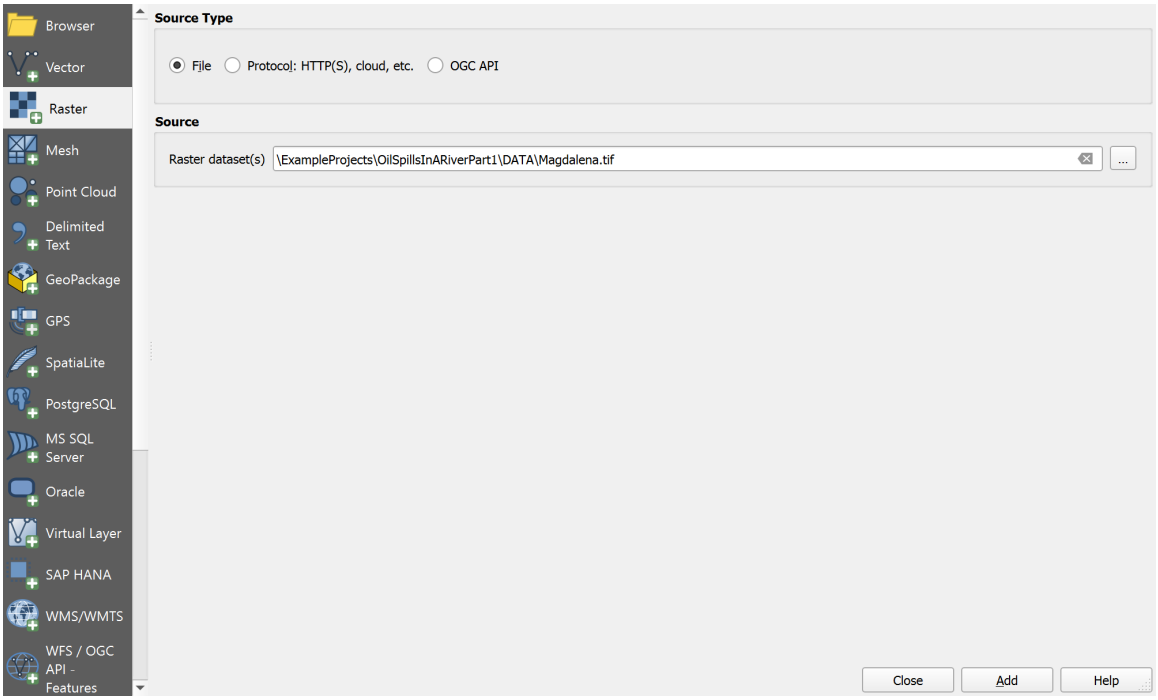


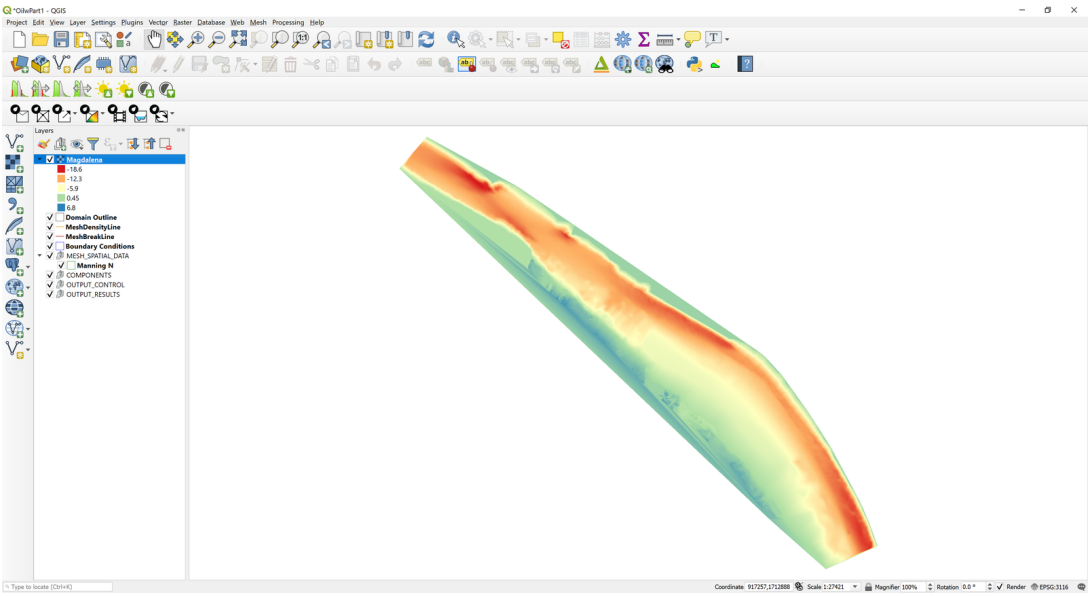
Figure 7.3 – Raster data import dialog.

1. Click *Add* and wait for the raster to load. Then click *Close* and click on the *Zoom Full* button  .

   The raster will be displayed on the screen, by default it is rendered in gray gradient.

2. Right-click on the label of the created layer and select *Properties* to change the rendering style for a more informative color palette.

3. In the *Symbology* tab, under *Render Type*, select Singleband pseudocolor. Under the *Color Ramp* dropdown menu, select *Spectral*. Then click the [Classify] button to create the classes and then click [OK] to close the *Layer Properties* window. The layer should look as follows:



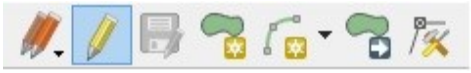
Digital elevation model in raster format created by interpolation.

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

7.5 Create the limits of the modeling area (Domain Outline)

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

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

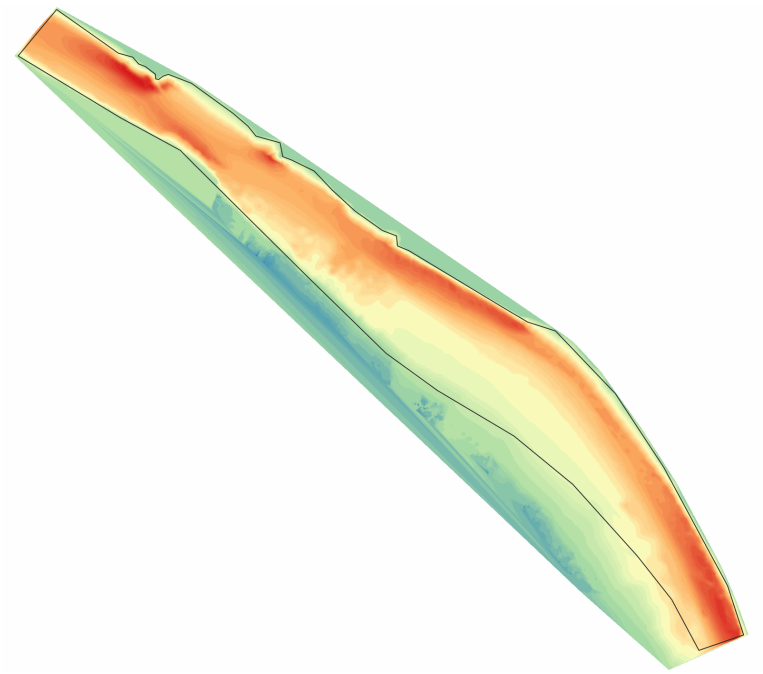


Menu buttons for digitalization toolbar.

2. Clicking on the button activates the rest of the editing buttons. Now click the *Add Polygon Feature* tool which is the bean looking polygon  .

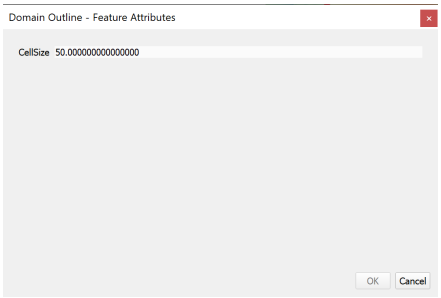
3. Proceed to delineate the outline of the polygon by marking the vertices clicking with the left mouse button.

   Make sure that the polygon is contained within the limits of the raster layer since OilFlow2D will not extrapolate elevations to areas that are outside of the available data on the raster layer.



Domain Outline layer.

4. To finalize and close the polygon, right-click anywhere on the screen. A dialog window to input the cell size attribute of the newly created polygon. The value for the reference size of the mesh cell is indicated. Enter a value of 50 m.



Input of attributes for the Domain Outline layer.

If you want to make any correction to the polygon, use the *Node* tool  to move, add or remove nodes as necessary.

5. Save the polygon by clicking the *Save* button  .

and click on *Toggle Editing* icon  to deactivate the layer edit mode and complete the *Domain Outline*.

7.6 Generating the triangular-cell mesh

Now that the *Domain Outline* layer has been created, proceed to create the mesh.

To generate the mesh, click on the icon  .

The following figure shows the generated mesh. You will also find the Layers panel one new layers: *Trimesh* :

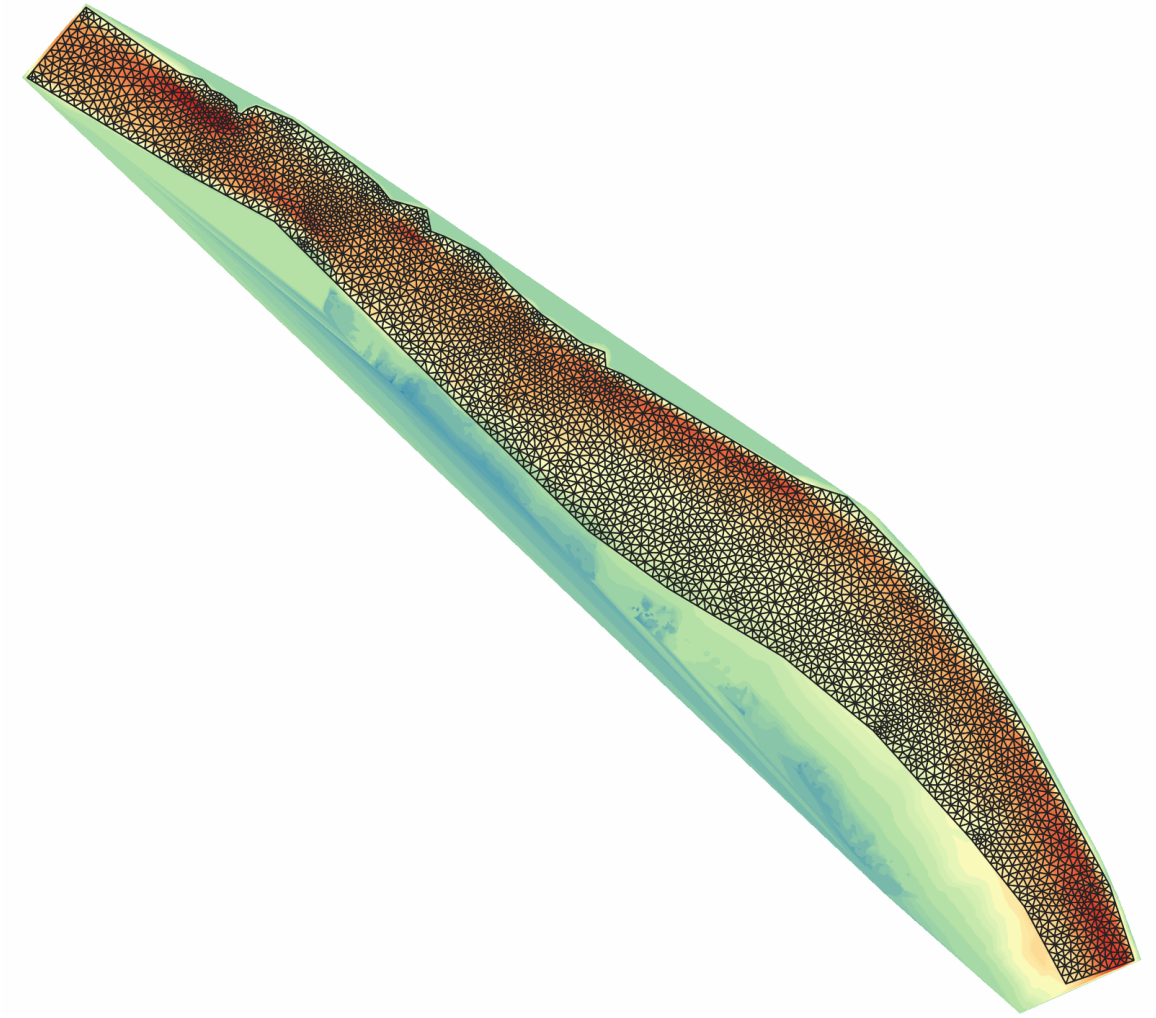


Figure 7.4 – Mesh generated.

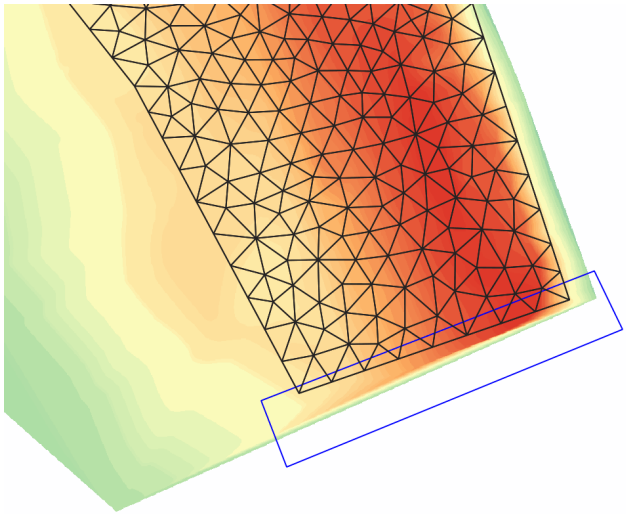
7.7 Setting the boundary conditions

In this tutorial we will define inflow and outflow conditions. To define the inflow boundary condition do as follows:



1. Enter the *INFLOW* boundary condition polygon: Click the Toggle Editing button to add the polygons that are going to indicate the nodes on which the inflow condition is imposed.

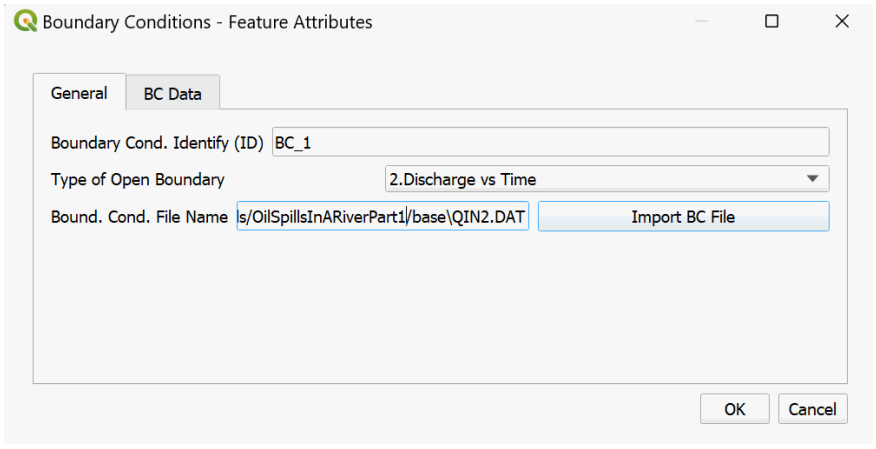
2. Draw a polygon that includes the nodes or vertices at the lower end of the mesh as indicated in the figure:



Polygon that covers the nodes defining the inflow boundary condition segment.

3. To finish the polygon, right-click on the last vertex. A window to enter the attributes of the newly created polygon is displayed. It is important that the boundary condition polygon would cover completely the outflow segment.

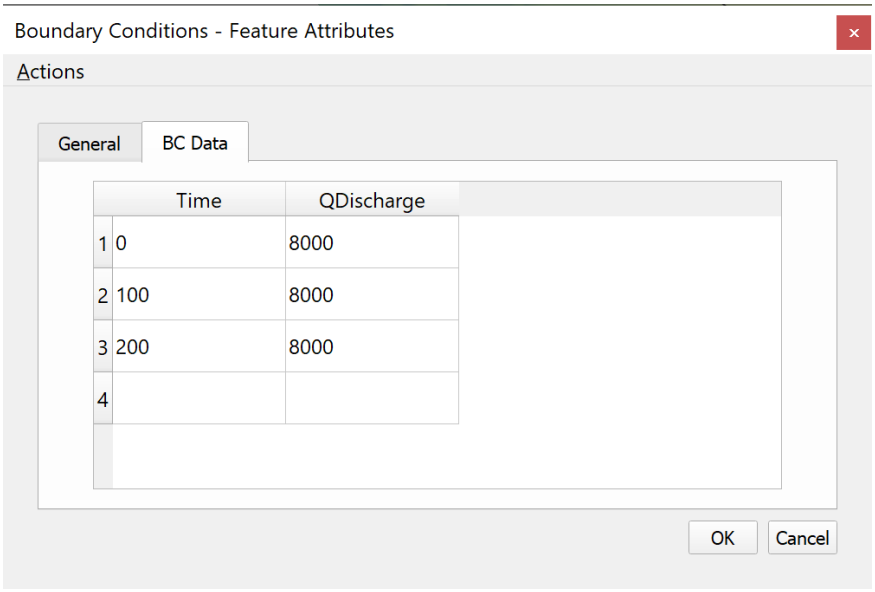
4. In the window change the *Type of Open Boundary* to *Discharge vs. Time*.



Inflow boundary condition data.

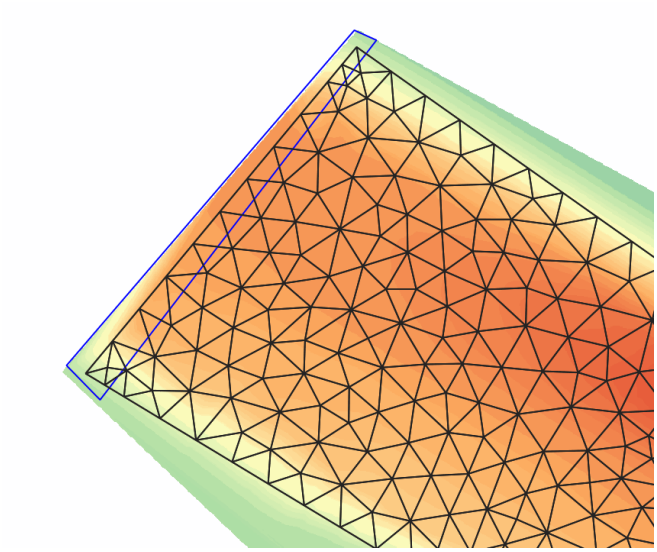
5. Click the *Import BC File* button and browse to the tutorial folder and select the 'QIN2.DAT' file, then click OK.

6. Ckick the *BC Data* tab. The data should look as follows:



Inflow hydrograph read from the 'QIN2.DAT' file.

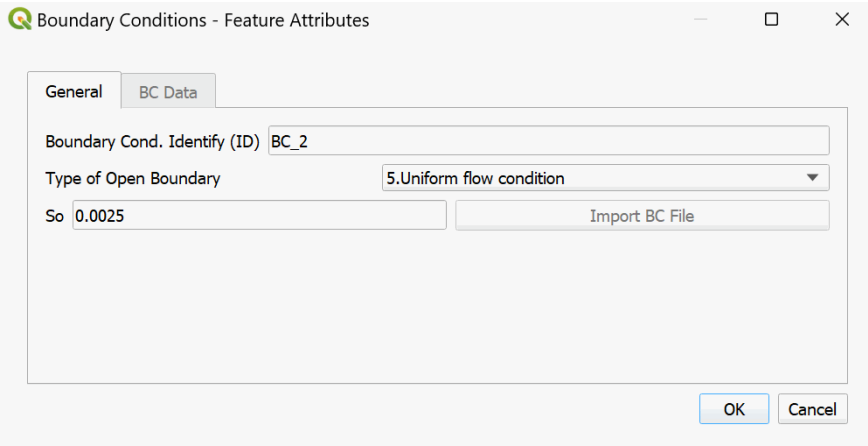
7. Enter the *OUTFLOW* boundary condition polygon: Draw a polygon that includes the nodes or vertices at the lower end of the mesh as indicated in the figure:



Polygon that covers the nodes defining the outflow boundary condition segment.

8. To finish the polygon, right-click on the last vertex. A window to enter the attributes of the newly created polygon is displayed. It is important that the boundary condition polygon would cover completely the outflow segment.

9. In the window set the boundary condition Type to: *Uniform flow condition*, then set the *So* to 0.0025.



Outflow condition data.

10. Click [OK] then click the *Save* button  to save the changes made to the layer:

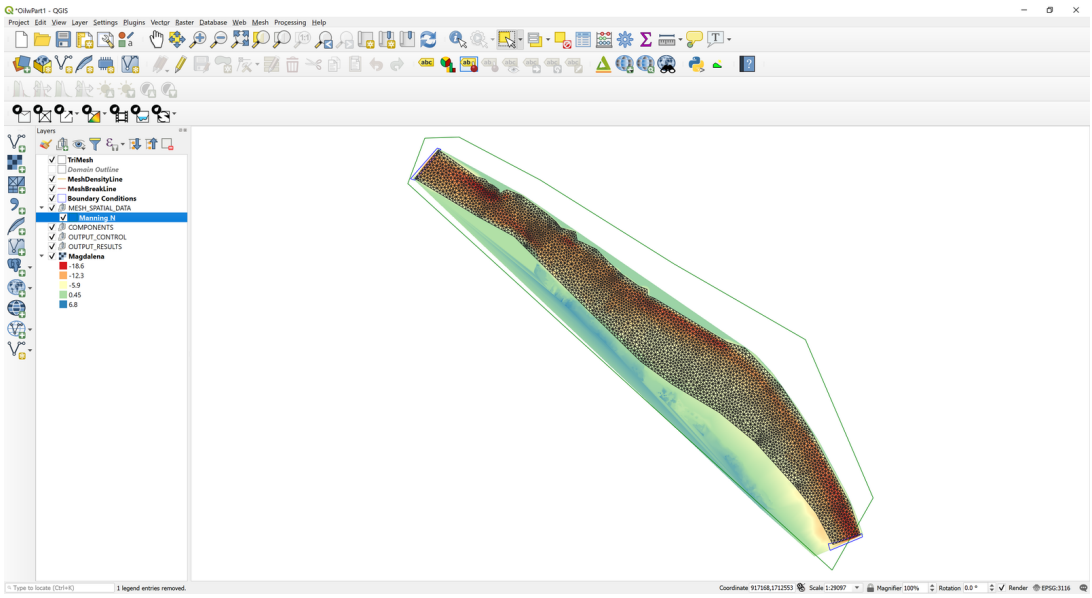
11. Deactivate editing mode by clicking on the *Toggle Editing* icon  .

7.8 Assigning Manning's n roughness coefficients

Manning's coefficients are used in OilFlow2D to represent the terrain roughness. To assign Manning's n values, we will enter polygons with particular n's. There can be as many polygons as required to reproduce the spatial variability of this parameter. In this example, a single polygon will be used for the entire area.

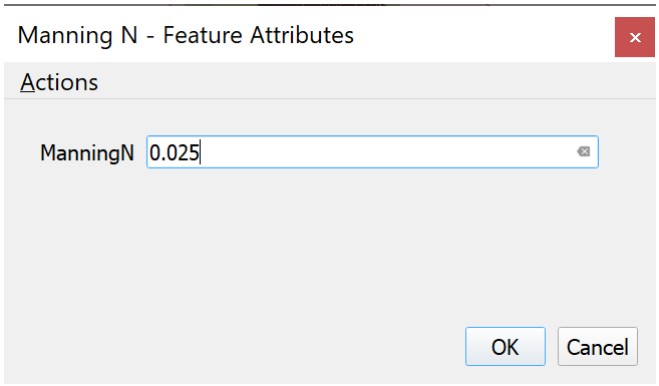
1. Select the Manning N layer and click the *Toggle Editing* button  .

2. Draw the polygon around the entire domain taking care that it covers all the cells. You should have an image like the one shown below:



Editing the Manning N layer.

3. Close the last vertices on the polygon by right-clicking on the desired position. The following dialog window is presented where you must input the Manning's n value associated to the polygon (0.025):



Dialog to input Manning N Feature Attributes.

4. Click the Save icon  and then click the Editing Tool icon  to deactivate editing mode.

7.9 Exporting the files to OilFlow2D and running the model

Once the layers with the input information to the model have been created, the next step is to run the tool that exports the data files in the proper format required by OilFlow2D. The process is as

follows:

1. From the QGIS menu bar, select *Plugins* > *OilFlow2D MS* > *Export OilFlow2D* > *Export Hydrodinamic OilFlow2D* to run the export plugin:



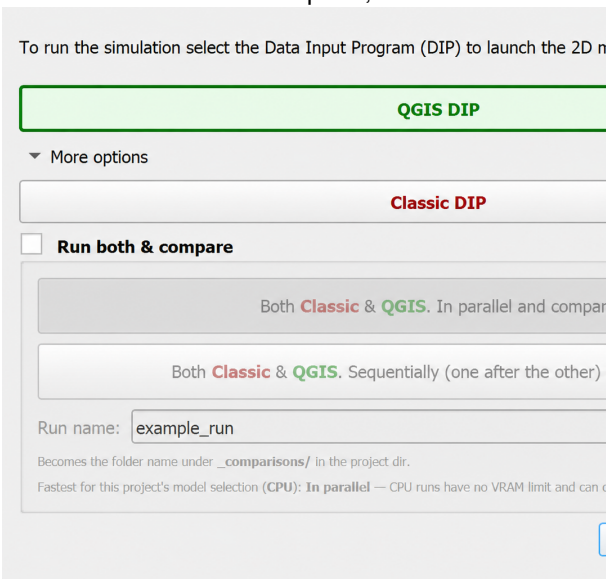
Plugins menu path to Export Hydrodinamic OilFlow2D.

2. In the *Export Files to OilFlow2D* dialog, make sure the option for *DEM (Single Raster)* has *Magdalena* selected. Click [OK].

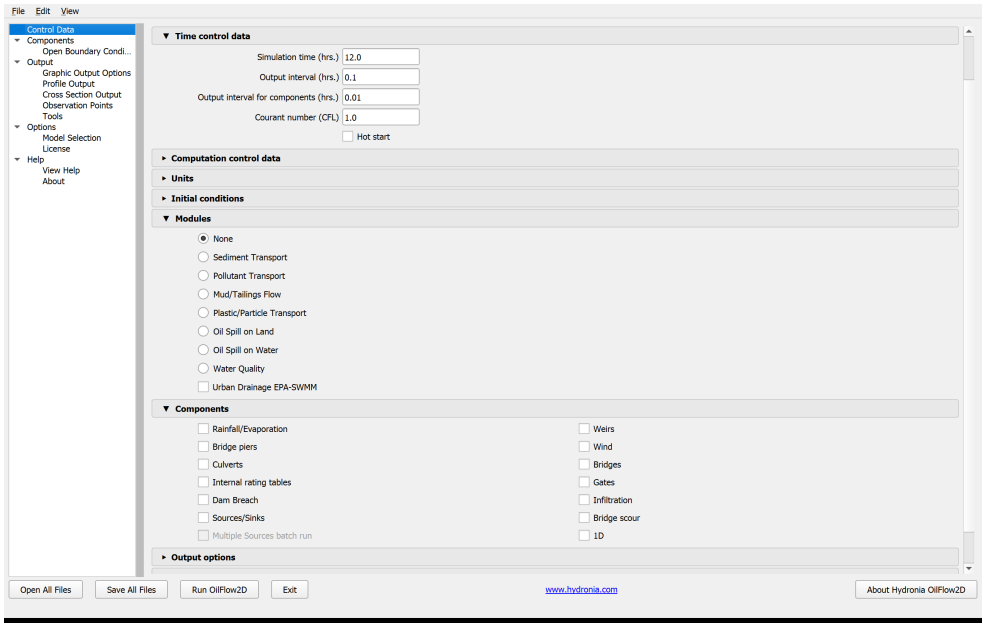


Window dialog of the Export to OilFlow2D tool with the example data.

While exporting the data, you will be able to see the task progress at the top of the message bar. After export completes, version 10 and later users see the Open DIP dialog. QGIS DIP is selected by default; click OK to open it inside QGIS. To use the historical standalone option, expand More options, select Classic DIP, and then click OK.



With QGIS DIP selected, the current interface is shown below:

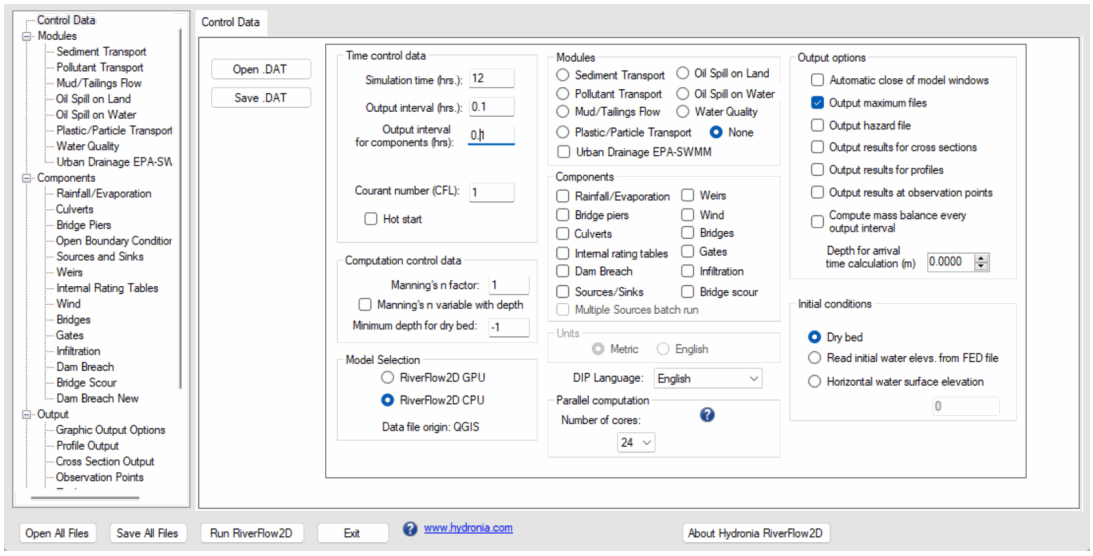


QGIS DIP Control Data panel for Oil Spills in a River Part I.

QGIS DIP and Classic DIP use the same project settings, but their interfaces look different. Older standalone or non-maintenance versions may open Classic DIP automatically. The project-specific Classic DIP settings and Run steps below are retained.

Enter the parameters as shown:

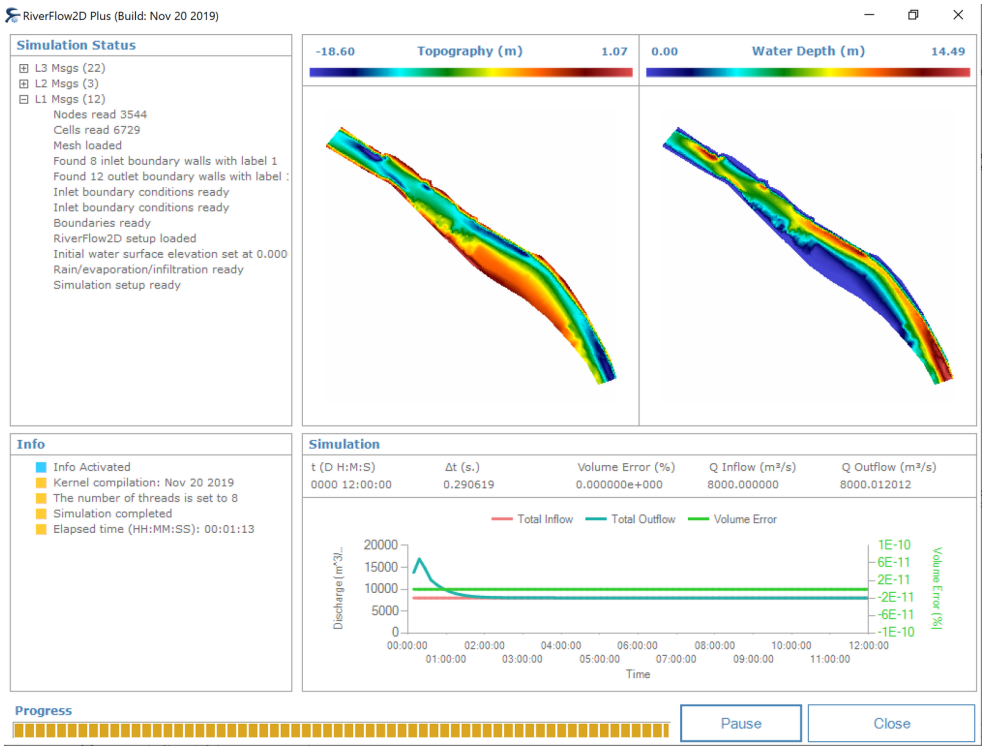
The following Classic DIP configuration is retained as a legacy values reference:



Legacy Classic DIP configuration window.

3. Then click the *Run OilFlow2D* button to run the model. When asked to save your changes, click [Yes]. Overwrite the 'base.dat' file, and click [OK].

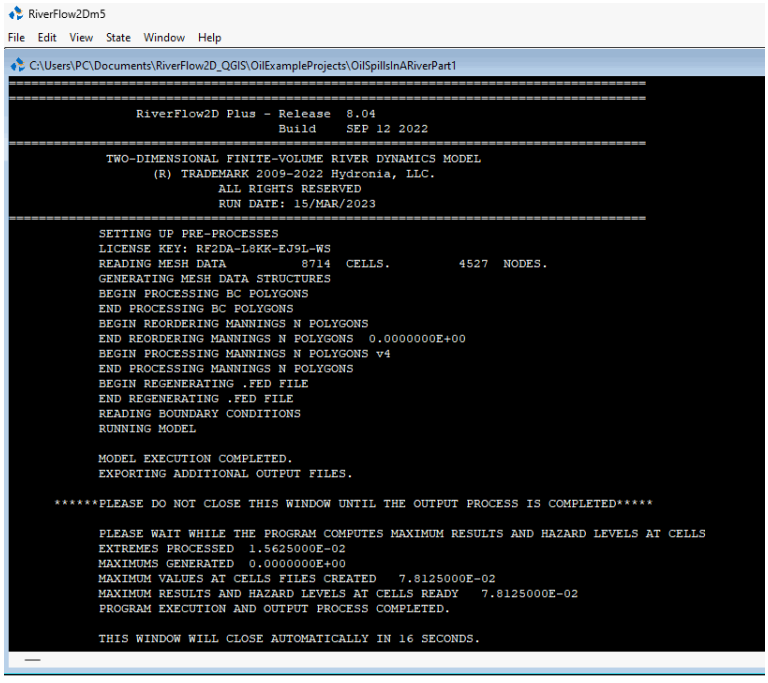
   An image similar to the one shown below should appear:



Window appearing when the model is running.

4. When you have finalized reviewing the information included in this window, you may click [Close].

   The following post processing window will appear indicating the final post-processed task performed by the OilFlow2D model. The window will close automatically.

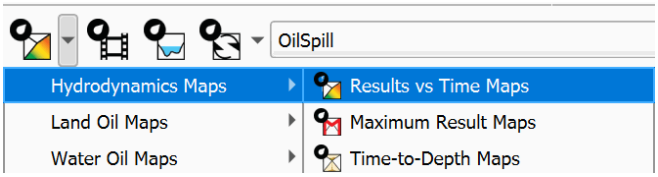


Final post-processes.

7.10 Creating a velocity field map

Once OilFlow2D has finished running, you may want to create maps using the model results. OilFlow2D QGIS plugin offers several tools to create maps and animations. In this section we will show how to create depth maps.

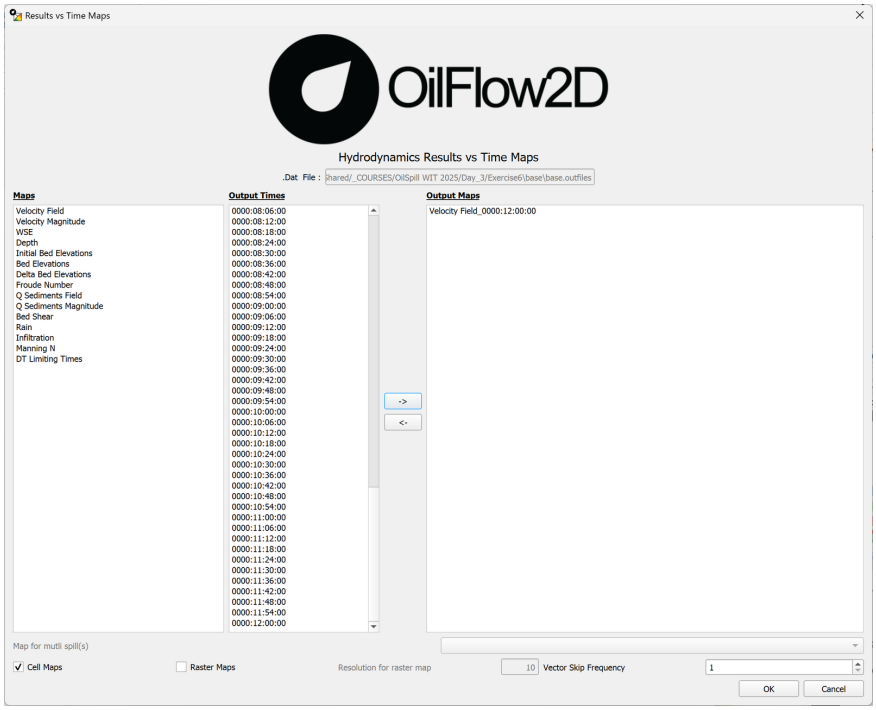
1. Click the *OilFlow2D Maps*  button dropdown menu and select *Results vs Time Maps*.



image

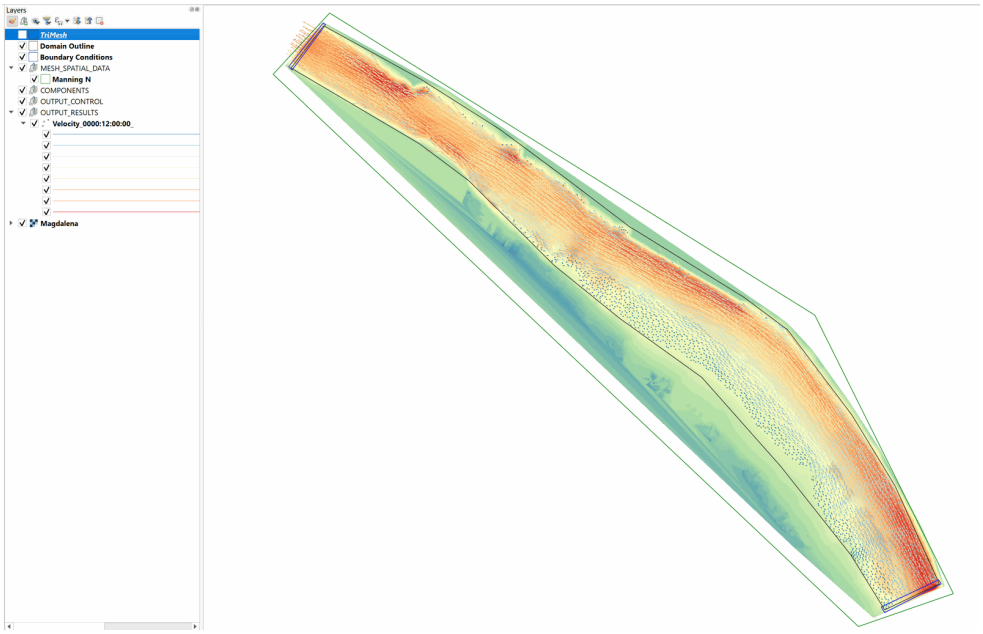
2. In the *Results vs Time Maps* dialog, click the Browse […] button to open the 'base.OUTFILES' file.

3. Under *Maps* select *Velocity Field*, under *Output Times* select 12 hours (0000:12:00:00). Click the arrow right button  to add the 12 hour time to the list of times for *Output Maps*. Click [OK].



image

After a few seconds wait, the velocity field plot will be displayed:



image

This concludes the tutorial *Oil spills in Rivers. Part I: Hydrodynamics*.

8

Oil spills in Rivers. Part II: Oil Spill

This is the second part of the Oil spills in rivers tutorial. The first part explained how to setup and run the hydrodynamic stage, where the model was used to generate the velocity field in the river. In this section we will show how to enter the spill locations, and other oil data to simulate oil trajectories and fate.

8.1 Opening the project with results obtained in Part I

We will use the files resulting from Part I of the tutorial that should have been generated before in the 'OilSpillsInARiverPart1' folder under '\ExampleProjects\OilSpillsInARiverPart1'.

Open QGIS and then open the project resulting from Part I, where we completed the hydrodynamic simulation of the river and created a velocity field map:

Alternately you may choose to open a premade version of the Part I project located in the *OilSpillsInARiverPart1Final* folder.

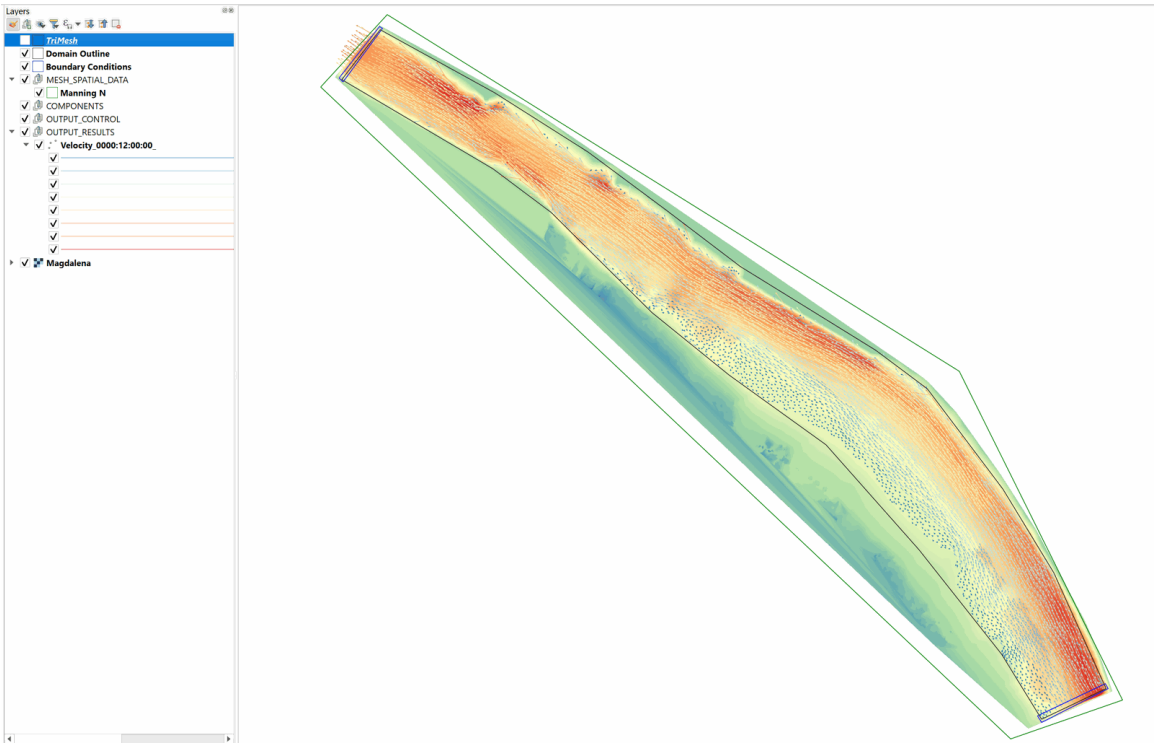
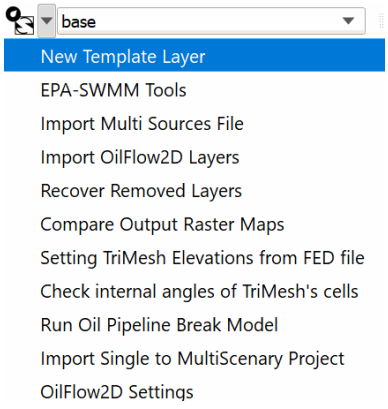


Figure 8.1 – Resulting project from the tutorial Oil Spill in Rivers Part I

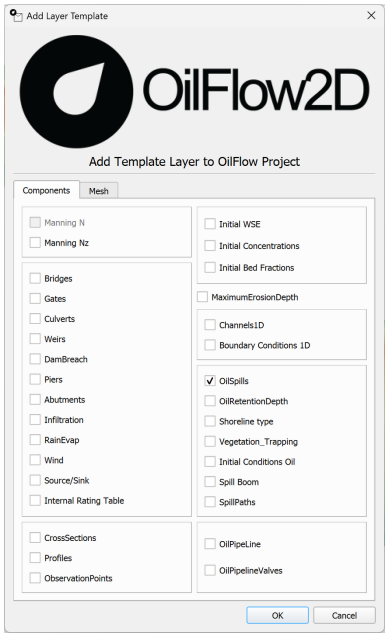
8.2 Create the OilSpills layer and enter the spill location and data

1. Create a new *OilSpills* layer: for this, go to the OilFlow2D toolbar of and click on *New Template Layer*



OilFlow2D dialog to add the new layer.

2. In the *Add Template Layer dialog*, select the *OilSpills* checkBox, as shown in the Figure below and click OK.

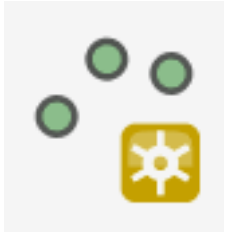


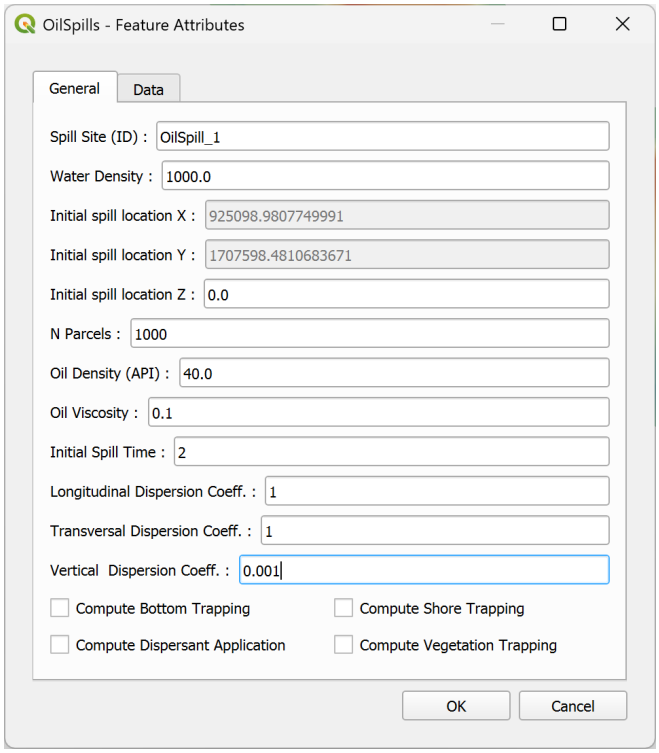
OilFlow2D dialog to add the new layer.

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

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

   .

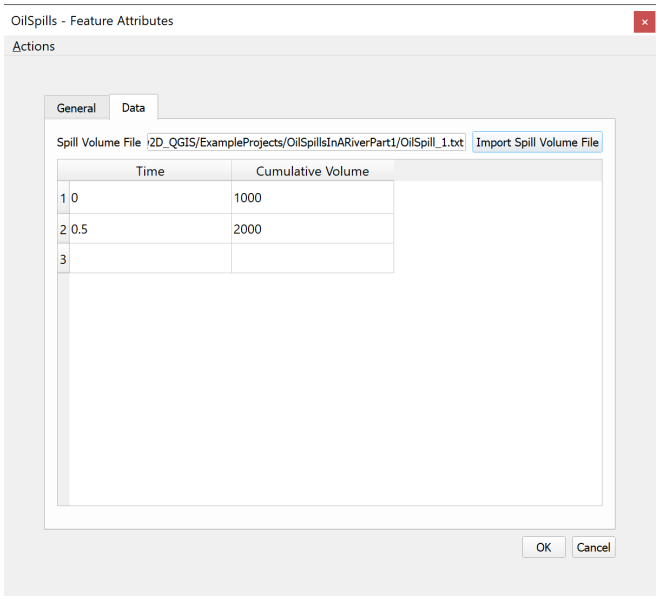
4. To enter the spill location click on the *Add Point Feature* button of the digitalization toolbar  and then click on the location where you want to set the spill point. The oil spill dialog will appear. Enter the data as indicated in the figure:



Oil Spill Dialog.

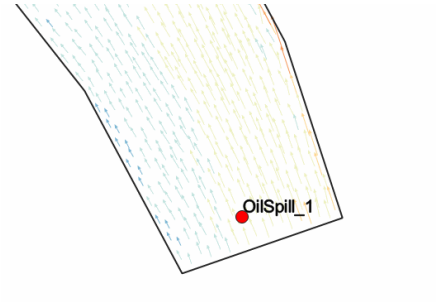
5. Click on the *Data panel* and open the spill volume file using the *Import Spill Volume* File button.

6. Browse to the project 'DATA' folder and select 'Oilspill_1.txt'.



Oil Spill Volume File.

7. Click OK to close the dialog and the oil spill point will be shown:

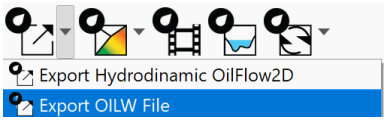


Oil Spill Point.

8. Click on the *Save Layer Edits* button  *Toggle editing* button then click the *Toggle Editing*  to finalize the editing mode.

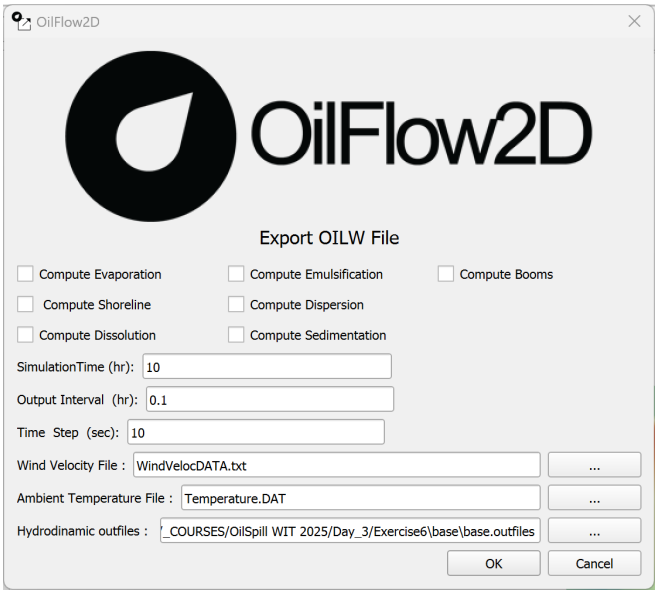
8.3 Export Oil Spill Data

1. Use the Export OILW command to export the data to run the model



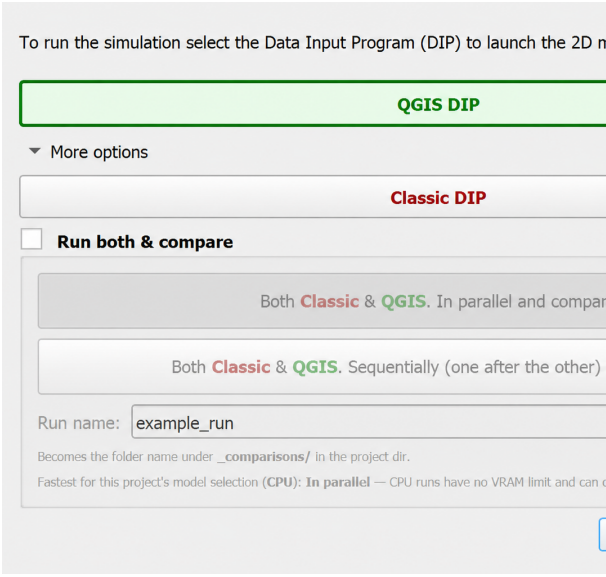
Export OILW command.

2. In the dialog enter the simulation time (12 h), output interval (0.5 h), the time step (10 sec.), import the wind velocity data ('WindVelocData.DAT') file, and ambient temperature file ('Temperature.DAT') as shown, and click OK.

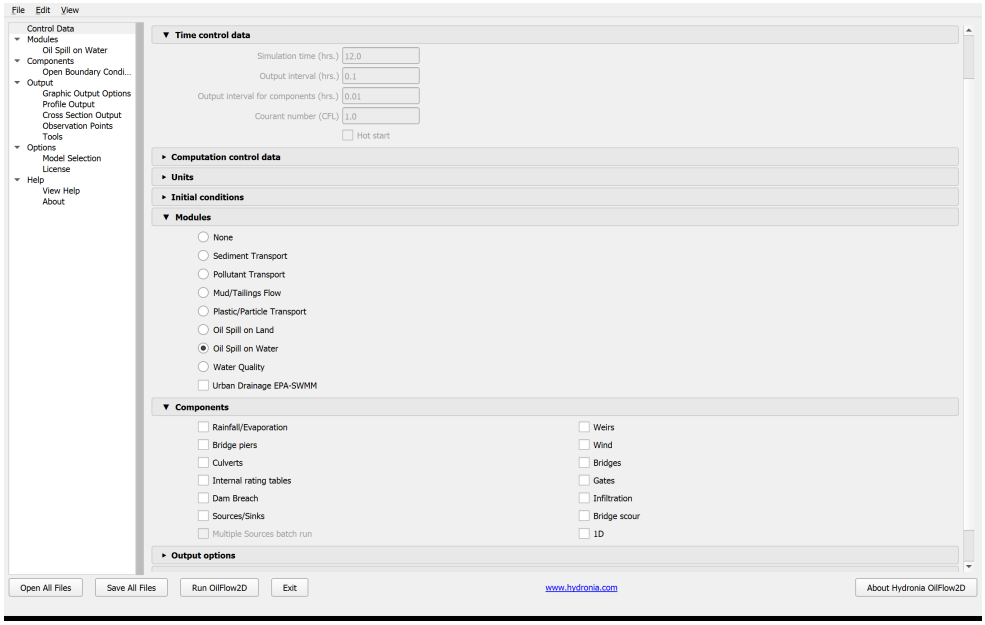


Export OILW Dialog.

After export completes, version 10 and later users see the Open DIP dialog. QGIS DIP is selected by default; click OK to open it inside QGIS. To use the historical standalone option, expand More options, select Classic DIP, and then click OK.



With QGIS DIP selected, the current interface is shown below:



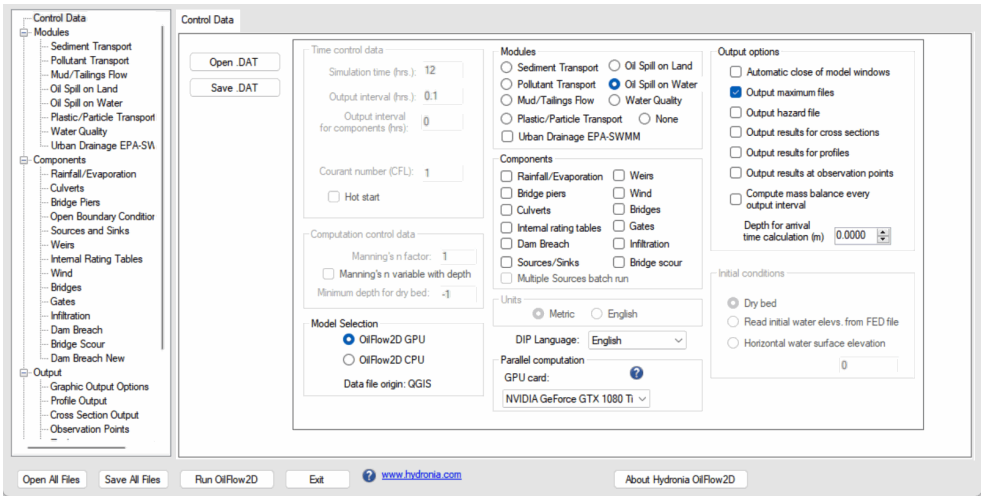
QGIS DIP Control Data panel for Oil Spills in a River Part II.

QGIS DIP and Classic DIP use the same project settings, but their interfaces look different. Older standalone or non-maintenance versions may open Classic DIP automatically. The project-specific Classic DIP settings and Run steps below are retained.

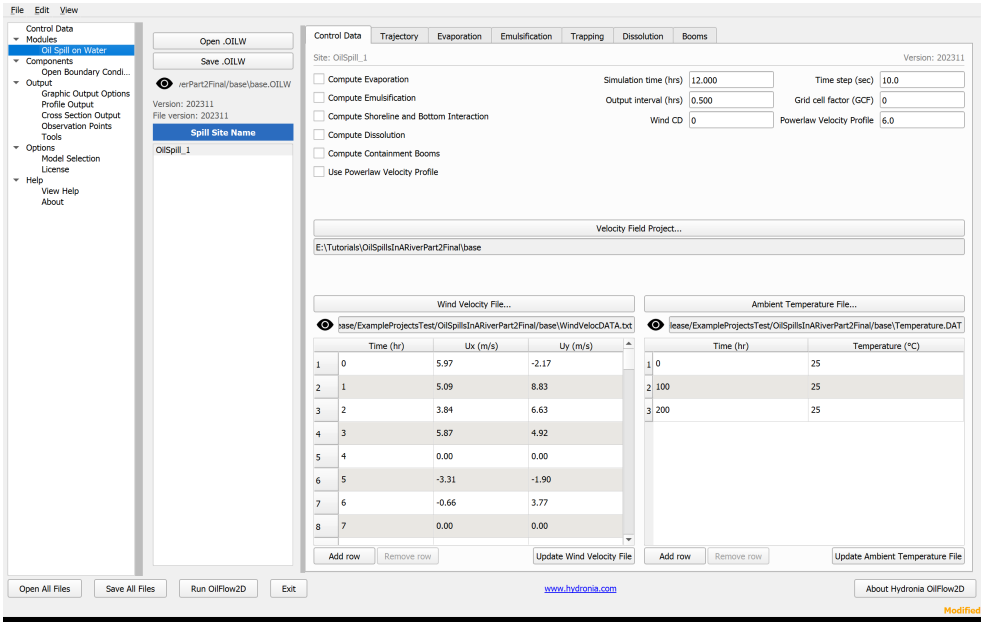
8.4 Enter additional data in the Hydronia Data Input Program and run OilFlow2D

1. Review the data in the Hydronia Data Input Program panels:

The following Classic DIP Control Data panel is retained as a legacy values reference:

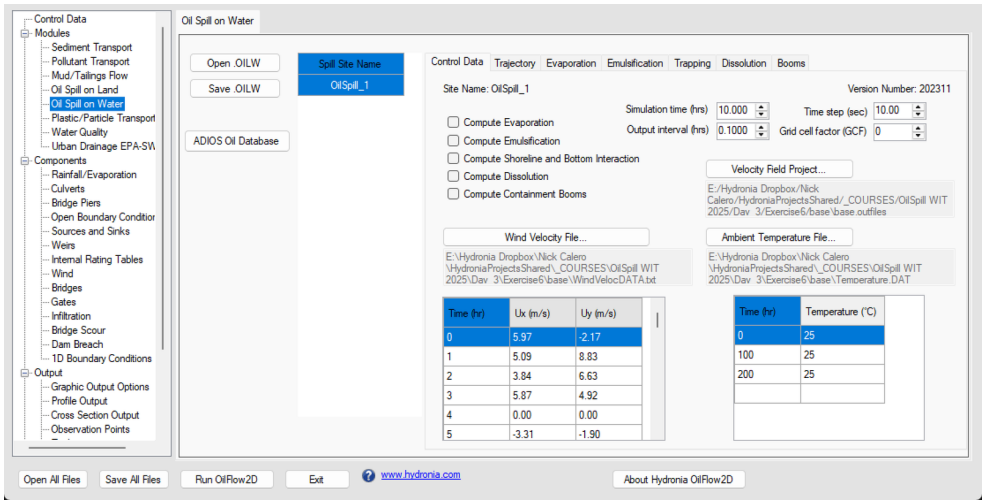


Legacy Classic DIP Control Data panel.

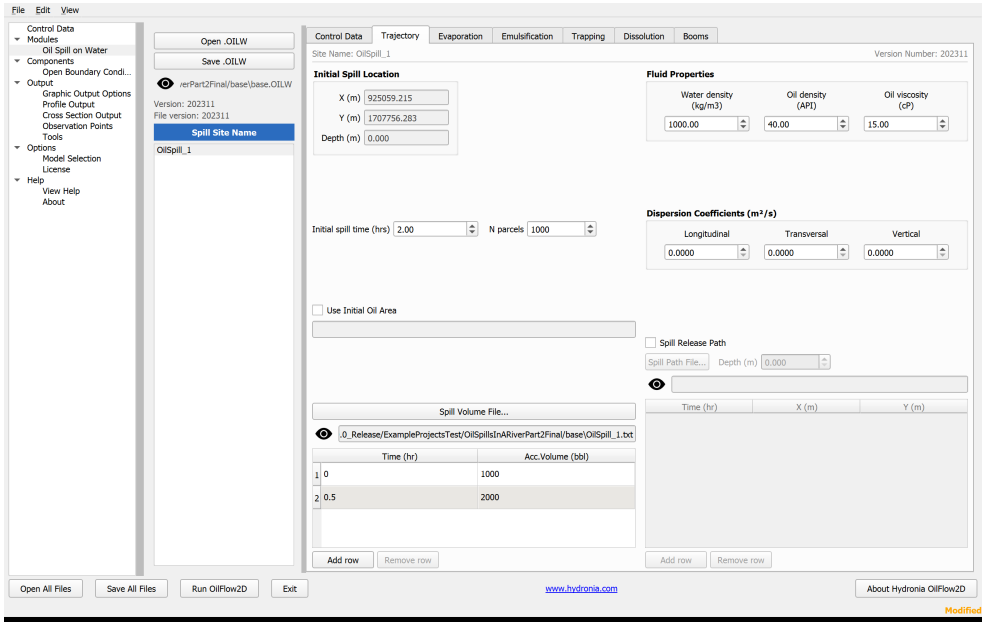


QGIS DIP Oil Spill on Water Control Data panel.

The following Classic DIP Oil Spill on Water Control Data panel is retained as a legacy values reference:

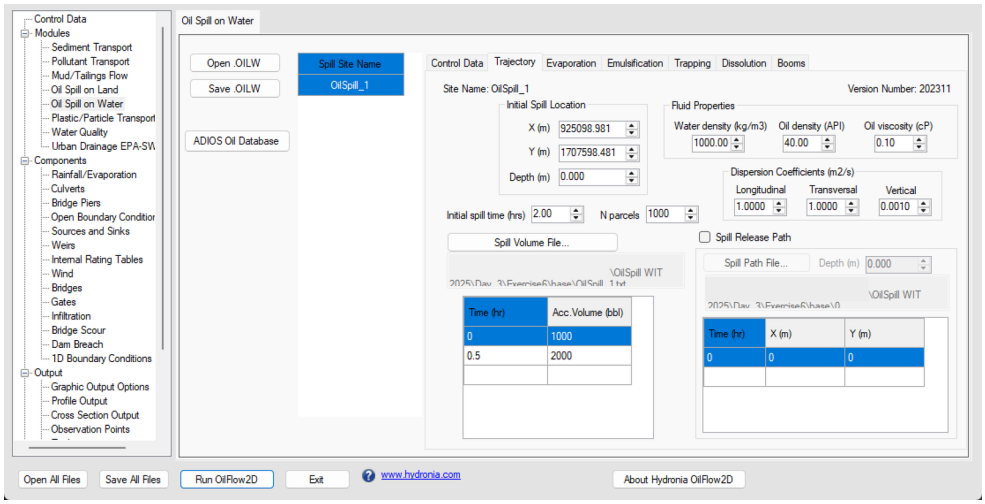


Legacy Classic DIP Oil Spill on Water Control Data panel.



QGIS DIP Oil Spill on Water Trajectory panel.

The following Classic DIP Oil Spill on Water Trajectory panel is retained as a legacy values reference:



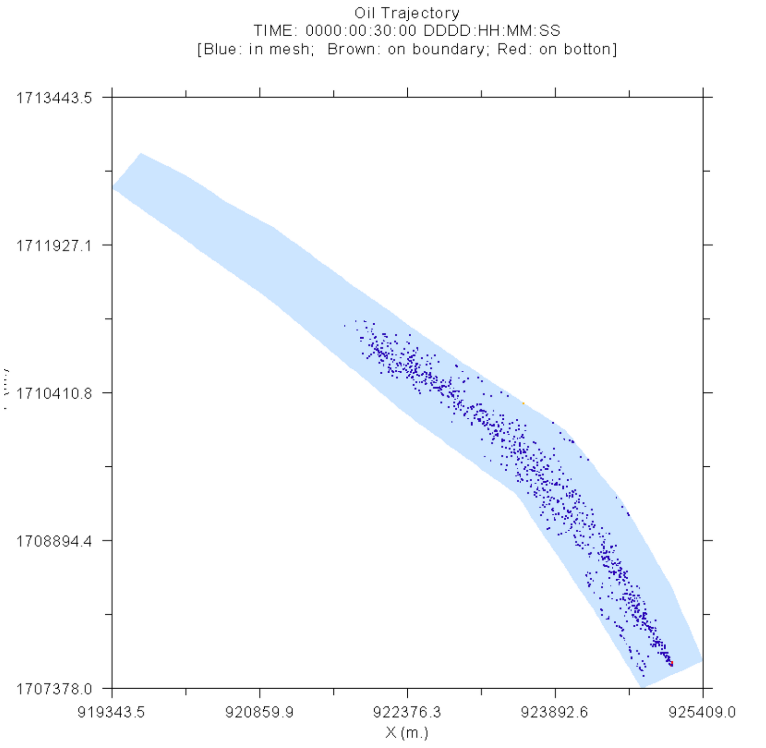
Legacy Classic DIP Oil Spill on Water Trajectory panel.

2. Set the Logitudinal, Transversal and Vertical dispersion values to 0.

3. Click [Save .OILW] and overwrite the ‘base.OILW’ file.

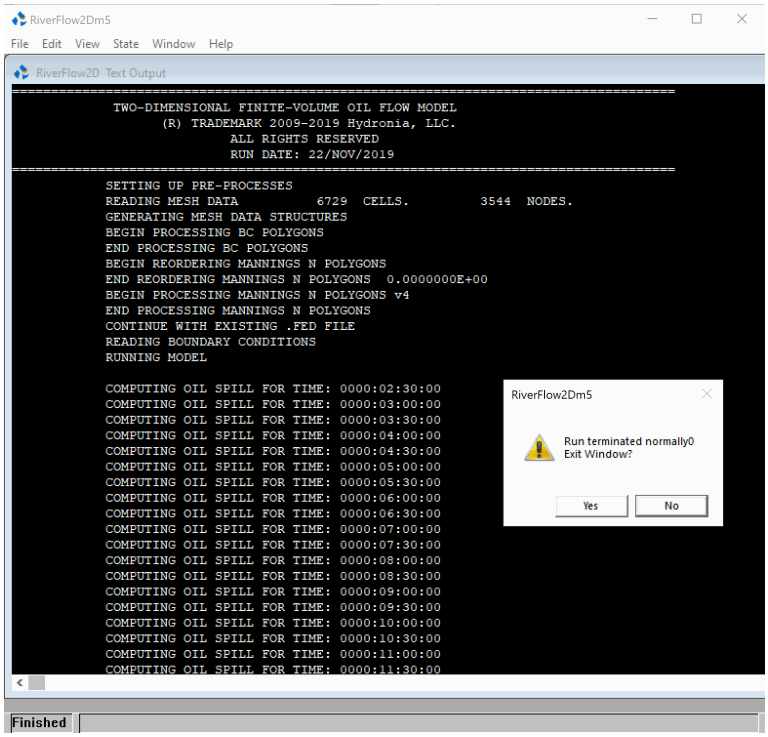
4. When ready to run the model click *Run OilFlow2D*.

5. A plot window will open while the model runs:



Model plot window.

The model run window will report the times being computed and a message will indicate that the model finished running:



Model execution window.

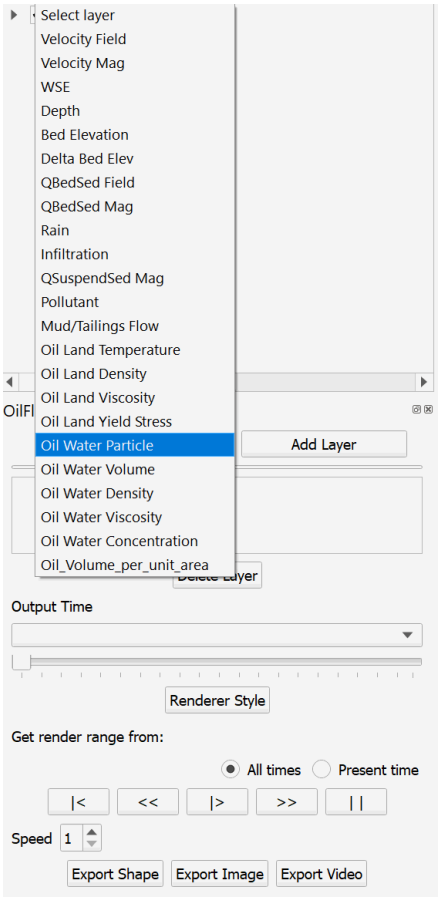
6. Click *Yes* to close the window.

8.5 Creating a Oil Spill Trajectory Animation

To check the model results we can create a oil trajectory animation using the following procedure:

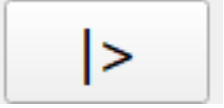
1. Select the Animation tool  :

2. On the Animation panel, click *Add Layer*, and select the 'Oil Water Particles' from the dropdown menu as shown:

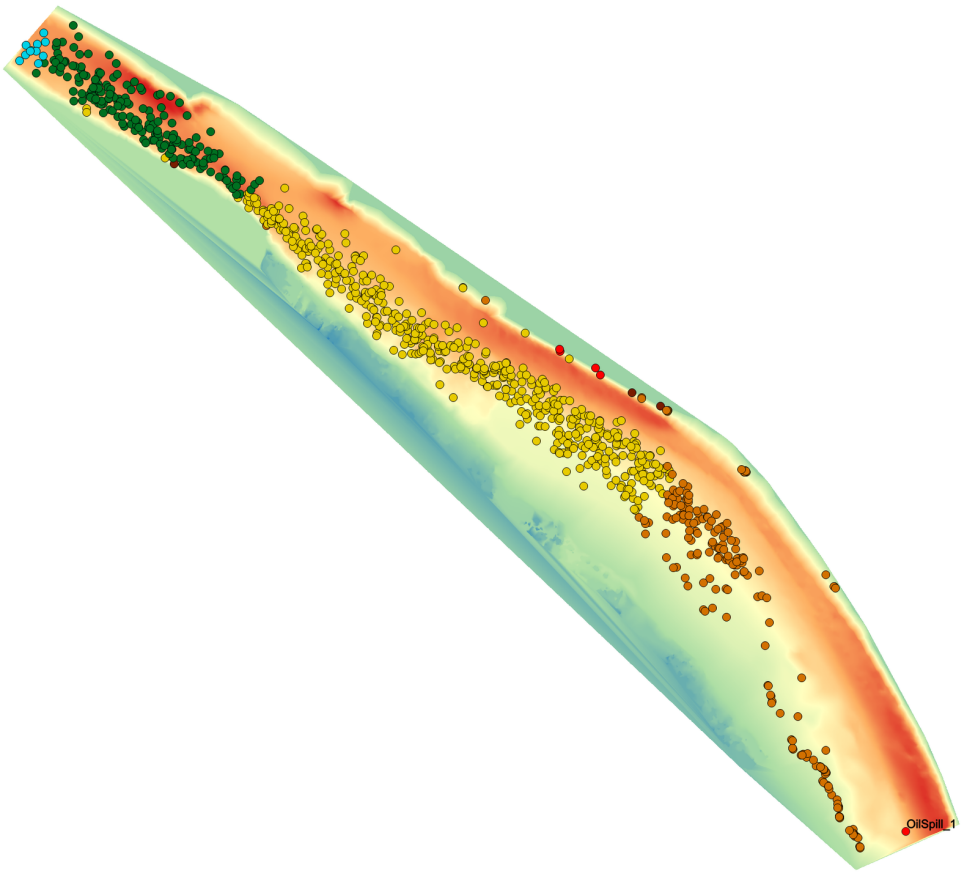


Select Oil Water Particles animation.

3. Click the *Add Layer* button, and select *Oilspill_1* and click [OK].

4. Then on the Animation panel, select the Oil_Water_Particle_Oilspill_01 layer and click 

to view the animation:



Oil trajectory during animation.

This concludes the tutorial *Oil spills in Rivers. Part II: Oil Spill*.

9

Oil spills in Coastal Areas. Part I: Coastal Hydrodynamics

This tutorial guides in the application of OilFlow2D to simulate oil spills in a coastal area. The first part explains how to setup and run the hydrodynamic stage, where the model is used to generate the velocity field in the area. The second part, included in a separate chapter, explains how to enter the spill locations, and other oil data to simulate oil trajectories and fate.

9.1 Tutorial data

Before starting the project with QGIS. It is recommended to create a folder that will contain all the necessary data such as the boundary condition files, the elevation data files, either in ASCII or in raster format, etc.
The files required to follow this tutorial can be extracted from the 'ExampleProjects' zip file under the 'OilSpillsInABayPart1' folder. This zip file is downloaded separately from your installation materials. The folder contains the files shown:

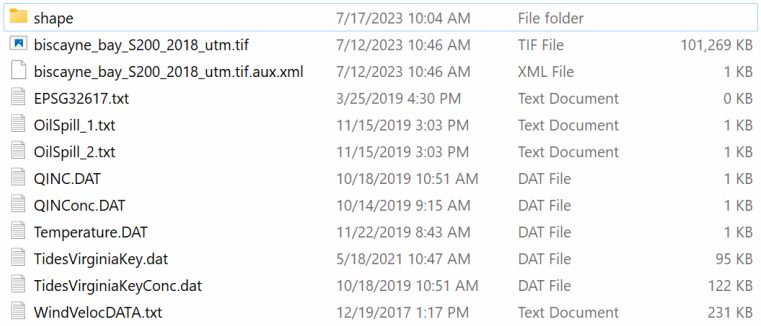


Figure 9.1 – Files with data required in this tutorial.

9.2 Starting QGIS

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

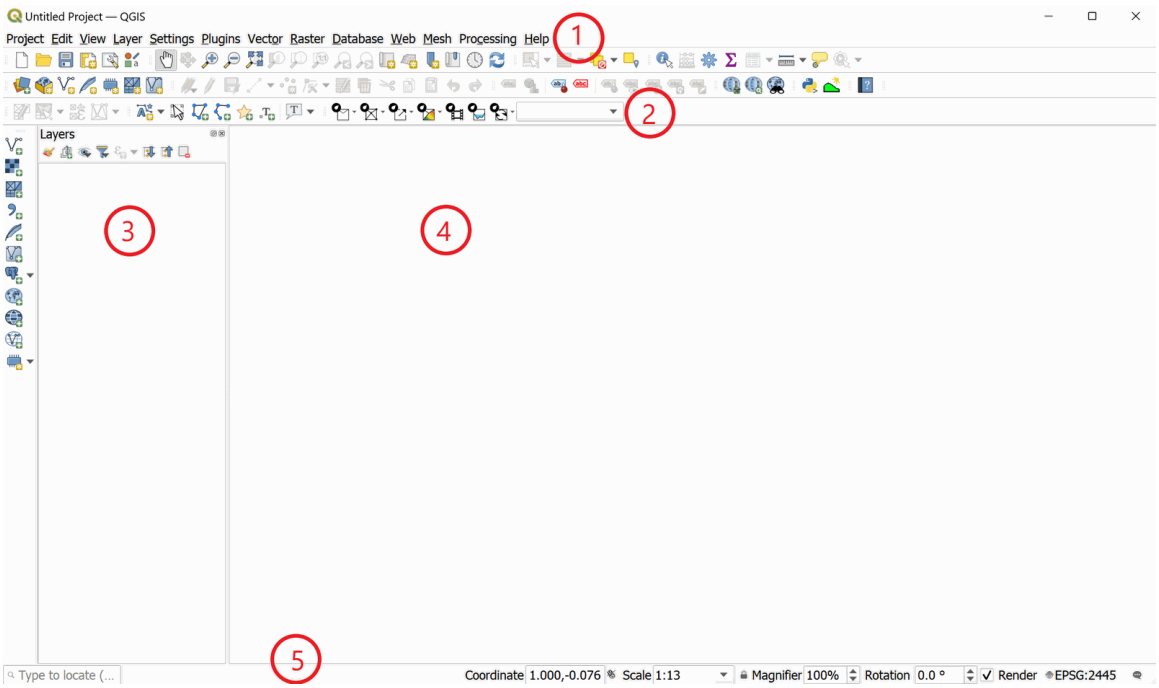


Figure 9.2 – QGIS interface for OilFlow2D indicating window areas. 1: Menu bar, 2: Tool bar, 3: Layer panel, 4: Maps view area, 5: Status bar.

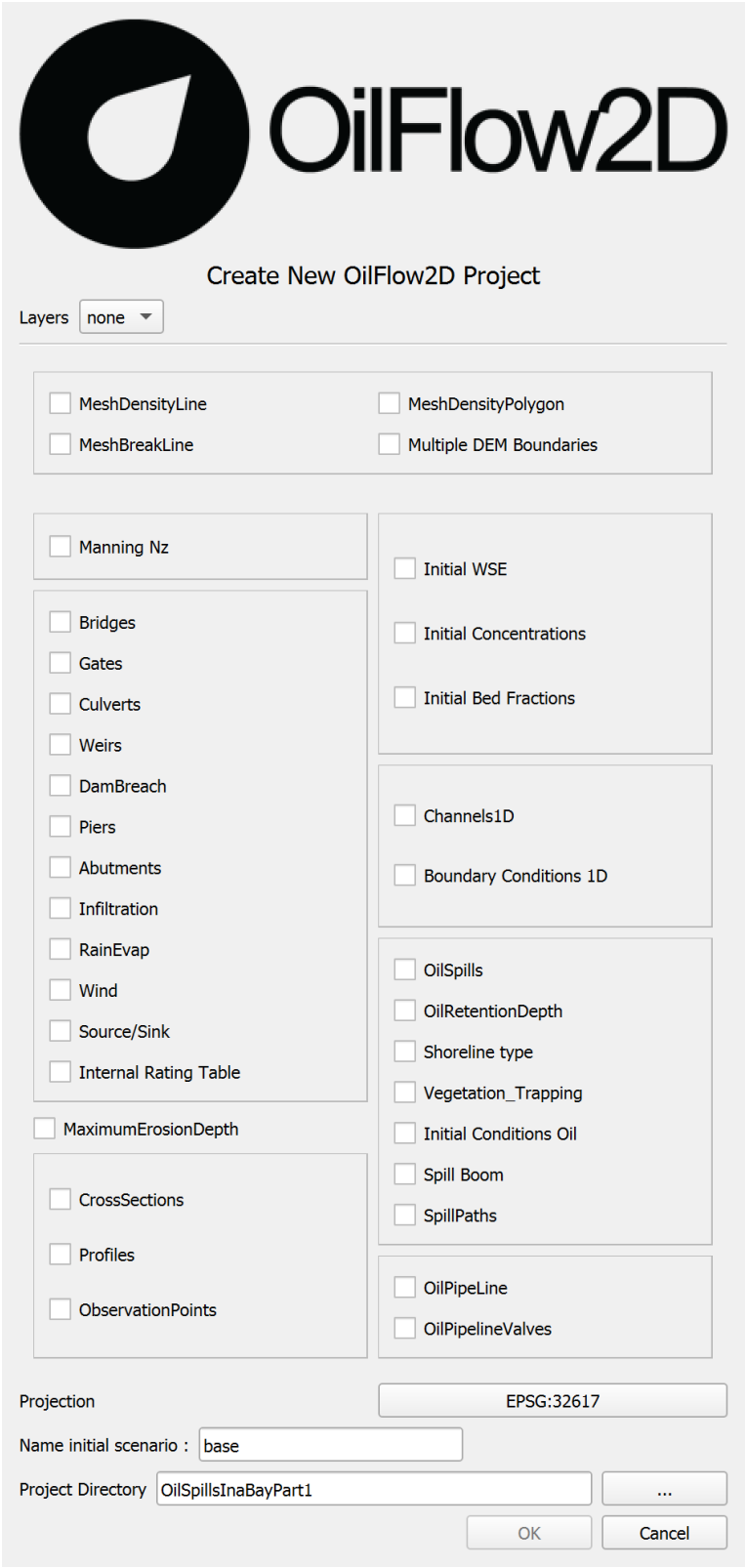
Note the tool bar contains icons that give access to all the OilFlow2D components. Some of the icons have a drop down list that allow getting to more tools.

9.3 Start a new OilFlow2D project



1. To create a new OilFlow2D project, click on the *New OilFlow2D Project* button in the toolbar. A dialog window appears where you select the optional component layers, the Coordinate Reference System (CRS), the initial scenario name, and the project directory.
2. Select *None* in the Layers drop down menu.
3. Select the *Projection* button and select: Coordinate Reference System (CRS) EPSG 32617 corresponding to *WGS 84 / UTM zone 17N - Projected (meters)*, and click OK.
4. Keep *Name initial scenario* set to base. Click the "…" button and choose the OilSpillsInaBayPart1 folder for the *Project Directory*. This is the folder where the model will write all results and output files.

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



Create New OilFlow2D Project configured for OilSpillsInaBayPart1.

OilFlow2D will use the unit system as that defined in the projection you selected. If the pro-

jection has coordinates in feet, units will be set to English. If the projection coordinates are in meters, units will be set to Metric or SI.

6. On the QGIS *Project* menu, click *Save*, to save the project assigning the name OilwPart1, in the same directory that you previously selected in the *Create New Project* dialog above.

9.4 Load elevation data

OilFlow2D uses elevation data in raster format. For instance, to load an ASCII grid file, from the *Layer* menu, click *Add Layer*, and then click *Add Raster Layer…*. You may also click the *Add Raster Layer* button  .

The following dialog will appear where you can select the raster file containing elevations. In this case, select biscayne_bay_S200_2018_utm.tif from ExampleProjects\OilSpillsInABayPart1\DATA.

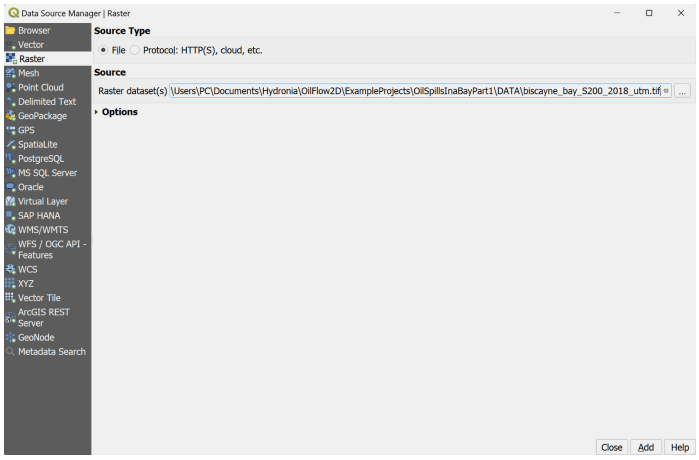


Figure 9.3 – Raster data import dialog.

Click Add, Close the dialog and click on the Zoom Full button  .

The raster will be displayed on the screen, by default it is rendered in gray gradient. Right-clicking on the label of the created layer and selecting *Properties* allows changing the rendering style for a more informative color palette. When changed to Singleband psudocolor it appears as shown:

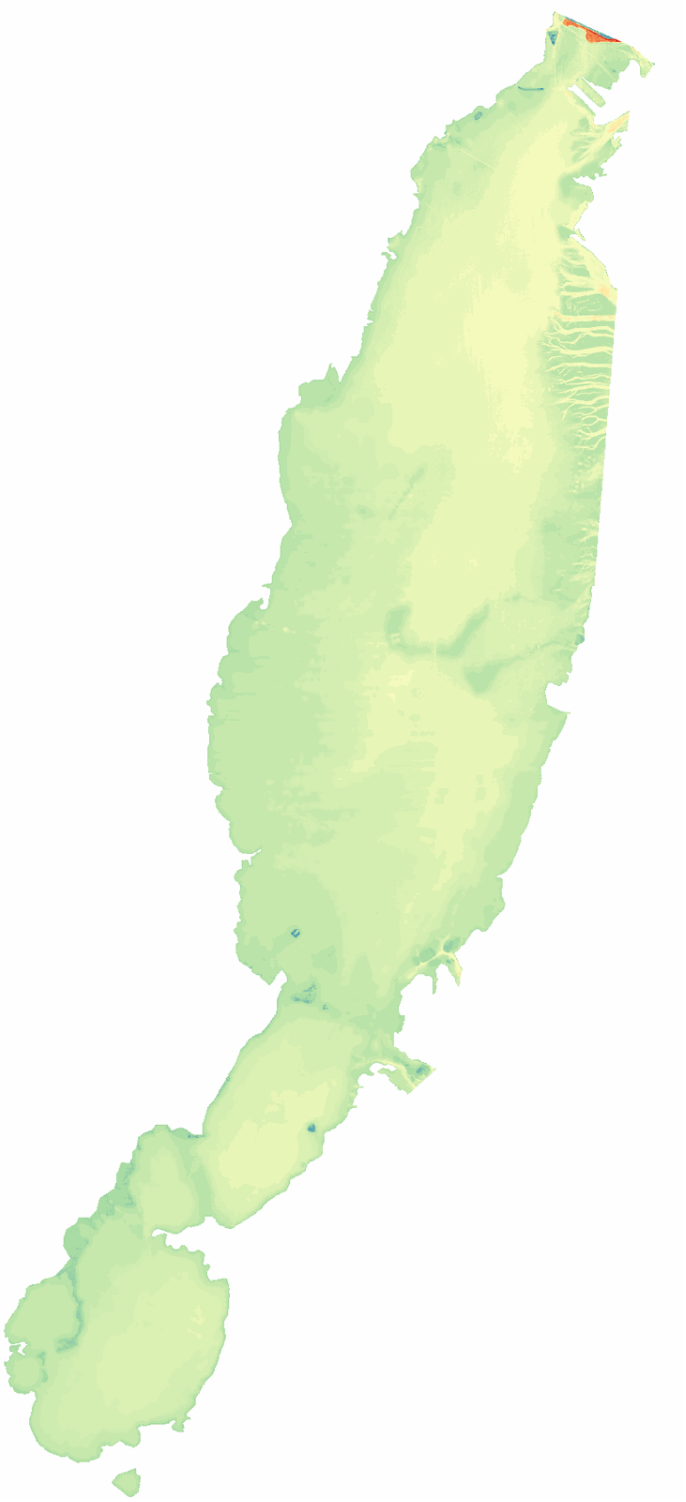


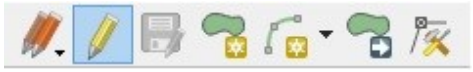
Figure 9.4 – Digital elevation model in raster format created by interpolation.

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

9.5 Create the limits of the modeling area (Domain Outline)

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

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

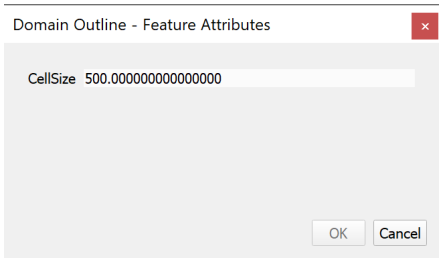
   Menu buttons for digitalization toolbar.

2. Clicking on the button activates the rest of the editing buttons. Now click the *Add Polygon Feature* tool which is the bean looking polygon  .

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

   Make sure that the polygon is contained within the limits of the raster layer since OilFlow2D will not extrapolate elevations to areas that are outside of the available data on the raster layer.

3. To finalize and close the polygon, right-click anywhere on the screen. A dialog window to input the cell size attribute of the newly created polygon. The value for the reference size of the mesh cell is indicated. Enter a value of 500 m.

   Input of attributes for the Domain Outline layer.

   If you want to make any correction to the polygon, use the *Node* tool  to move, add or remove nodes as necessary.

4. Save the polygon by clicking the *Save* button  .

   and click on *Toggle Editing* icon  to deactivate the layer edit mode and complete the *Domain Outline*.

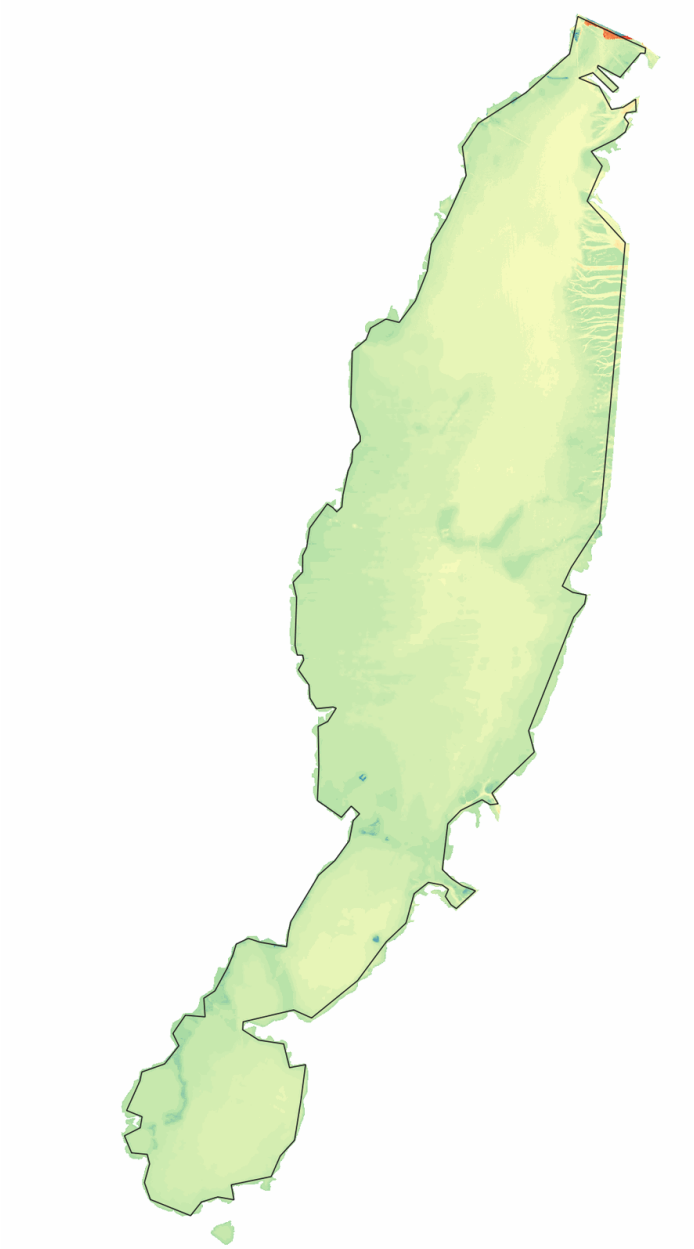


Figure 9.5 – Domain Outline layer.

9.6 Generating the triangular-cell mesh

Now that the *Domain Outline* layer has been created, proceed to create the mesh.

To generate the mesh, click on the icon  .

The following figure shows the generated mesh. You will also find the Layers panel one new layers: *Trimesh* :

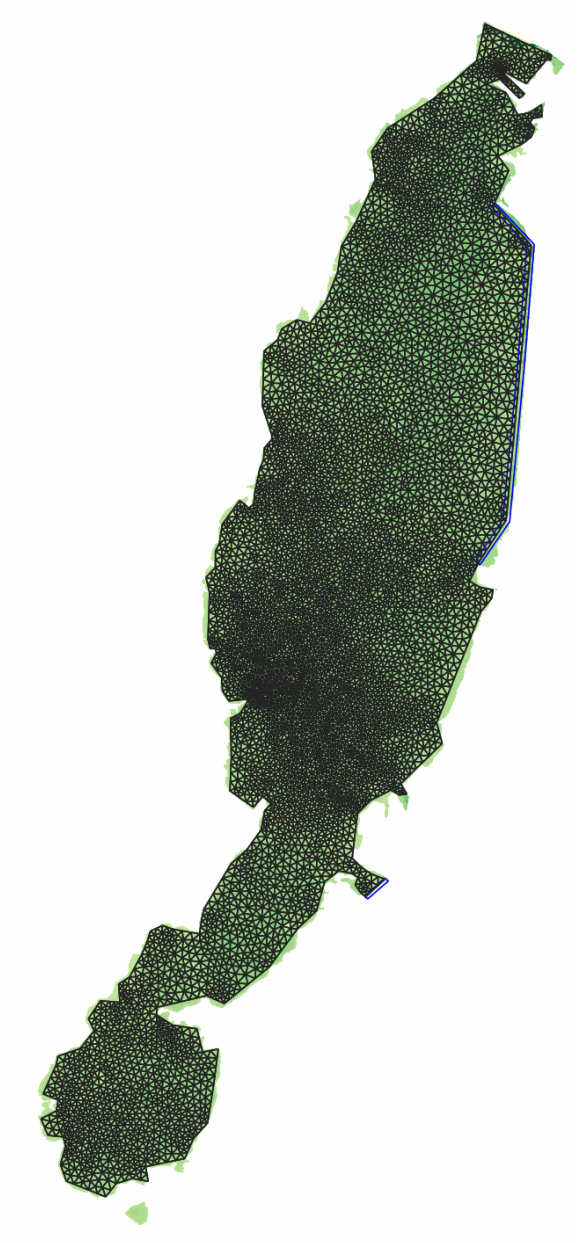


Figure 9.6 – Mesh generated.

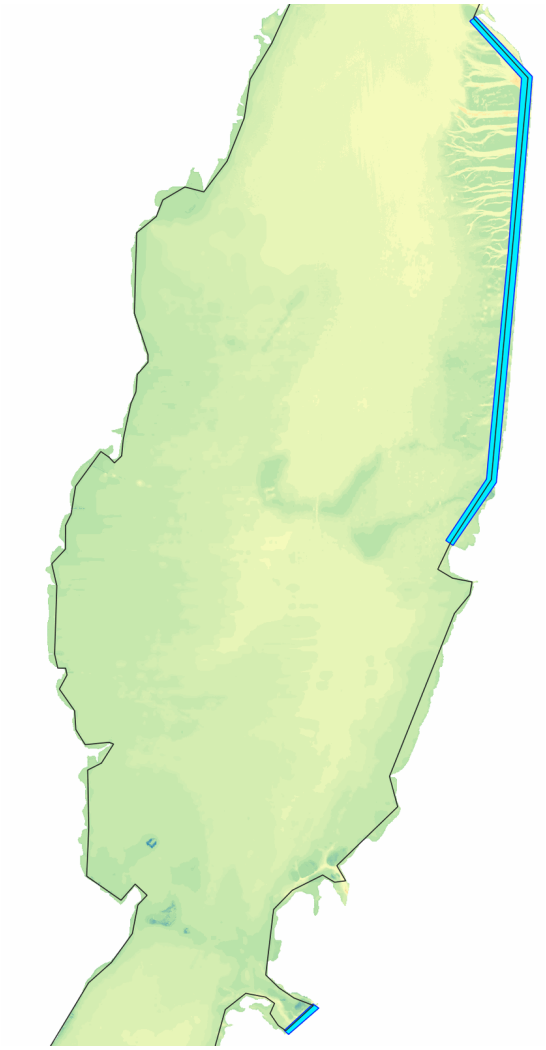
9.7 Setting the boundary conditions

In this tutorial we will define tidal stage condition at open boundaries. To define the tide boundary condition do as follows:

1. Click the Toggle Editing button  to add the polygons that are going to indicate

the nodes on which the inflow condition is imposed.

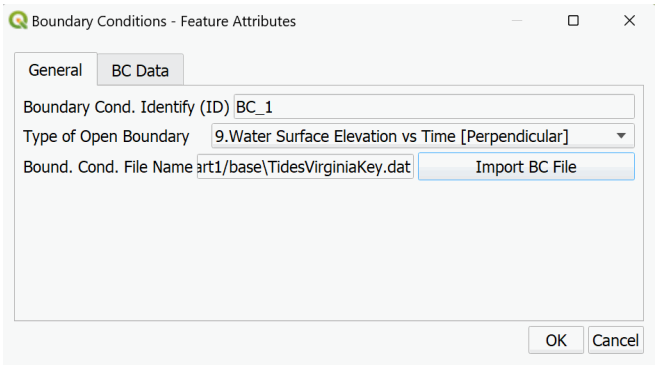
2. Draw a polygon that includes the nodes or vertices at the lower end of the mesh as indicated in the figure:



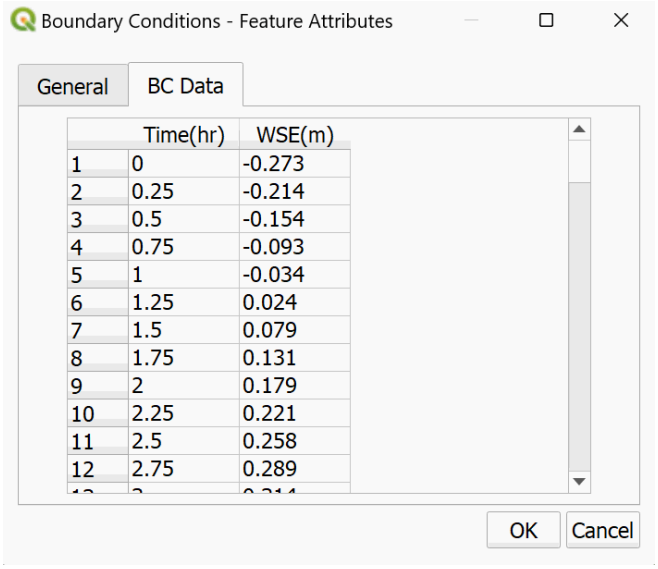
Polygon that covers the nodes defining the inflow boundary condition segments.

3. To finish the polygon, right-click on the last vertex. A window to enter the attributes of the newly created polygon is displayed. It is important that the boundary condition polygon would cover completely the inflow/outflow segment.

4. In the window the (*Boundary Cond. ID*) is *BC_1*, and the boundary condition Type: *9. Water Surface Elevation vs. Time[Perpendicular]*.



Inflow boundary condition data.



Tidal condition read from the TidesVirginiaKey.DAT file.

5. Repeat the process for other tidal boundaries as necessary.

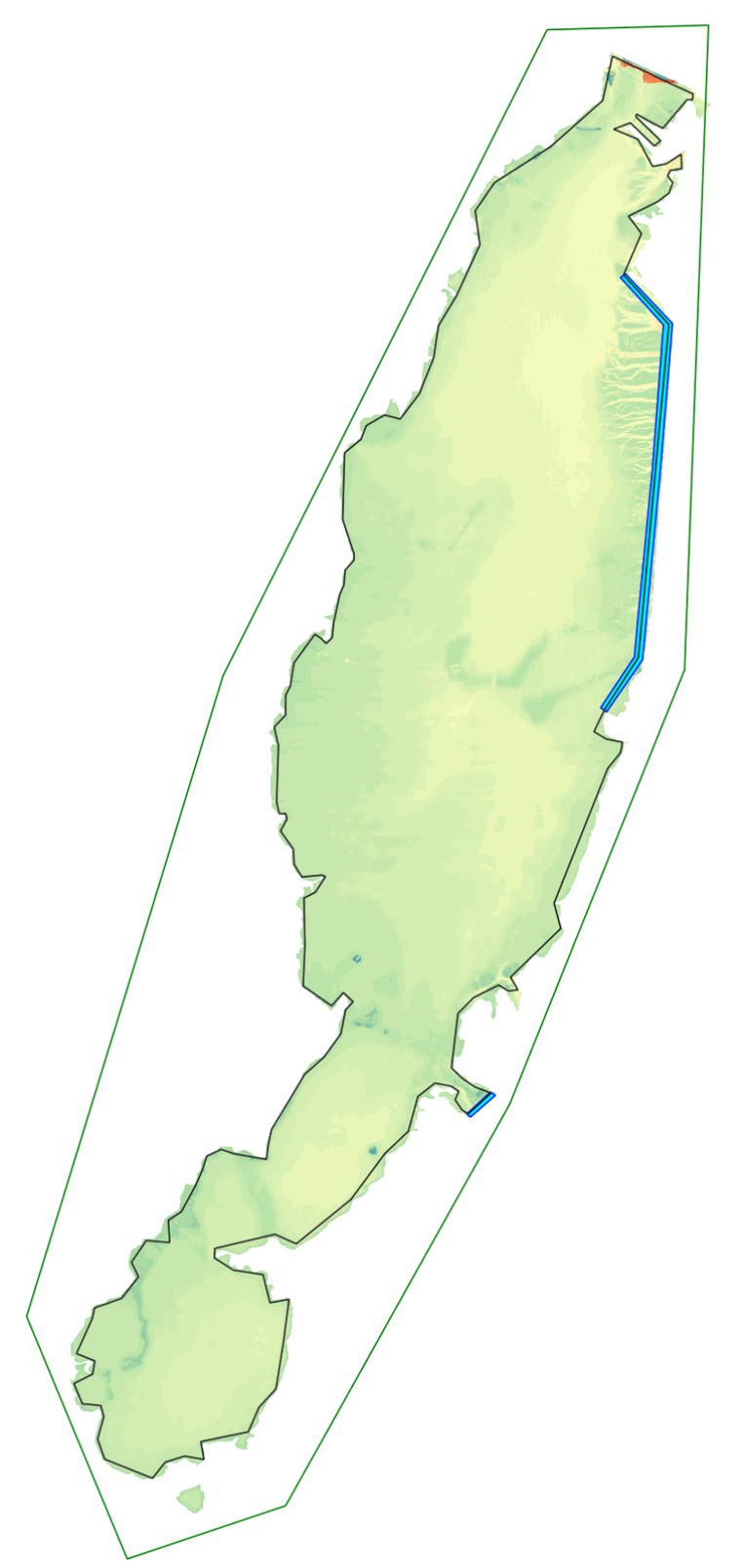
There are no outflow conditions in this case, since the open boundaries will consider inflow and outflow though the tidal cycle.

9.8 Assigning Manning's n roughness coefficients

Manning's coefficients are used in OilFlow2D to represent the terrain roughness. To assign Manning's n values, we will enter polygons with particular n's. There can be as many polygons as required to reproduce the spatial variability of this parameter. In this example, a single polygon will be used for the entire area.

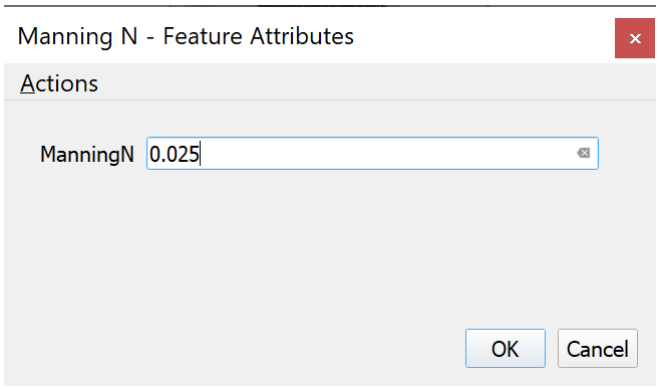
1. Select the Manning N layer and click the *Toggle Editing* button  .

2. Draw the polygon around the entire domain taking care that it covers all the cells. You should have an image like the one shown below:



Editing the Manning N layer.

3. Close the last vertices on the polygon by right-clicking on the desired position. The following dialog window is presented where you must input the Manning's n value associated to the polygon (0.025):



Dialog to input Manning N Feature Attributes.

4. Click the Save icon  and then click the Editing Tool icon  to deactivate editing mode.

9.9 Exporting the files to OilFlow2D and running the model

Once the layers with the input information to the model have been created, the next step is to run the tool that exports the data files in the proper format required by OilFlow2D. The process is as follows:

1. Unselect the *MeshDensityLine* and *MeshBreakline* layers, since they are not used in this example.

2. Run the *Export Hydrodynamic OilFlow2D* plugin:



image

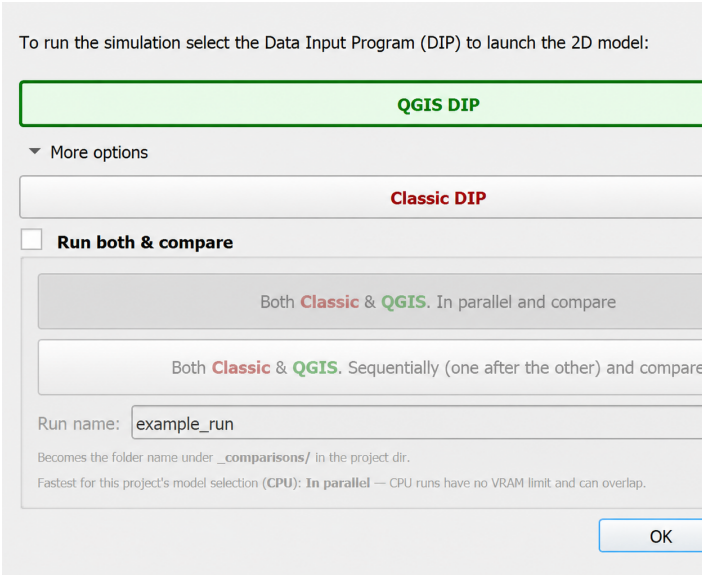
3. A dialog window is presented, where we will indicate the raster layer Magdalena containing the Digital Elevation Model (DEM).

4. Click OK to start the export process.

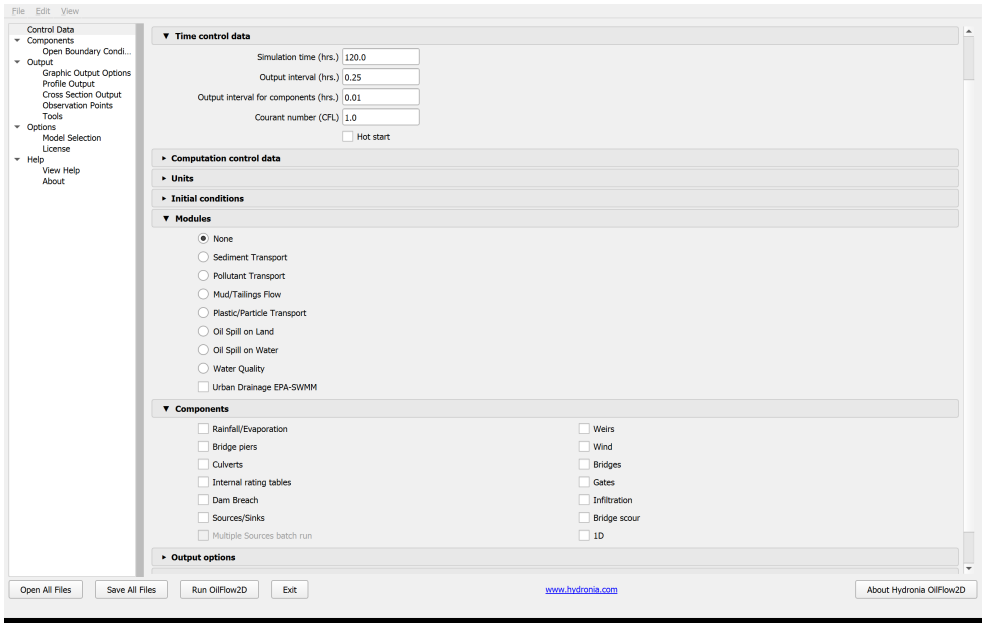


Window dialog of the Export to OilFlow2D tool with the example data.

While exporting the data, you will be able to see the task progress at the top of the message bar. After export completes, version 10 and later users see the Open DIP dialog. QGIS DIP is selected by default; click OK to open it inside QGIS. To use the historical standalone option, expand More options, select Classic DIP, and then click OK.



With QGIS DIP selected, the current interface is shown below:

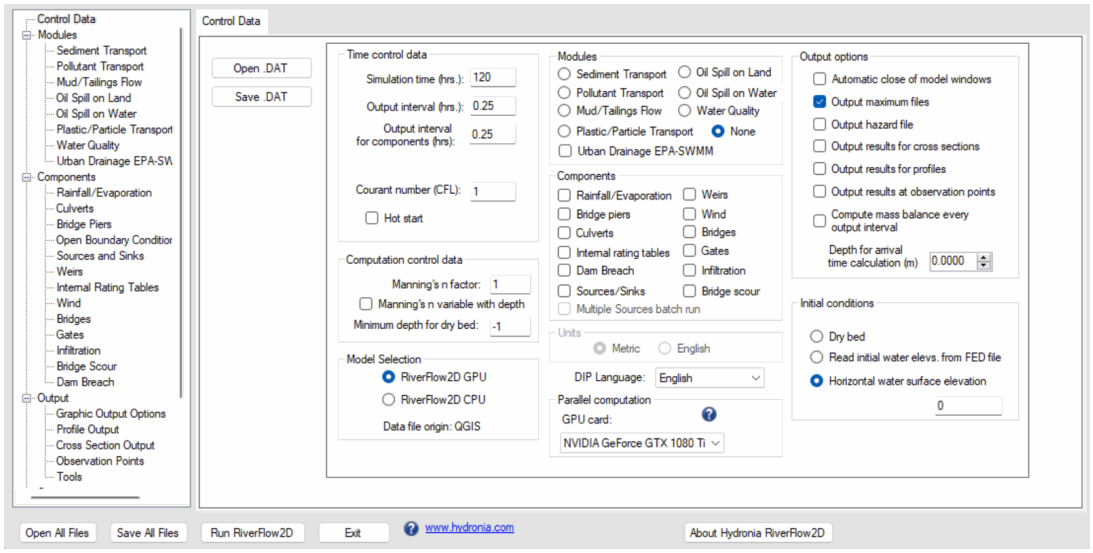


QGIS DIP Control Data panel for Oil Spills in a Bay Part I.

QGIS DIP and Classic DIP use the same project settings, but their interfaces look different. Older standalone or non-maintenance versions may open Classic DIP automatically. The project-specific Classic DIP settings and Run steps below are retained; in this case, base.DAT should already be set.

5. Enter the parameters as shown:

   The following Classic DIP configuration is retained as a legacy values reference:

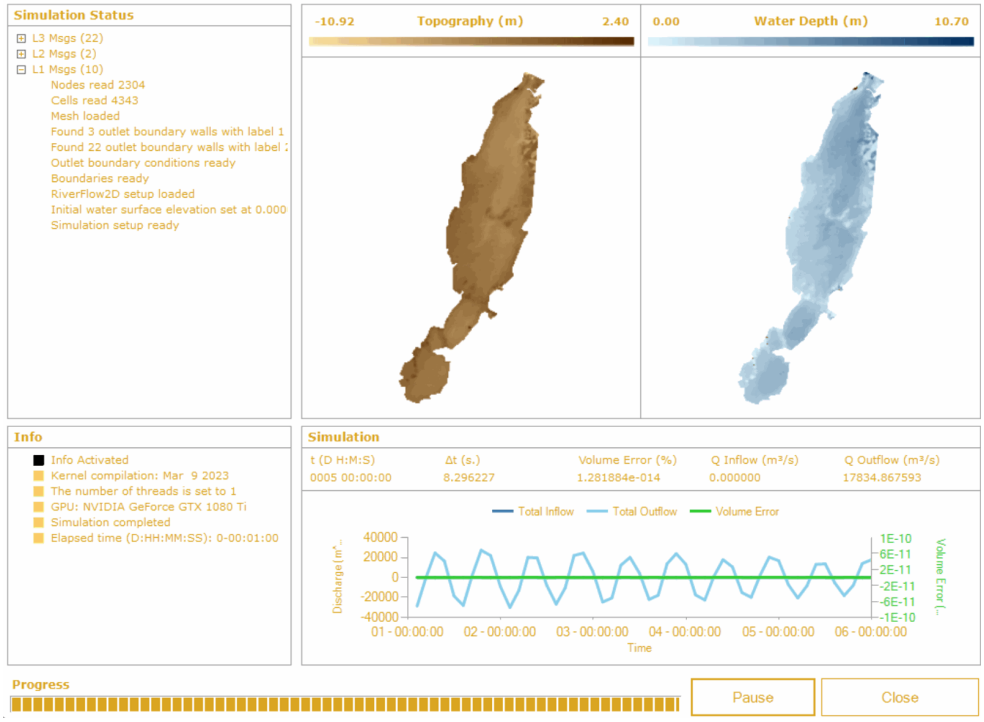


Legacy Classic DIP configuration window.

6. Then click the *Run OilFlow2D* button to run the model.

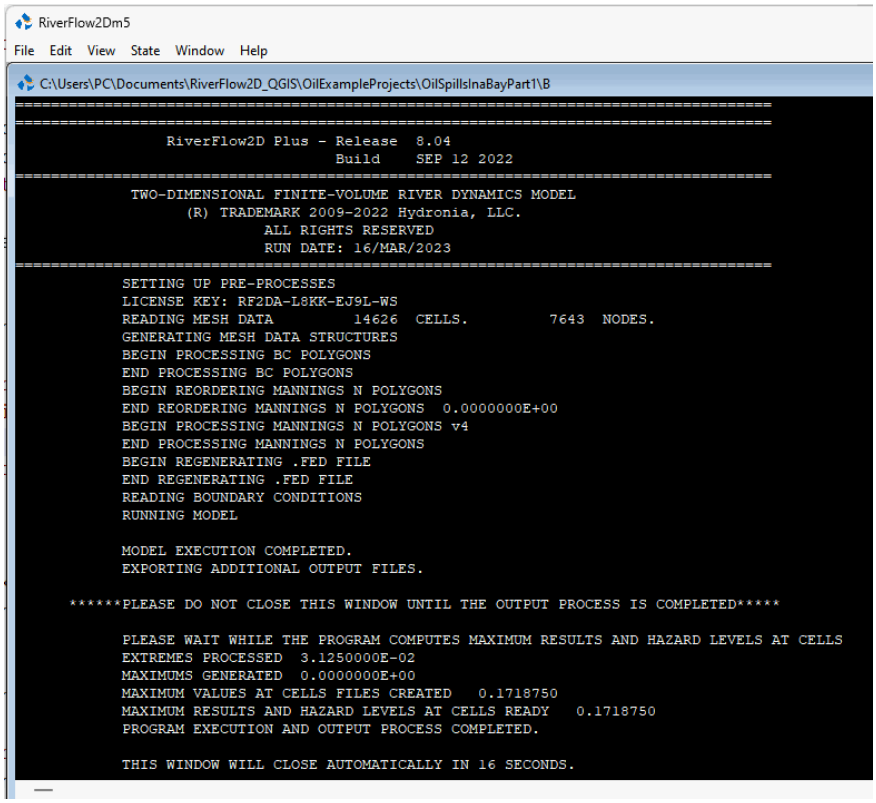
7. A dialog box will appear asking to save your changes, click [Yes]. By default it will use the name of the *Scenario* as the name of the .dat file. Confirm to overwrite the file.

8. A few console windows will appear to initiate the model, then an image similar to the one shown below should appear.



Window appearing when the model is running.

9. When you have finalized reviewing the information included in this window, close it. The following post processing window will appear indicating the final post-processed task performed by the OilFlow2D model. The window will close automatically.



Final post-processes.

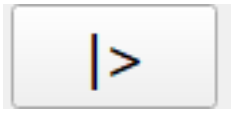
9.10 Creating an Animation of the Velocity Field in the Bay

To check the model results we can create an animation of the velocity field in the bay using the following procedure:

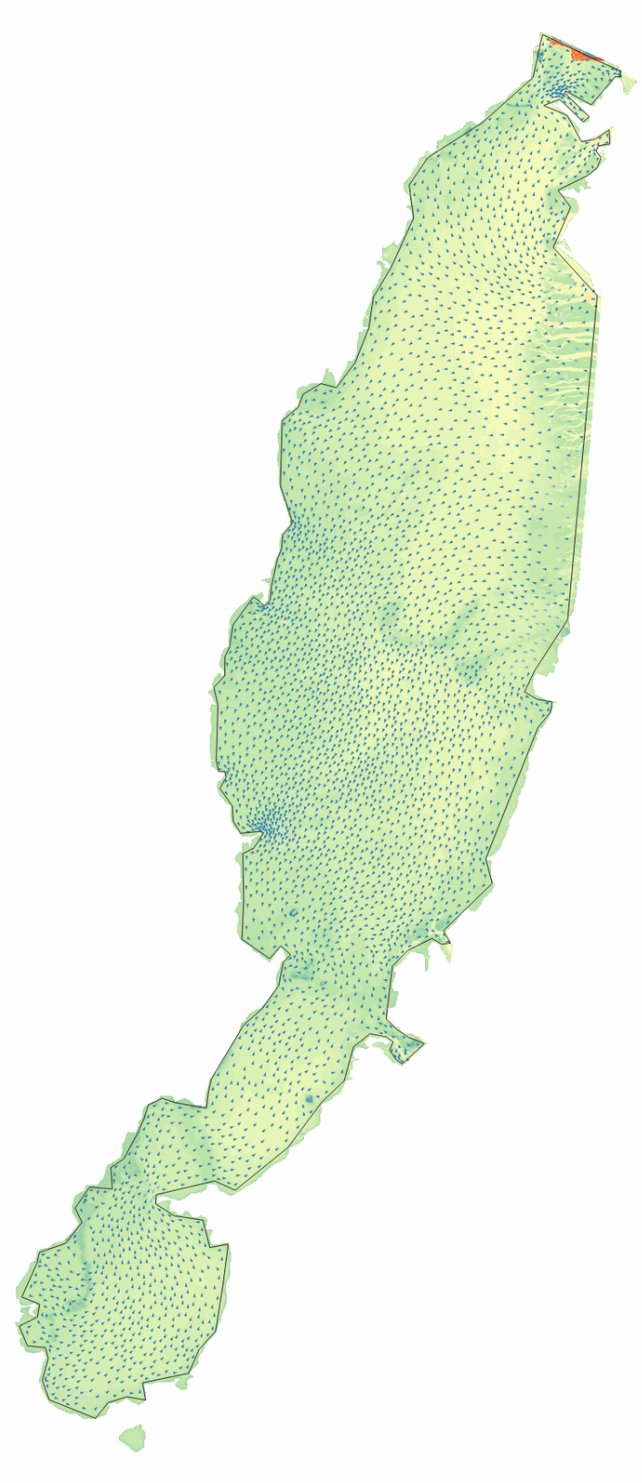
1. Select the Animation tool  :

   Wait for the animation to load. It will appear in the *Animation list*.

2. On the Animation panel, click *Add Layer*, and select the 'Velocity Field'

3. Then on the Animation panel, select the Velocity_field layer and click  to view the animation

   To see the animation, the layer needs top be moved above the DEM and / or other satellite map layers.



Detail of the velocity field during the animation.

This concludes the tutorial *Oil spills in a coastal area. Part I: Coastal Hydrodynamics*.

10

Oil spills in Coastal Areas. Part II: Oil Spill

This is the second part of the Oil spills in coastal areas tutorial. The first part explained how to setup and run the hydrodynamic stage, where the model was used to generate the velocity field in a bay. In this section we will show how to enter the spill locations, and other oil data to simulate oil trajectories and fate.

10.1 Opening the project with results obtained in Part I

We will use the files resulting from Part I of the tutorial that should have been generated before in the folder under '\ExampleProjects\OilSpillsInaBayPart1'.

Open QGIS and then open the project resulting from Part I, where we completed the hydrodynamic simulation of the bay and created a velocity field map:

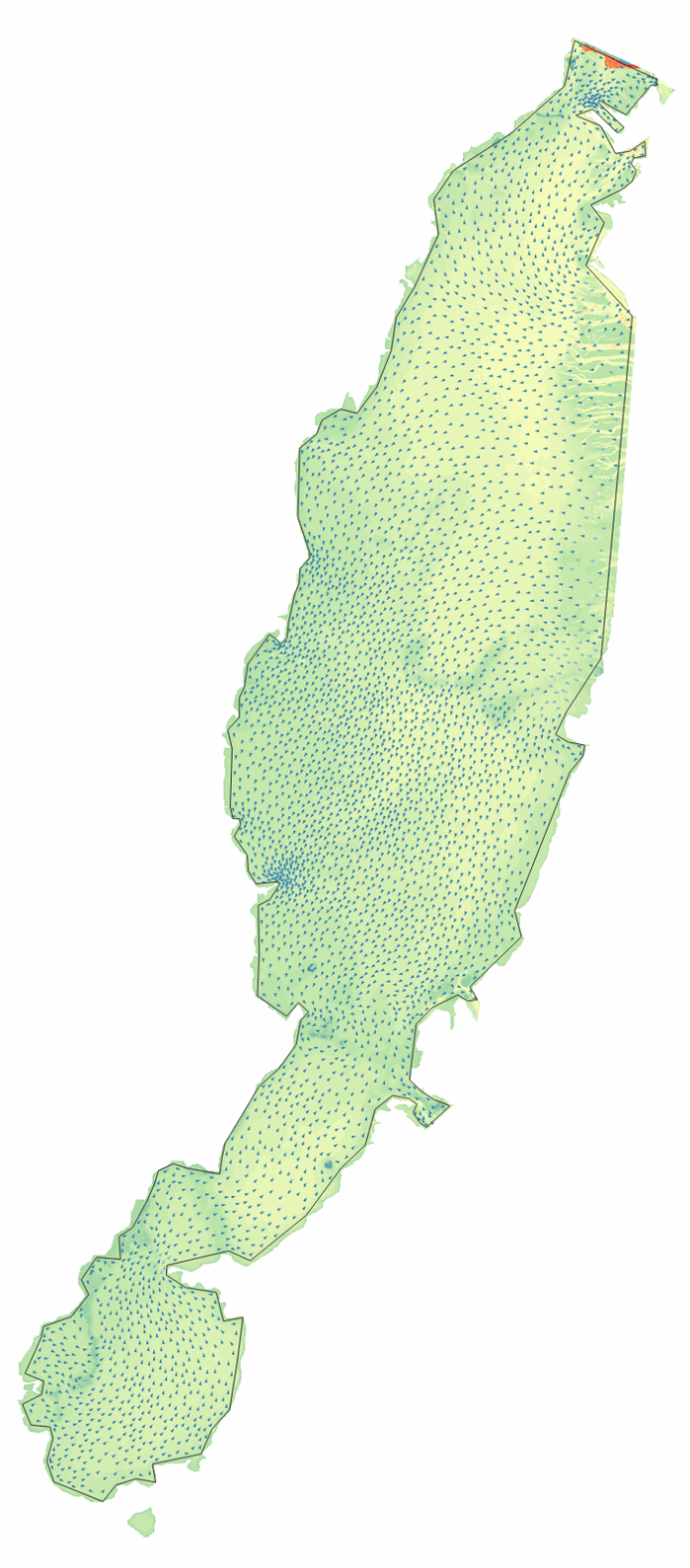
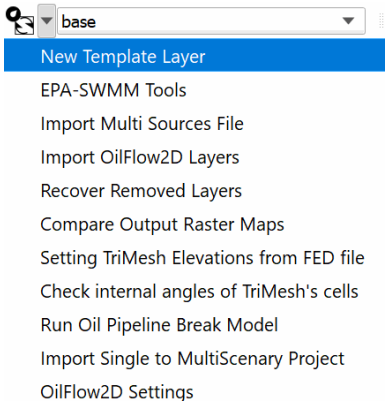


Figure 10.1 – Resulting project from the tutorial Oil Spill in Coastal Areas Part I

Alternately you may choose to open a premade version of Part 1 project located in the *OilSpillsInaBayPart1Final* folder.

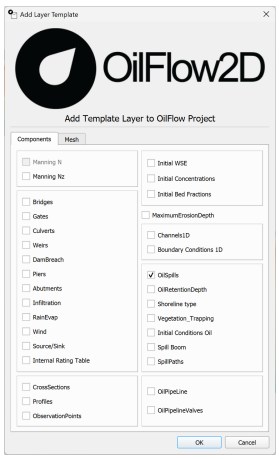
10.2 Create the OilSpills layer and enter the spill location and data

1. Create a new *OilSpills* layer: for this, go to the OilFlow2D toolbar of and click on *New Template Layer*



image

2. In the *Add Template Layer dialog*, select the *OilSpills* checkBox, as shown in the Figure below and click OK.

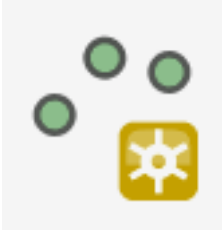


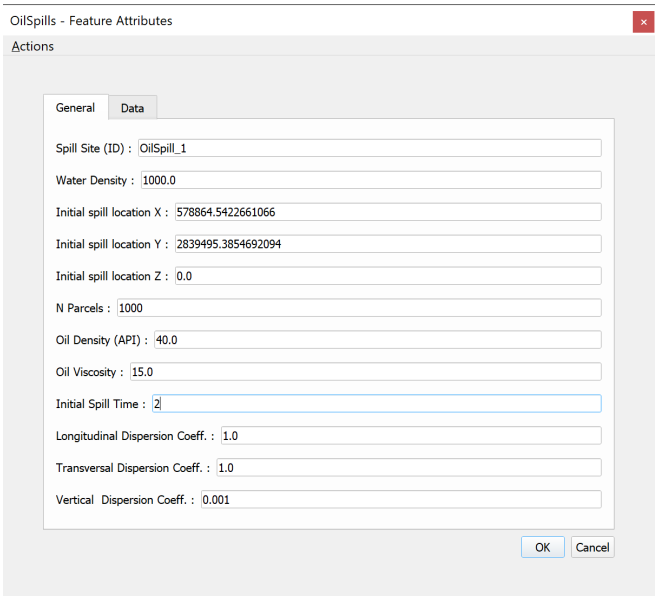
OilFlow2D dialog to add the new layer.

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

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

.

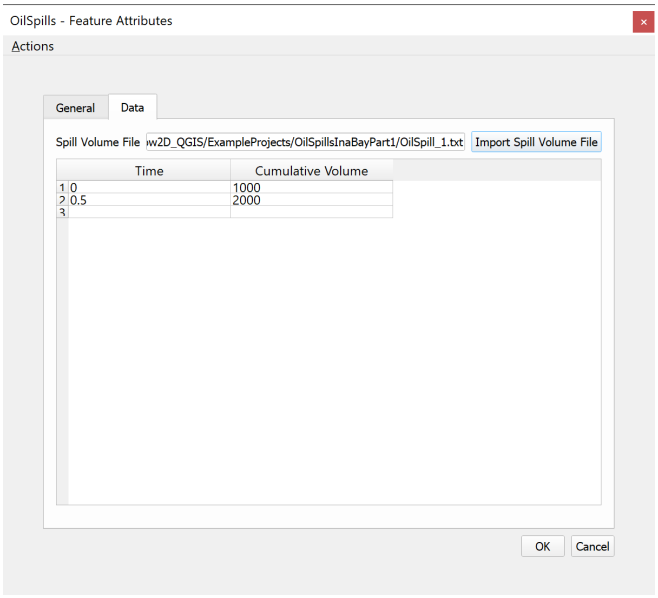
4. To enter the spill location click on the *Add Point Feature* button of the digitalization toolbar  and then click on the location where you want to set the spill point. The oil spill dialog will appear. Enter the data as indicated in the figure:



Oil Spill Dialog.

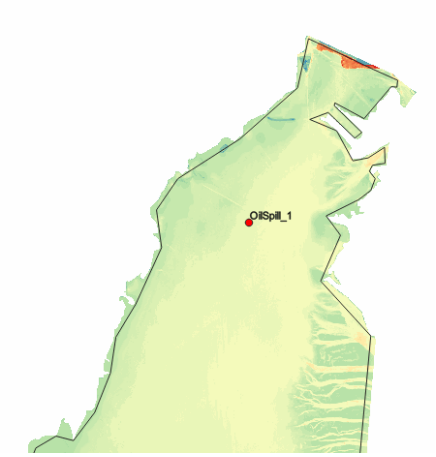
5. Click on the *Data panel* and open the spill volume file using the *Import Spill Volume* File button.

6. Browse to the project 'DATA' folder and select 'Oilspill_1.txt'.



Oil Spill Volume File.

7. Click OK to close the dialog and the oil spill point will be shown:



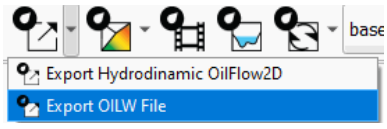
Oil Spill Point.

8. Save the changes to the layer by clicking the *Save* button 

9. Click on the *Toggle editing* button  to finalize the editing mode.

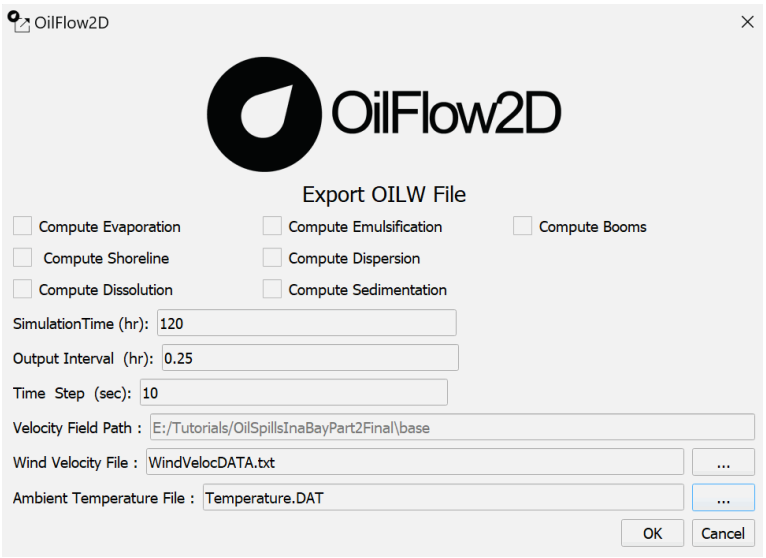
10.3 Export Oil Spill Data

1. Use the Export OILW command to export the data to run the model



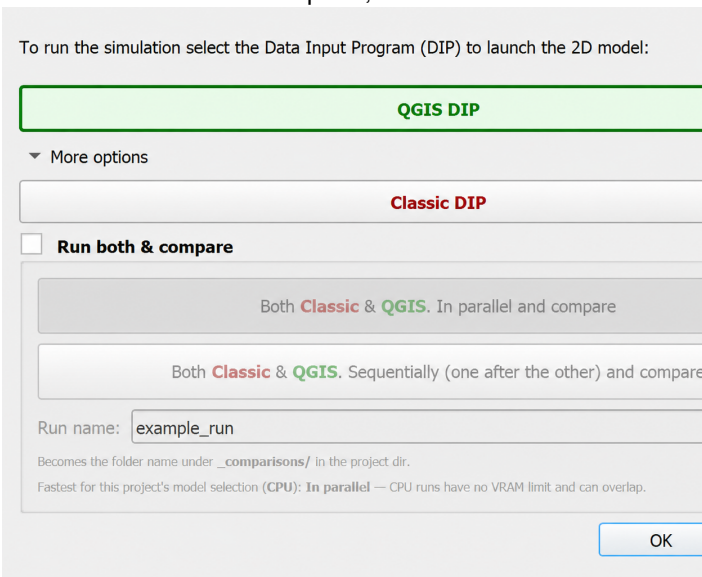
Export OILW command.

2. In the dialog enter the simulation time (48 h), output interval (0.25 h), the time step (10 sec.), import the wind velocity data ('WindVelocData.DAT') file, and ambient temperature file ('Temperature.DAT') from the 'DATA' folder as shown, and click OK.

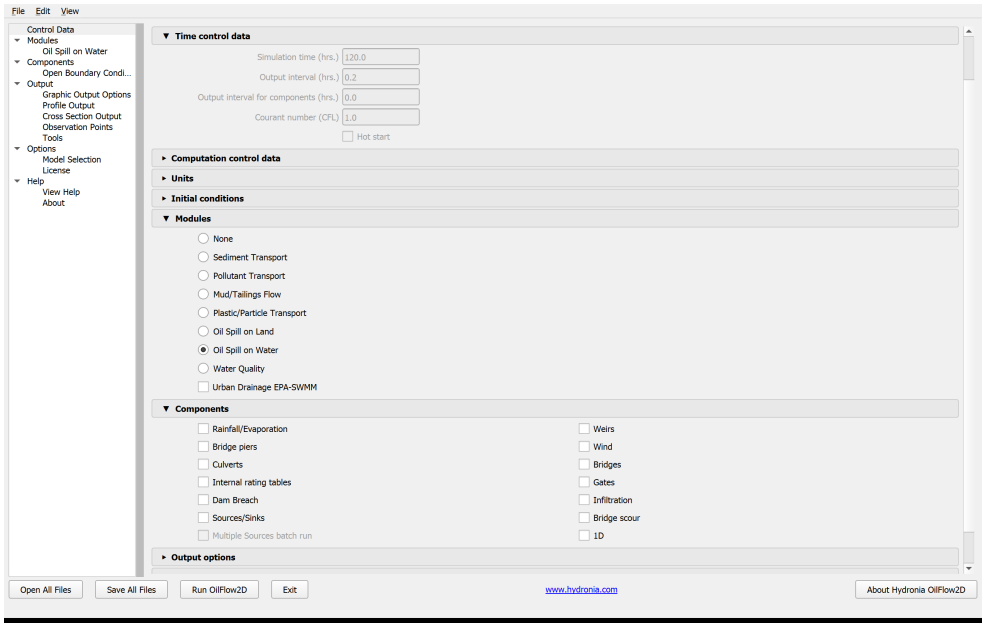


Export OILW Dialog.

After export completes, version 10 and later users see the Open DIP dialog. QGIS DIP is selected by default; click OK to open it inside QGIS. To use the historical standalone option, expand More options, select Classic DIP, and then click OK.



With QGIS DIP selected, the current interface is shown below:



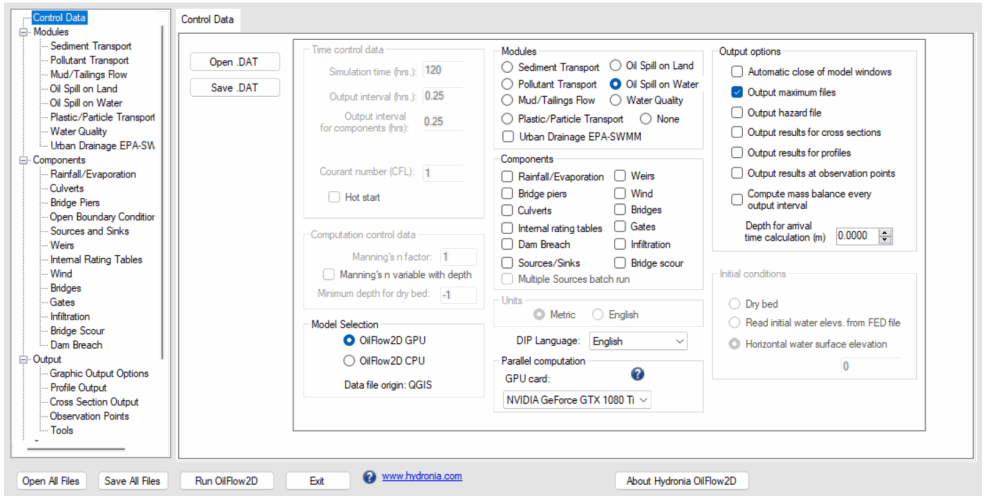
QGIS DIP Control Data panel for Oil Spills in a Bay Part II.

QGIS DIP and Classic DIP use the same project settings, but their interfaces look different. Older standalone or non-maintenance versions may open Classic DIP automatically. The project-specific Classic DIP settings and Run steps below are retained.

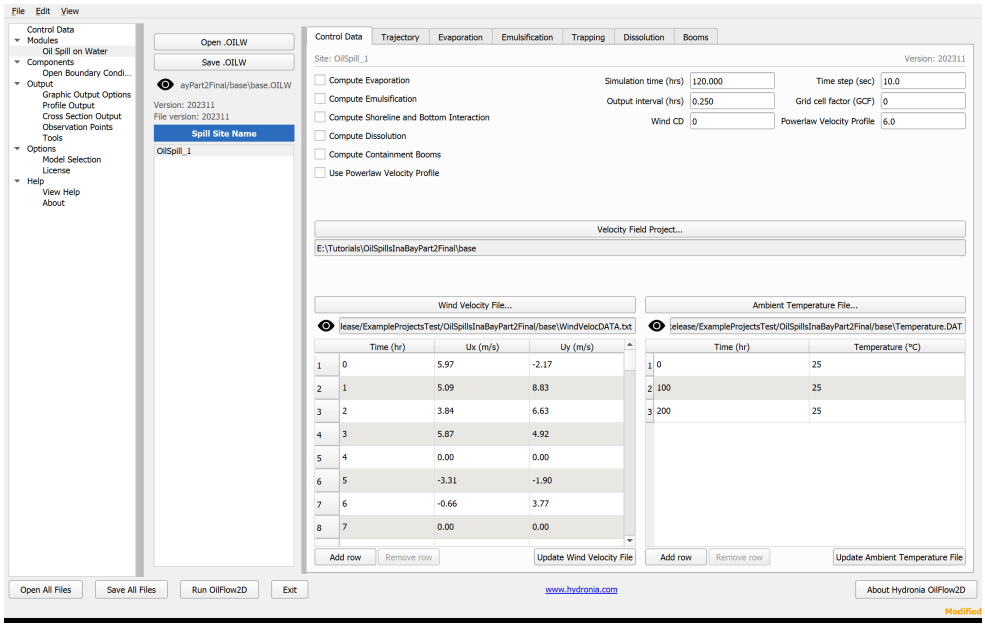
10.4 Enter additional data in the Hydronia Data Input Program and run OilFlow2D

1. Review the data in the Hydronia Data Input Program panels:

   The following Classic DIP Control Data panel is retained as a legacy values reference:

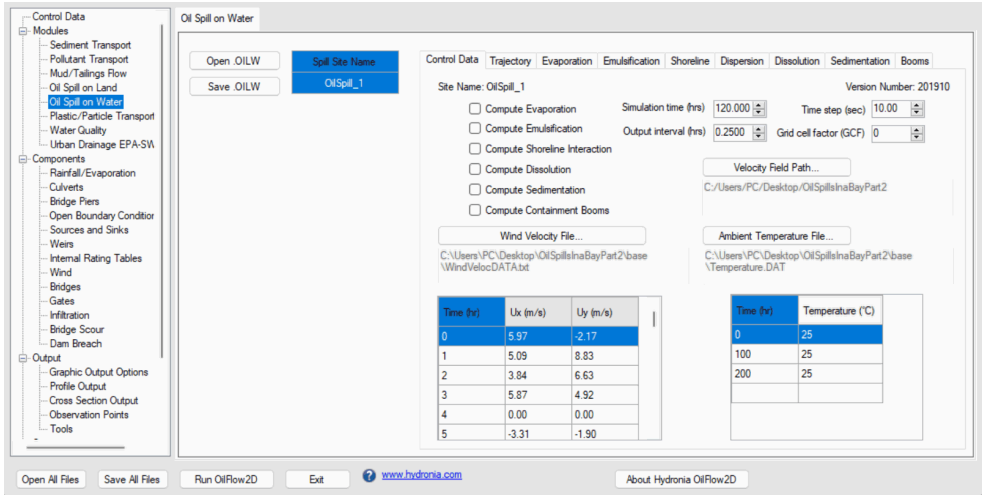


Legacy Classic DIP Control Data panel.

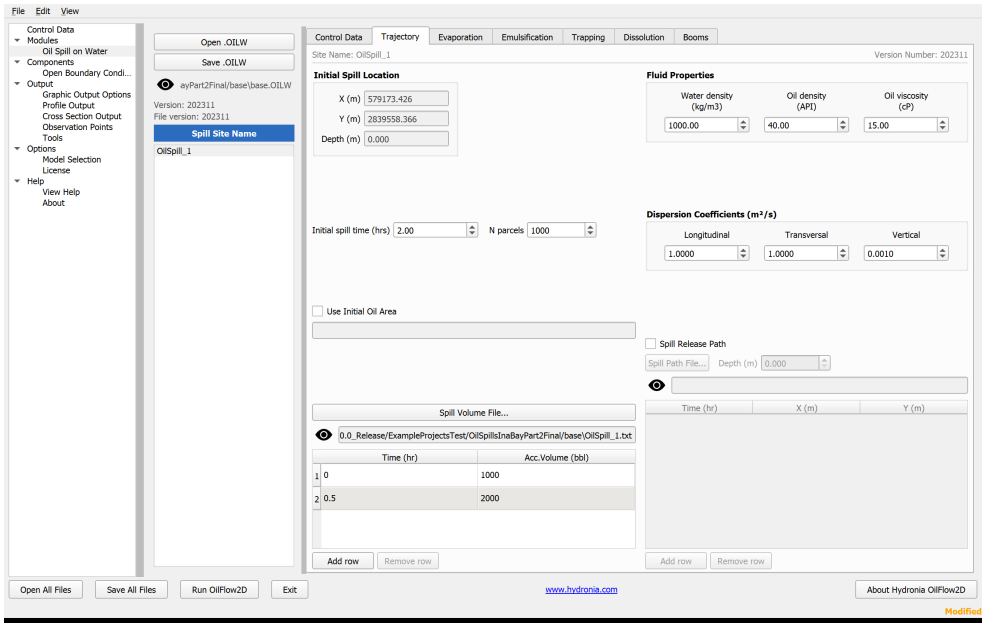


QGIS DIP Oil Spill on Water Control Data panel.

The following Classic DIP Oil Spill on Water Control Data panel is retained as a legacy values reference:

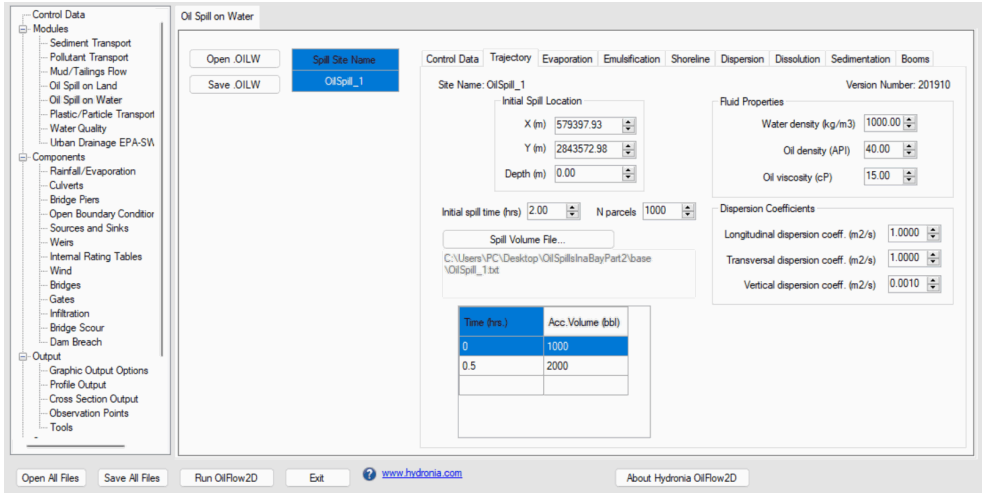


Legacy Classic DIP Oil Spill on Water Control Data panel.



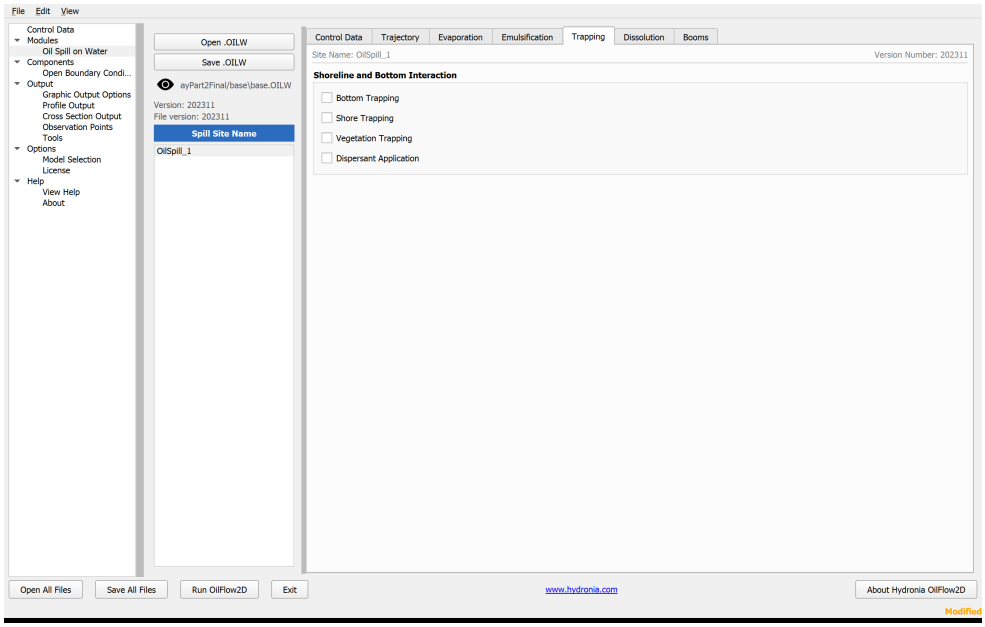
QGIS DIP Oil Spill on Water Trajectory panel.

The following Classic DIP Oil Spill on Water Trajectory panel is retained as a legacy values reference:



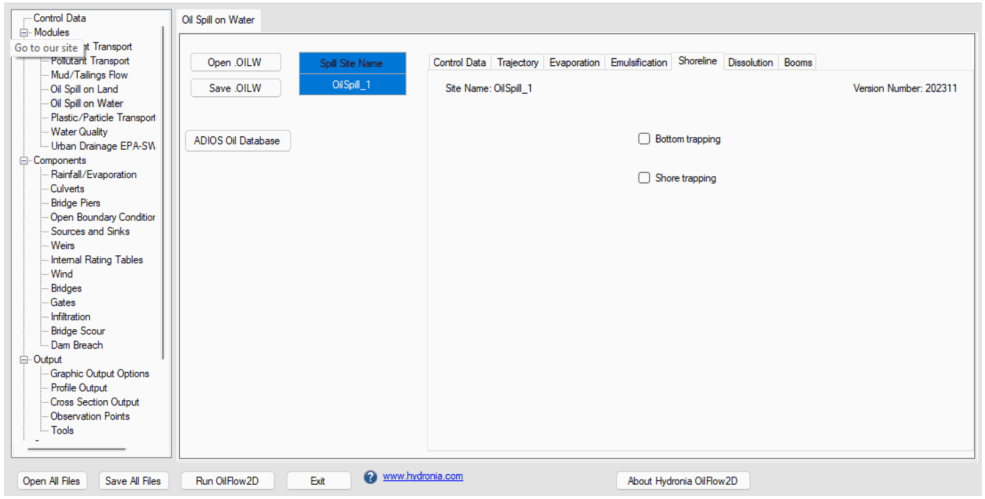
Legacy Classic DIP Oil Spill on Water Trajectory panel.

2. Uncheck the *Shoreline trapping* checkbox in the *Shoreline* panel.



QGIS DIP Oil Spill on Water panel with Shoreline trapping unchecked.

The following Classic DIP shoreline panel is retained as a legacy values reference:

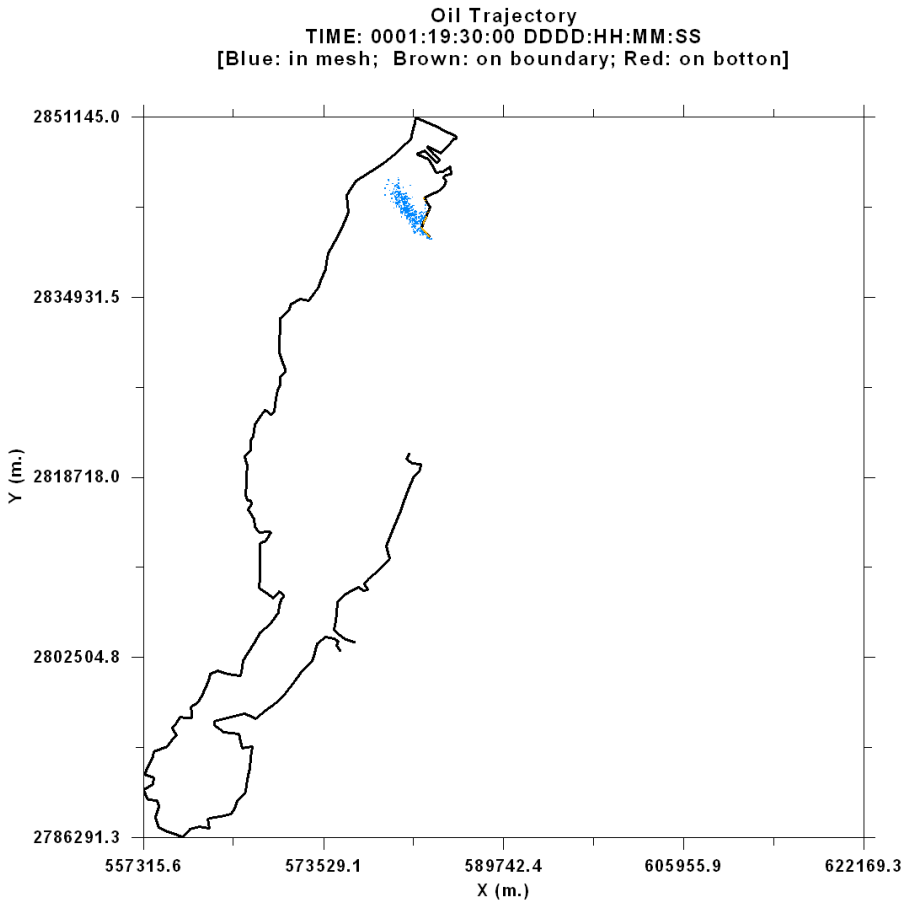


Legacy Classic DIP shoreline panel.

3. Click the [Save .OILW] button to save the changes to the .OILW file.

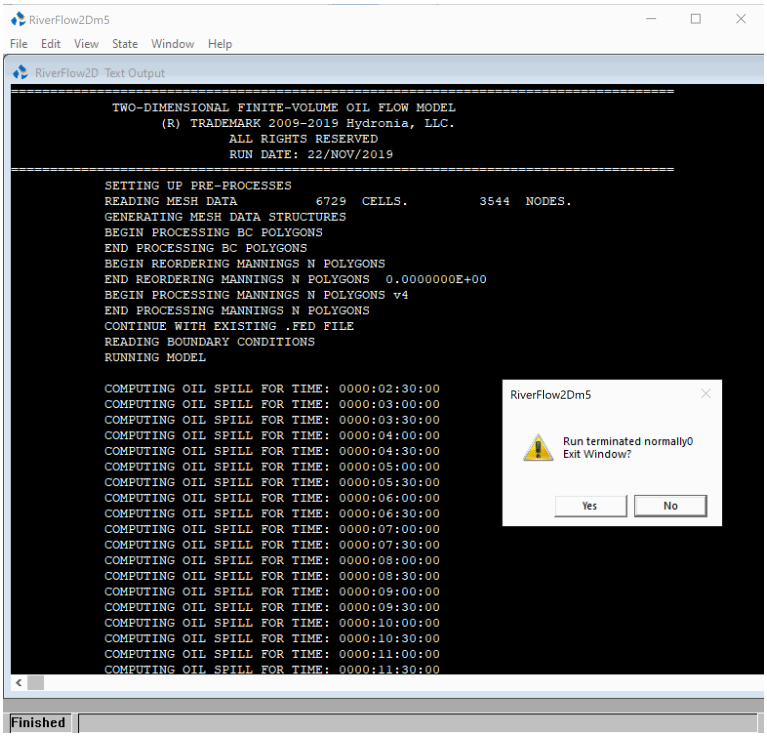
4. When ready to run the model click *Run OilFlow2D*.

5. A plot window will open while the model runs:



Model plot window.

The model run window will report the times being computed and a message will indicate that the model finished running:



Model execution window.

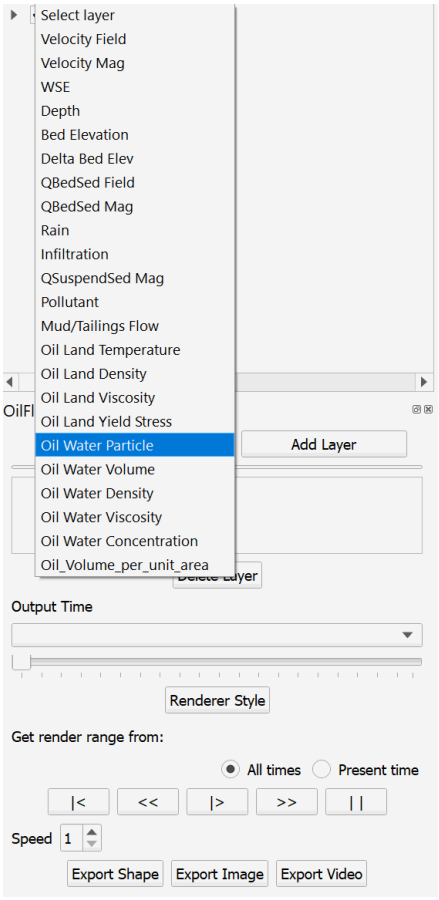
6. Click *Yes* to close the window.

10.5 Creating a Oil Spill Trajectory Animation

To check the model results we can create a oil trajectory animation using the following procedure:

1. Select the Animation tool  :

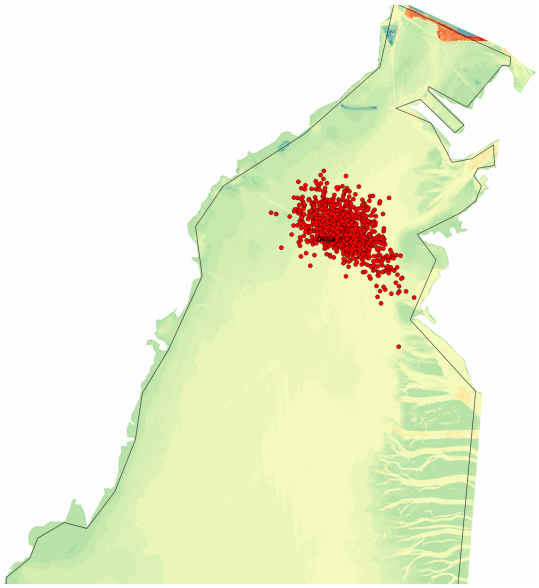
2. On the Animation panel, click *Add Layer*, and select the 'Oil Water Particles' drom the drop-down menu as shown:



Select Oil Water Particles animation.

3. Then on the Animation panel, select the Oil_Water_Particle_01 layer and click 

to view the animation



Oil trajectory during animation.

This concludes the tutorial *Oil spills in Coastal Areas. Part II: Oil Spill*.

11 Wind driven circulation

OilFlow2D and OilFlow2D allow defining wind velocity on the water surface to account for the effect of the wind stress on the flow velocities. The conceptual model of a wind driven simulation require a series of non-overlapping polygons that determine the wind velocity data to the model. Only areas covered by polygons will be affected by the wind stress. Each wind velocity polygon should be associated with a file containing a wind velocity time-series file containing the two components of the wind velocity vector for each time. The user will need to generate the wind velocity data file associated with each polygon, and copy them to the project folder, prior to running the model.
This tutorial illustrates how to perform a wind drive simulation using the QGIS interface. The procedure includes the following steps:

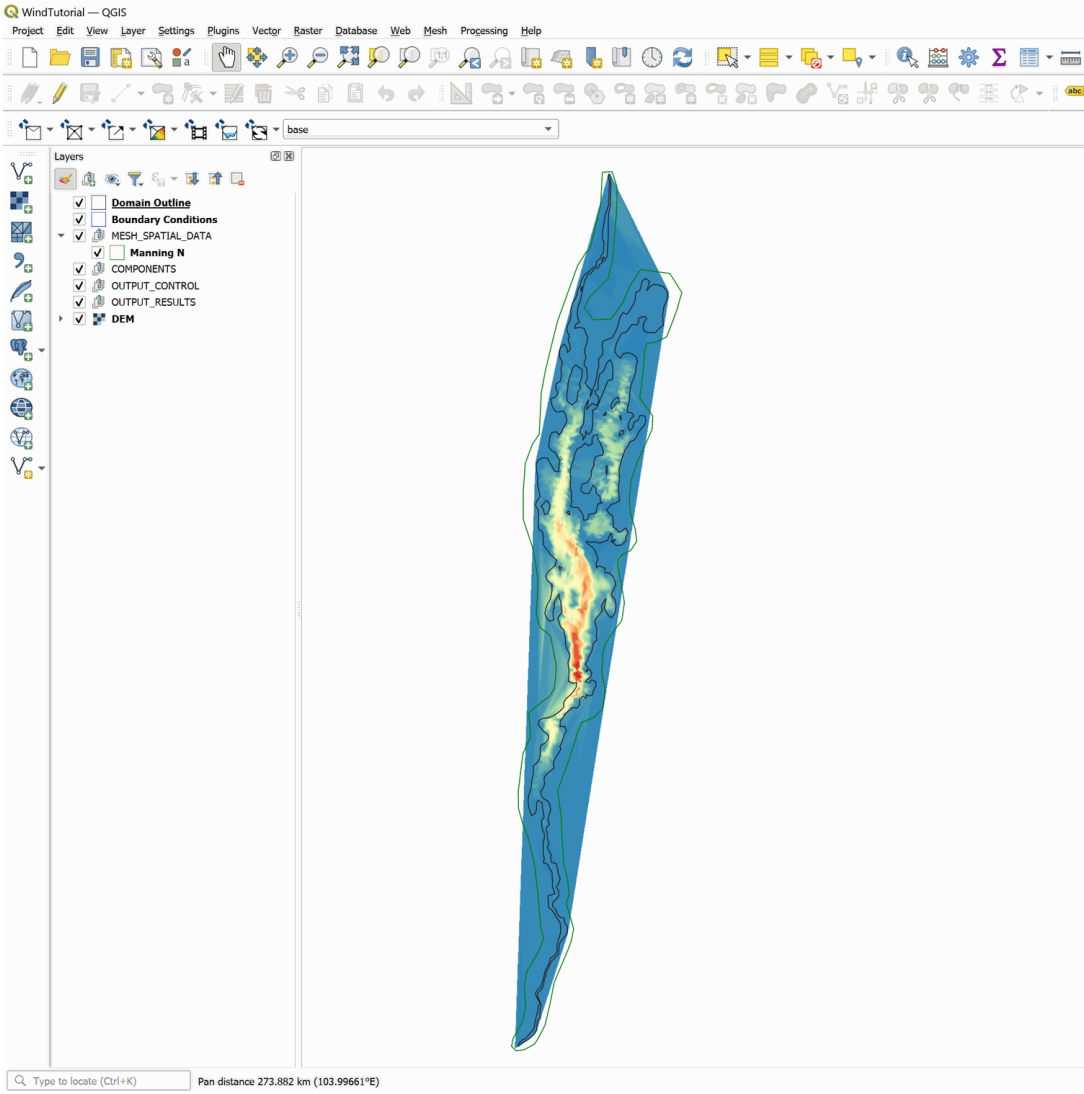
1. Create time series data for wind speed.
2. Open an existing OilFlow2D project.
3. Create the template of the wind layer and the wind speed polygons.
4. Generate the mesh.
5. Running the model.
6. Review wind output files.

The files required to follow this tutorial can be extracted from the 'ExampleProjects' zip file under the 'WindTutorial' folder. This zip file is downloaded separately from your installation materials.

11.1 Open an existing project

1. Open QGIS
2. On the *Project* menu click *Open…* to load the existing project: 'WindTutorial.qgz'.

This project contains the layers of the domain contour and the layer of the Digital Elevation Model (DEM) of Lake Champlain in the USA. When the project is opened, a project image will be loaded in QGIS as shown in Figure 15.1.



11.2 Wind velocity time-series data file

To run a wind driven simulation you create polygons over which the wind velocity data will be applied. Each polygon will have an associated velocity time series. These files can be created with any text editor such as Notepad or Wordpad. The wind velocity file has the following format:

Line 1: Number of points in the time series of wind velocity

NP

NP lines containing:

TIME Wvx Wvy

where Wvx and Wvy, are the wind velocity components in *x* and *y* directions respectively in m/s or ft/s.

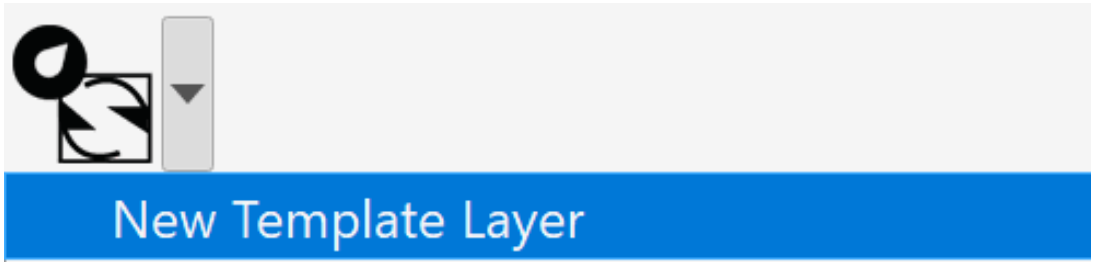
The following table is an excerpt of the 'WindVelocDATA.TXT' file that is included in the Data folder for this tutorial:

```
6544
0          5.97        -2.17
1          5.09         8.83
2          3.84         6.63
3          5.87         4.92
4          0.00         0.00
5         -3.31        -1.90
...
6543       3.84        -6.63
```

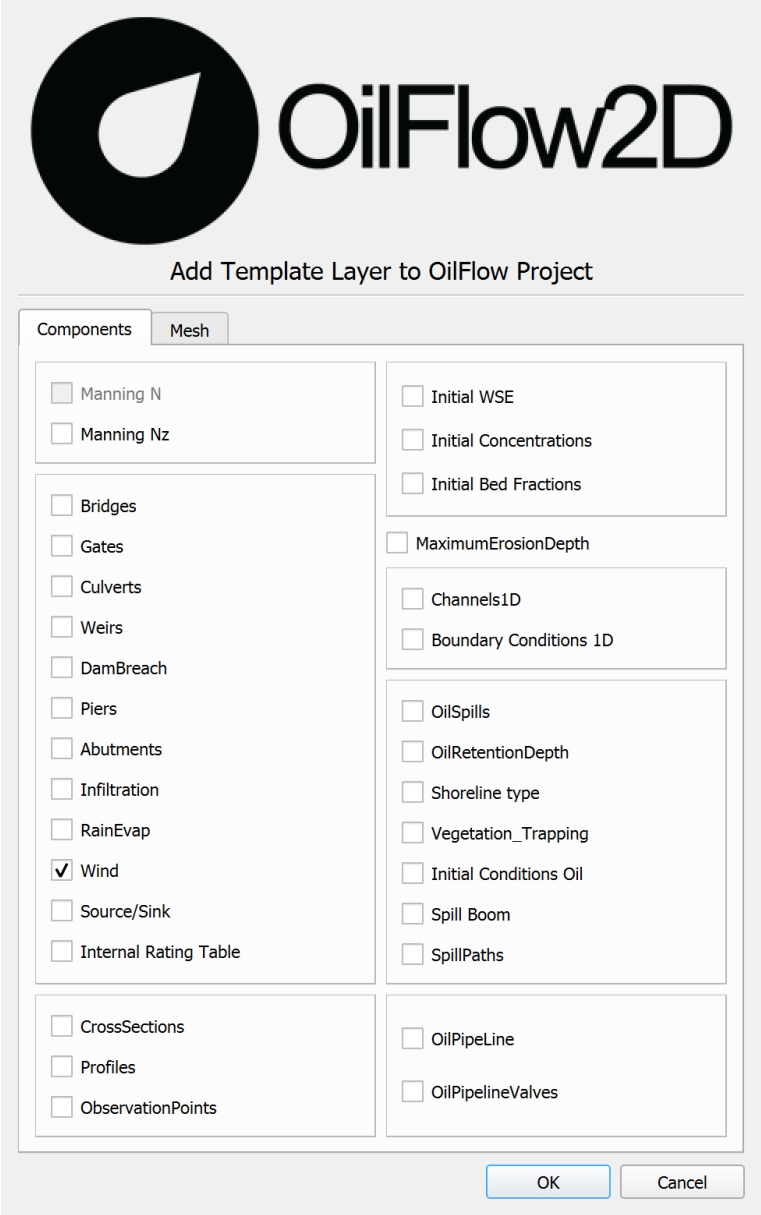
11.3 Create the template for the wind layer and the wind speed polygons

To add the template where the polygons are drawn with the wind speed time series data involves the following steps:

1. Create the template for the Winds layer: In the model toolbar click on the *New Template Layer* command



2. The existing WindTutorial.qgz project uses the base scenario and EPSG:6623. In the *Add Template Layer to OilFlow Project* window, activate the *Wind* checkbox, as shown in the Figure below:



Add Template Layer to OilFlow Project with Wind selected.

3. Edit the Wind layer: In the layers panel, we select the Wind layer and in the digitization toolbar we click on the *Toggle Editing* tool  . A pencil icon will appear in the Wind layer, indicating that the layer is in edit mode:



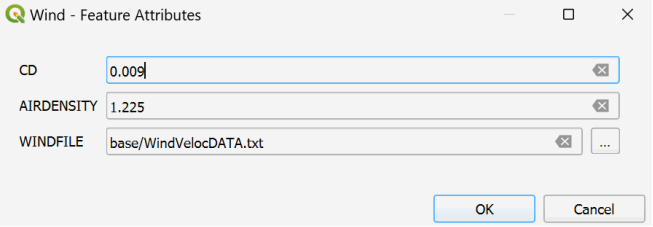
4. Draw the polygon that demarcates the Wind area: Using the *Add Feature* tool from the digiti-

zation toolbar  .

Draw the polygon that defines the wind area. In this case, the tracing of the polygon must be done in such a way that it covers all the cells of the mesh. Once you finish drawing the polygon a window to input the polygon parameters is immediately opens, which are three:

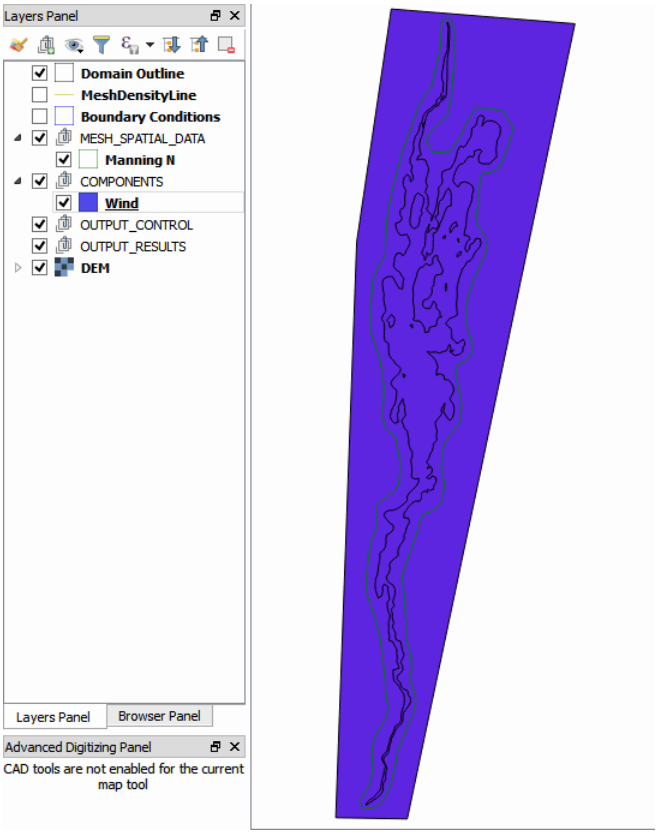
- Wind stress coefficient CD: 0.009,
- Air density: 1.225, and
- Wind speed time series File: 'WindVelocDATA.txt'.

The parameter window should be as shown below:



Window to input the Wind polygon parameters.

5. After input the values, click *OK* and accept the changes. There should be an image similar to the one shown in the following figure:

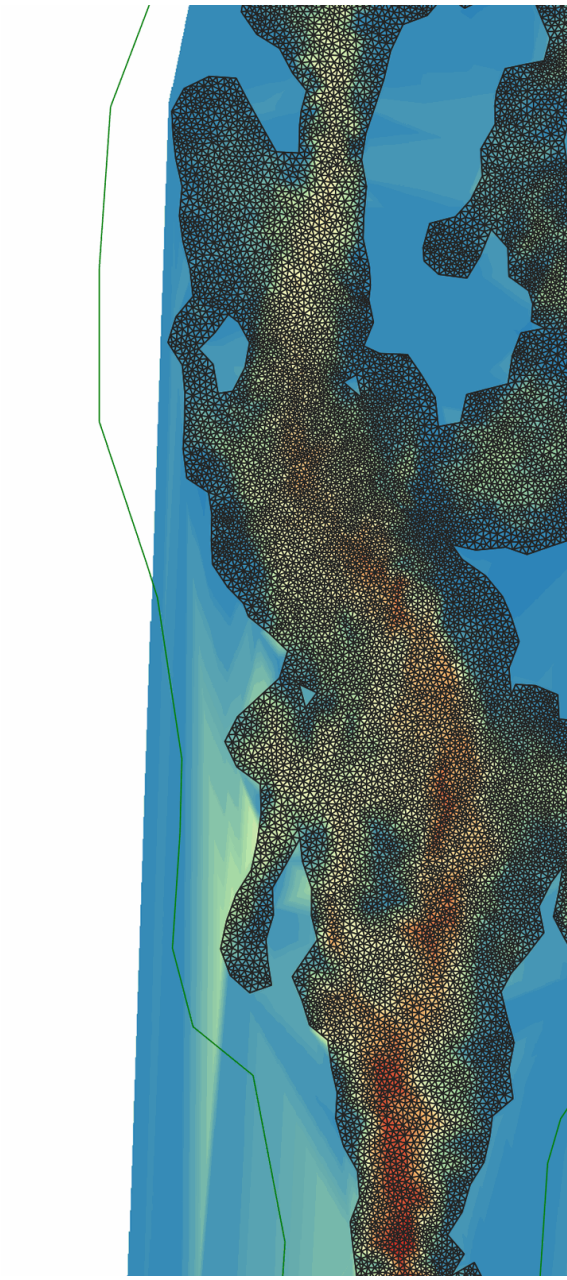


Wind layer polygon.

11.4 Generate the mesh

Then the mesh is generated with the *Generate TriMesh* button





The results obtained as shown in Figure 15.5 (mesh of around 17,500 cells).

11.5 Exporting files

After generating the mesh and preparing the other layers, export the files in the format required by OilFlow2D.

1. Uncheck the *Boundary Conditions* layer.

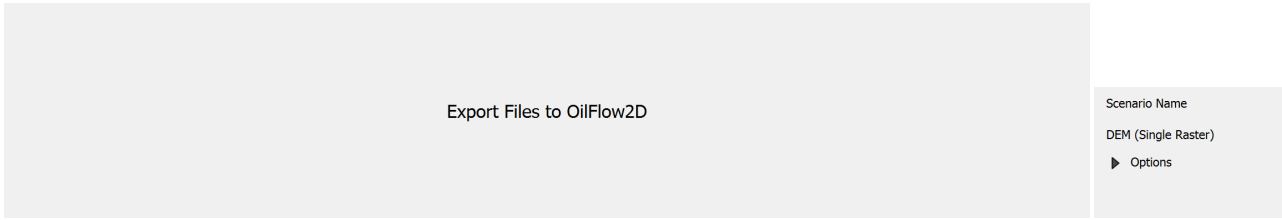
2. Activate the DEM layer.

3. Use the *Zoom Full* button  to center the image if required.

4. Open Plugins > OilFlow2D MS > Export OilFlow2D and click the exact visible action Export Hydrodinamic OilFlo

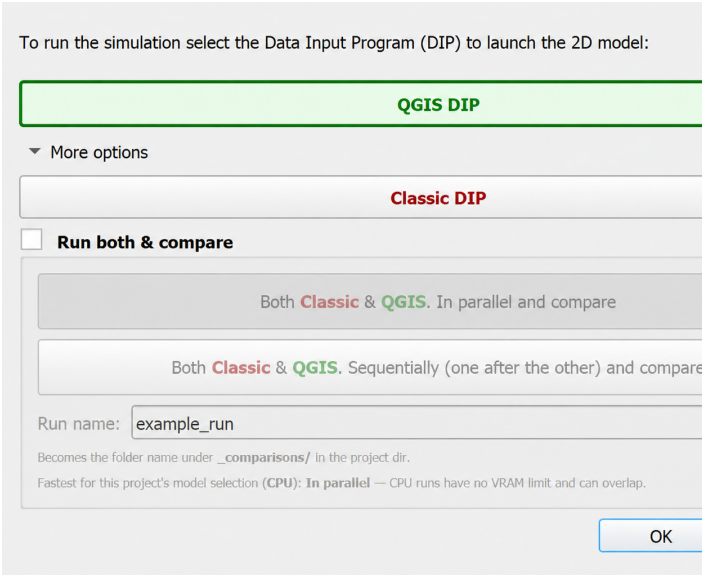


5. In the OilFlow2D export dialog, keep the Scenario Name value at base and choose DEM in the DEM (Single Raster) field. The dialog labels the operation Export Files to OilFlow2D.



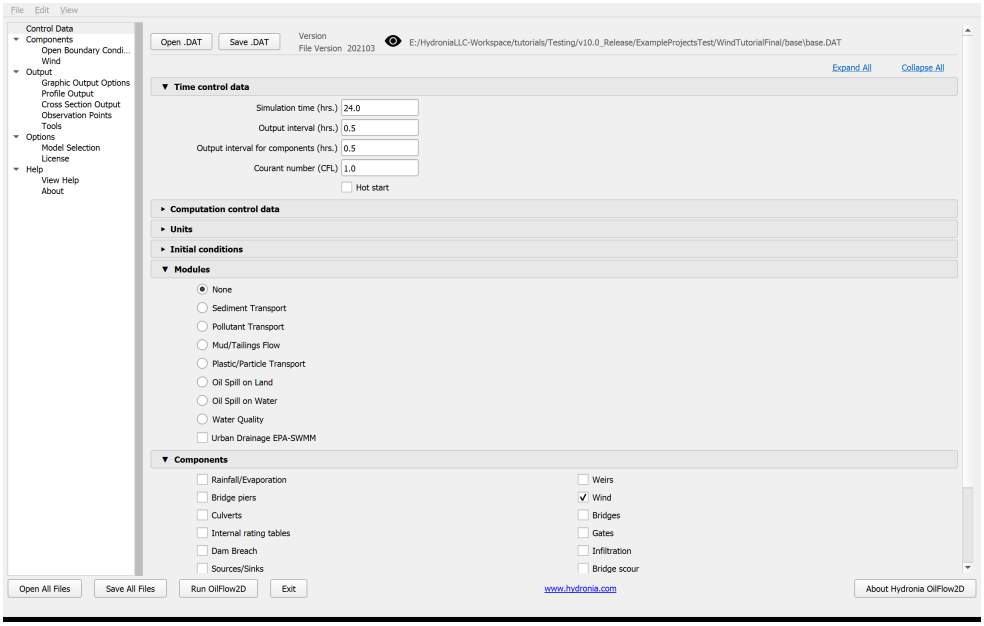
6. Click OK after checking the values.

   After export completes, version 10 and later users see the Open DIP dialog. QGIS DIP is selected by default; click OK to open it inside QGIS. To use the historical standalone option, ex-



pand More options, select Classic DIP, and then click OK.

With QGIS DIP selected, the current interface is shown below:



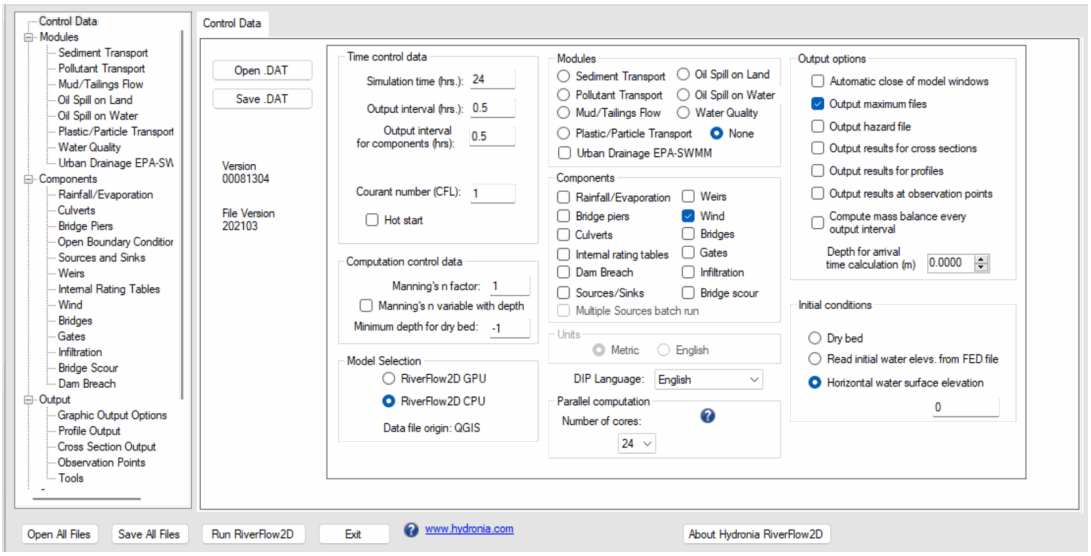
QGIS DIP Control Data panel for the Wind tutorial, with no modules selected.

QGIS DIP and Classic DIP use the same project settings, but their interfaces look different. Older standalone or non-maintenance versions may open Classic DIP automatically. The project-specific Classic DIP settings and Run steps below are retained.

11.6 Running the model

After exporting the files, Hydronia Data Input Program is loaded with the project file from the 'Wind-Tutorial.DAT' example and shows the *Control Data* panel.

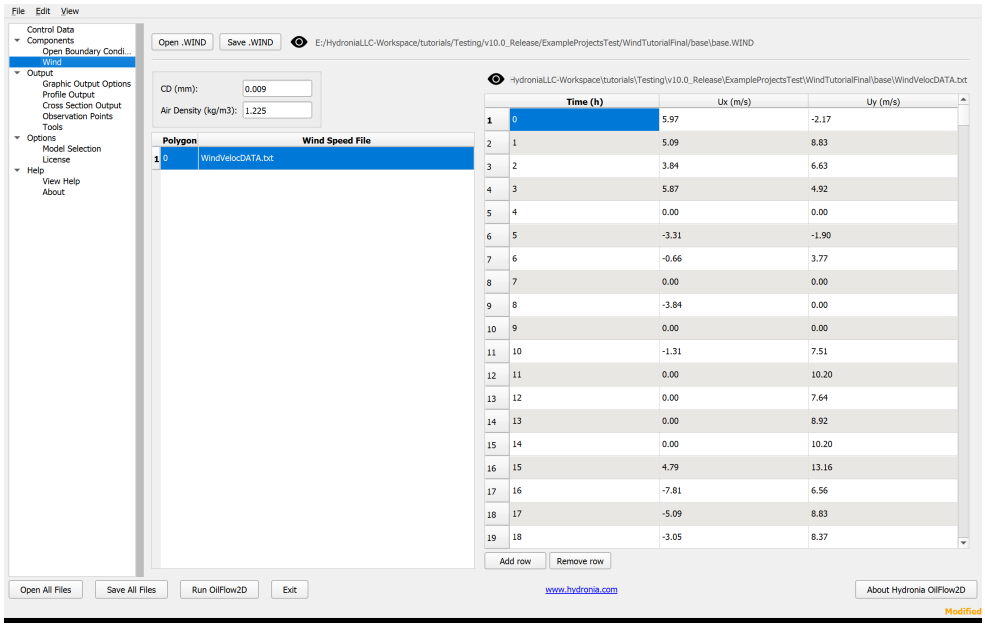
1. Enter the information as illustrated in Figure 15.7 The following Classic DIP Control Data panel is retained as a legacy values reference:



Legacy Classic DIP Control Data panel.

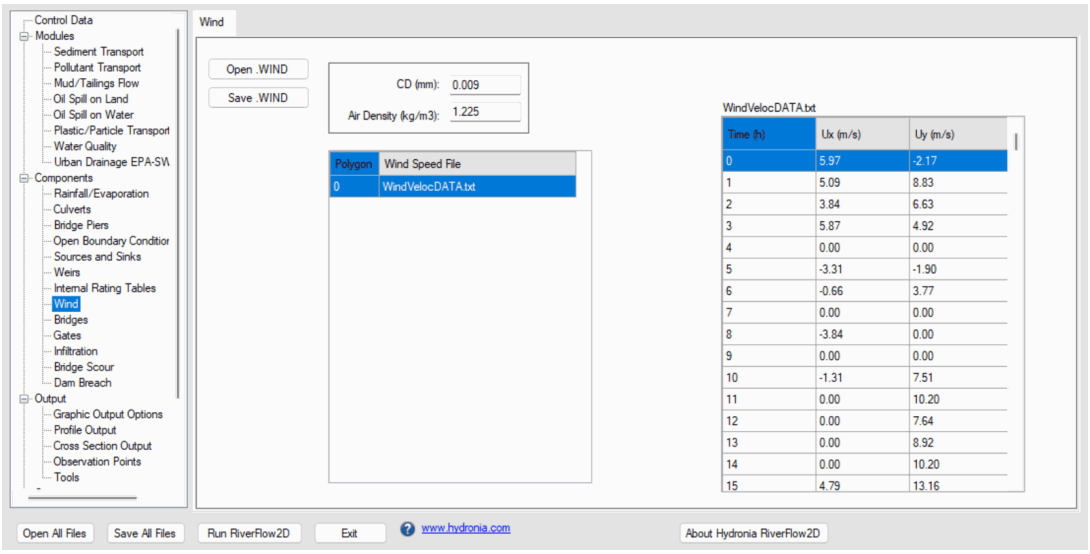
2. Verify that the *Wind* component is selected.

3. Select the *Wind* component from list in the left side of the panel. The window with the information of the wind parameters will appear as can be seen in the figure below:



QGIS DIP Wind component panel.

The following Classic DIP Wind panel is retained as a legacy values reference:



Legacy Classic DIP Wind panel.

4. Verify that the simulation time is set to 24 hours and the output interval is set to 0.5 hours.

5. Verify that the *Initial conditions* is set to *Horizontal water surface elevation* and 0 on the text box. Leave all other parameters at their default values.

6. Click on the *Run OilFlow2D* button in the lower section of Hydronia Data Input Program.

7. Save the changes with the same name as the ‘base.DAT’ file, then a window will appear indicating that the model started running.

The model window that appears during the run model shows several runtime parameters.

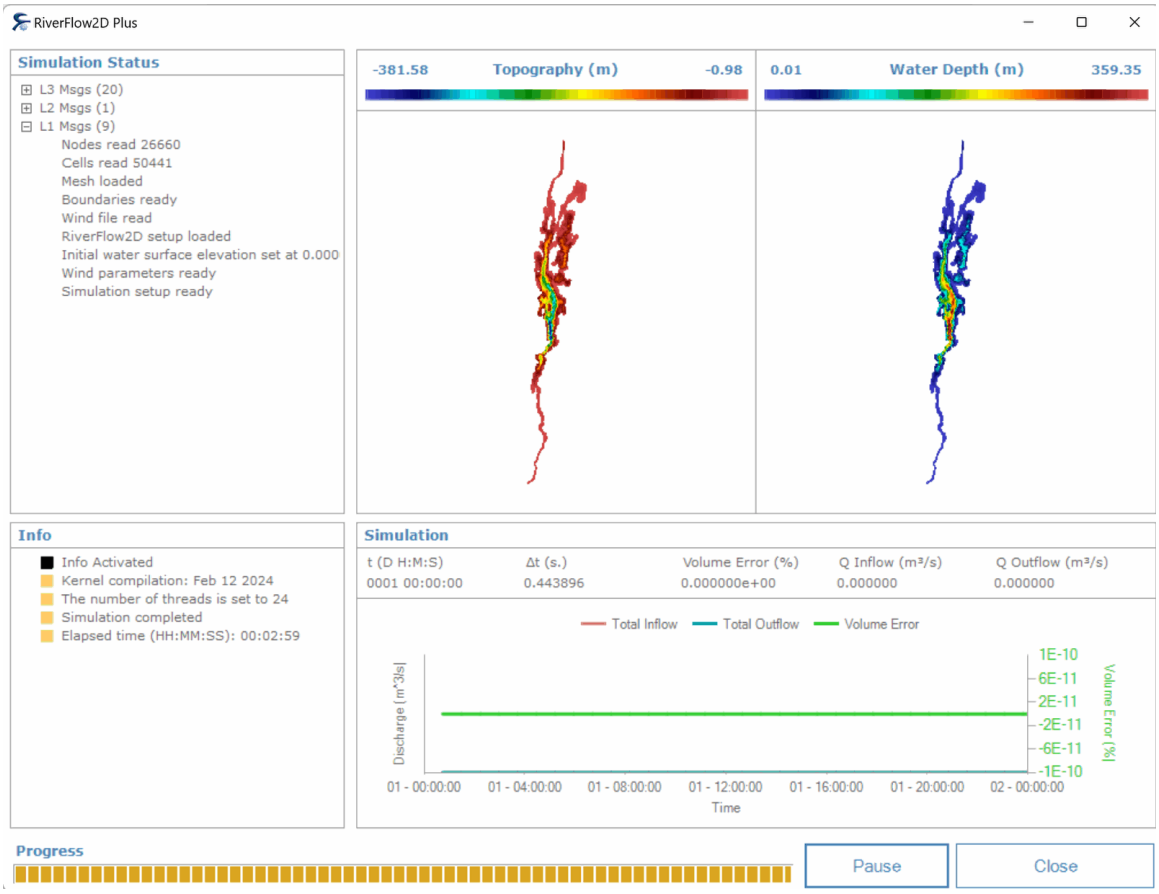
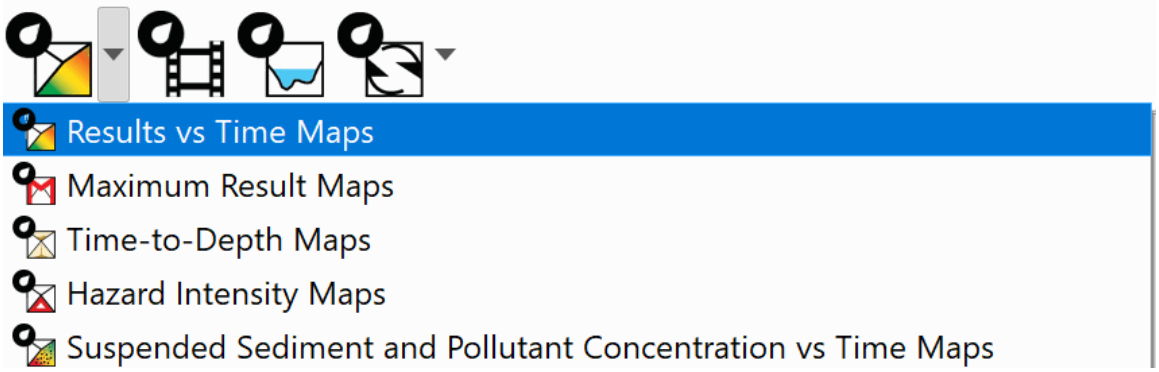


Figure 11.1 – Model window.

11.7 Check the wind output files

The model creates the following files for each output time as defined in the Control panel:
'CELL_TIME_METRIC_DDDD_HH_MM_SS.TEXTOUT' (Metric Units) or
'CELL_TIME_ENG_DDDD_HH_MM_SS.TEXTOUT' (English Units)
where DDDD indicates the day, HH, hour, MM minutes and SS seconds.
In these files, columns 1, 2 and 3 report the velocity components in Vx, Vy and the module respectively. We can visualize the water velocity fields generating layers either in raster or vectorial format from the aforementioned files using the *Maps of Results vs Time* tool.



The following figure shows the water velocity field map for time 0000:20:00:00:

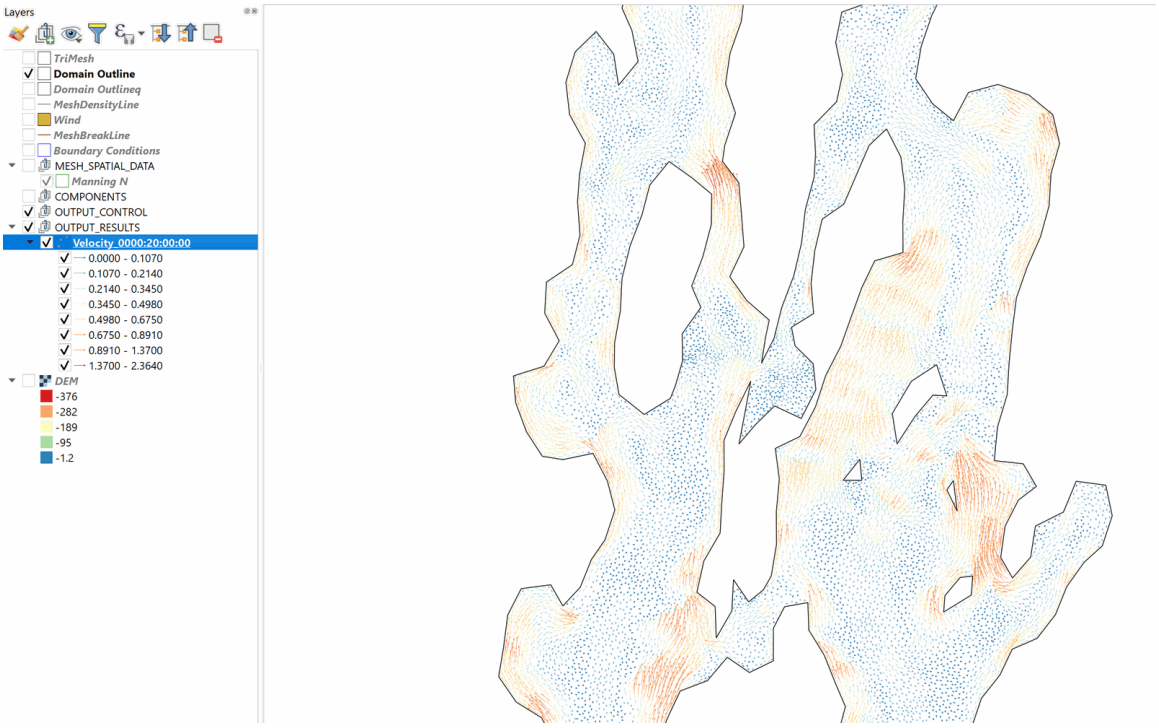


Figure 11.2 – Map with the speed field for the time 20 hours.

This concludes the *Wind driven circulation* tutorial.

12

Using Manning's n ESRI shape files

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

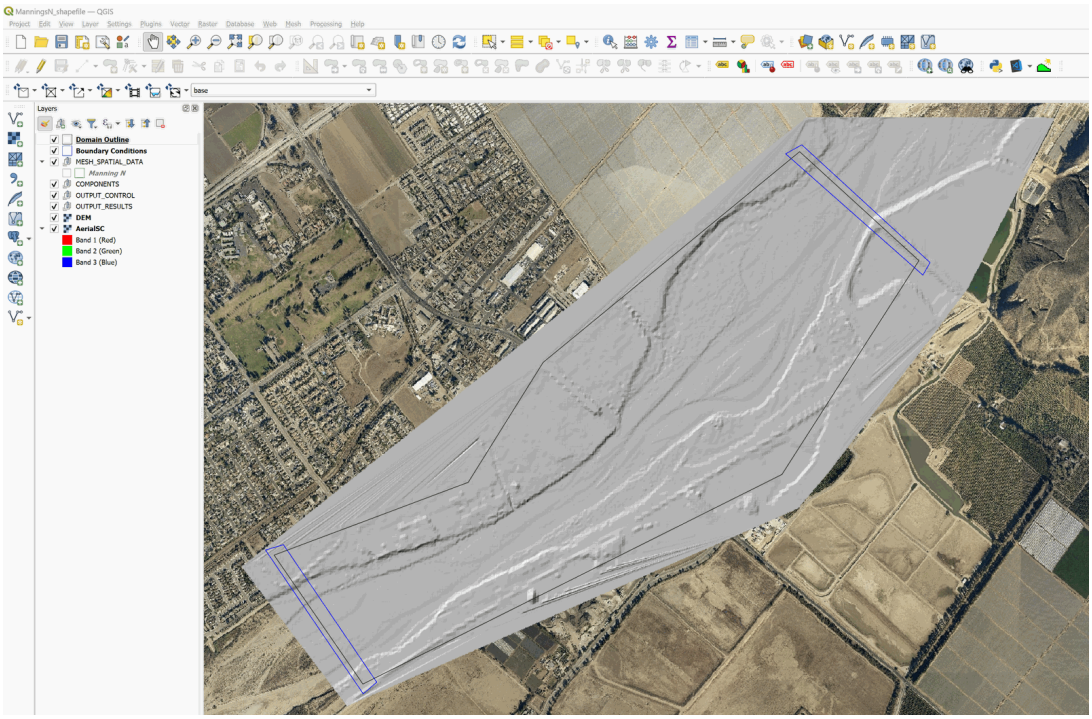
1. Open an existing OilFlow2D project.
2. Load the shape files with the Manning's n polygons.
3. Import the Manning's n geometry and values to the *Manning N* layer.

The files required to follow this tutorial can be extracted from the 'ExampleProjects' zip file under the 'ManningsNShapefileTutorial' folder. This zip file is downloaded separately from your installation materials.

12.1 Open an existing project

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

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



in 16.1.

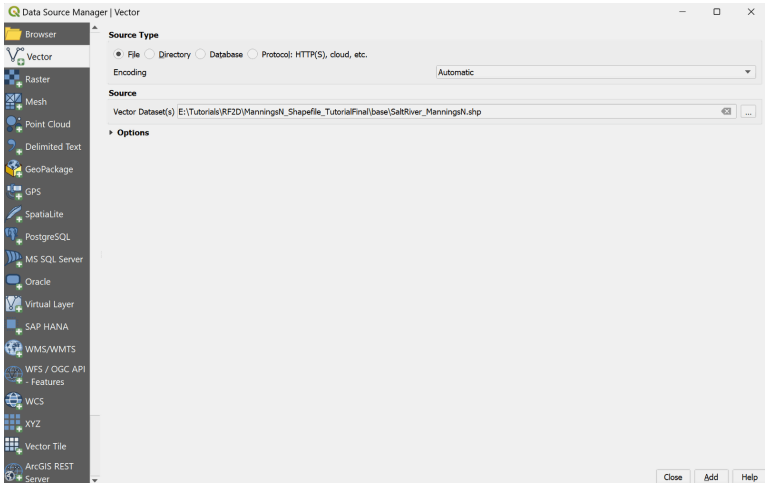
12.2 Load the shape file with the Manning's n polygons

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

   of Manager layer toolbar or from the main menu *Layer* > *Add layer* > *Add Vector Layer…*

2. In tutorial folder under the *base* subfolder, select the 'SaltRiver_ManningsN.shp' file (Figure 16.2).

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



SaltRiver_ManningsN Layer.

12.3 Import the Manning's n geometry and values to the Manning N layer

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

1. Check the field names of the shape file: Right-click on the *SaltRiver_ManningsN* layer label and in the pop-up menu select the option *Open attribute table* (Figure 16.4).

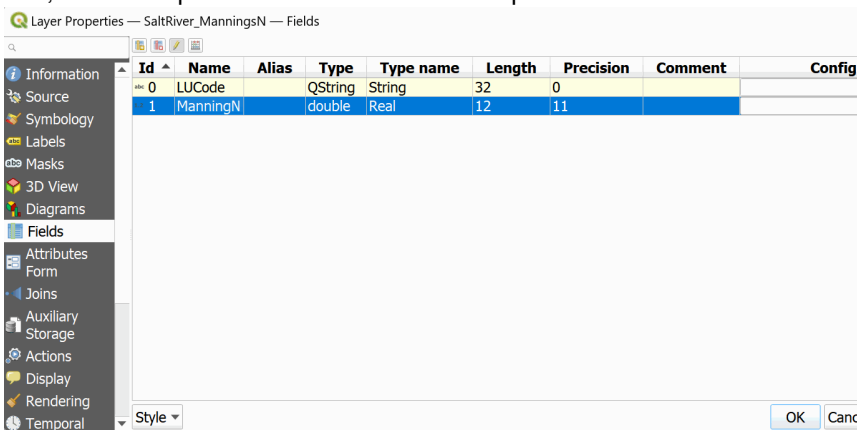
SaltRiver_ManningsN — Features Total: 69, Filtered: 69, Selected:

	LUCode	N_Value
1	Farm_medium	0.06
2	Veg_low	0.04
3	Veg_medium	0.055
4	Veg_medium	0.055
5	Farm_medium	0.06
6	Water	0.03
7	Farm_low	0.05
8	Veg_low	0.04
9	Veg_low	0.04
10	Veg_medium	0.055
11	Veg_medium	0.055
12	Veg_low	0.04
13	Farm_dense	0.085
14	Urban_blocked	0.13
15	Farm_low	0.05

Show All Features

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

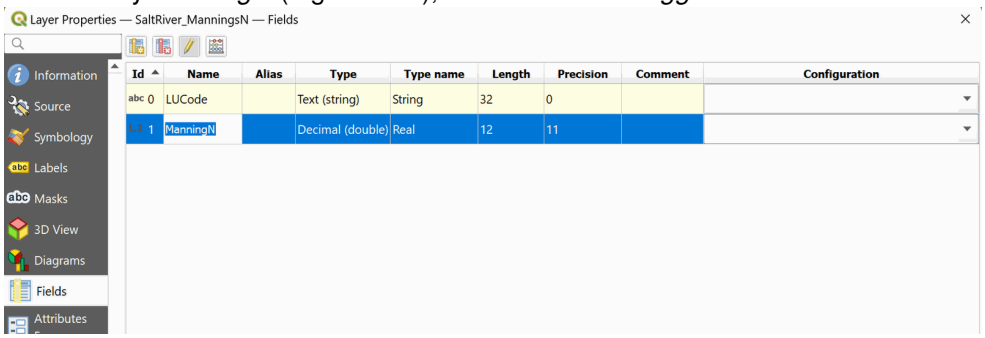
2. Proceed to change the name of the field *N_Value* to *ManningN*. Close the table of attributes and right-click on the layer label. In the pop-up menu, select Properties then in window that opens select the Fields tab as shown in Figure 16.5:



3. Click on the *Toggle Editing* button

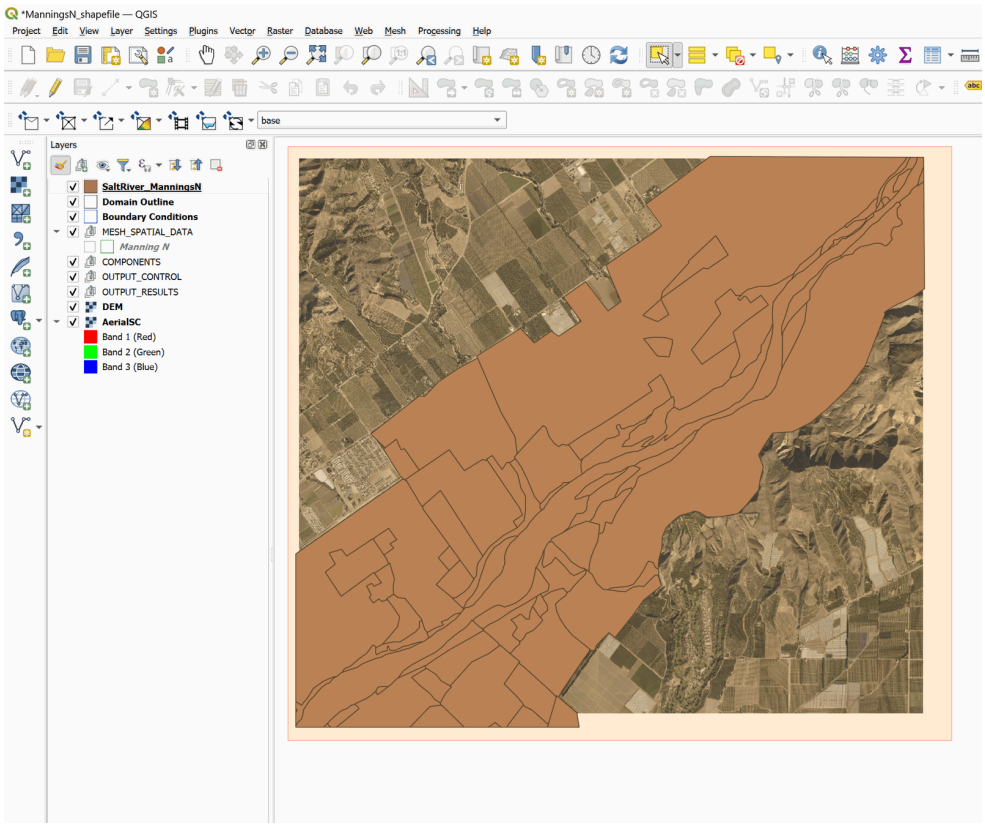


then change the *N_Value* field name by *ManningN* (Figure 16.6), and click on the *Toggle Editing* button again, and save.



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

5. With the select tool  we draw a rectangle that covers the entire layer:



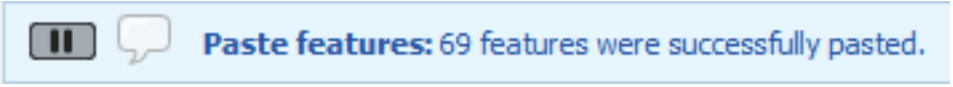
Selecting all the polygons in the SaltRiver_ManningsN layer.

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

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

8. Click on the *Paste Feature* button  .

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

9. Click on the *Toggle Editing* button again  .

   Confirm and save the changes made to the layer.

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

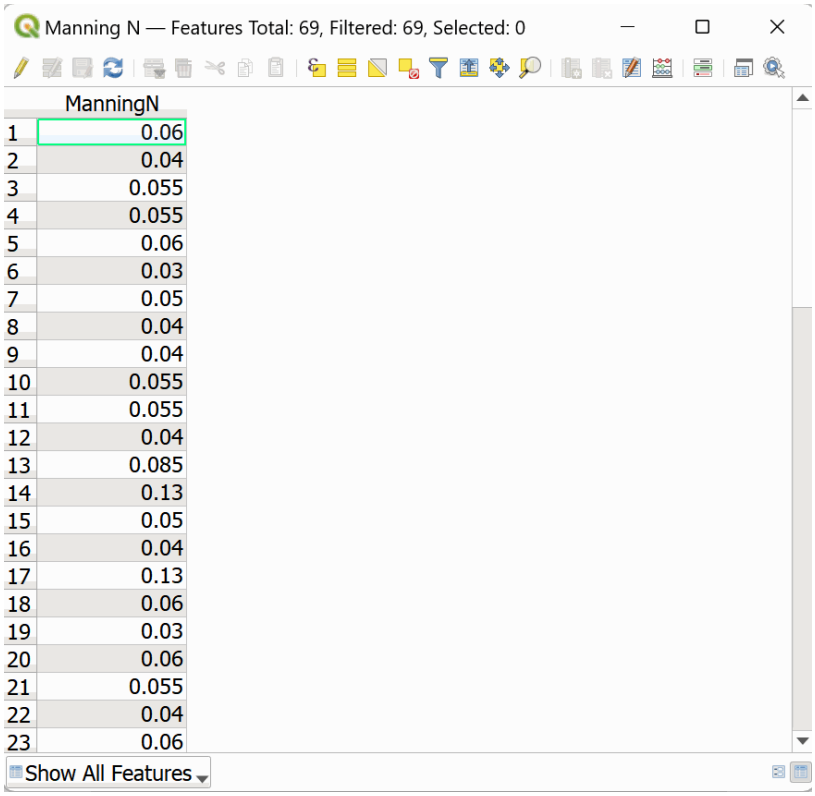


Figure 12.1 – Attribute table of the Manning N layer.

You can now remove the *SaltRiver_ManningsN* layer from the Layer Panel.
This concludes the *Using Manning's n ESRI shape files* tutorial.

13

Post-processing calculations

OilFlow2D has three output controls that make it easier for the user to analyze the results of the runs at specific sites in the domain calculator. These output controls are: Observation points, Cross sections and Profiles.
This tutorial illustrates how to incorporate the output controls in a model using the QGIS interface. The procedure includes the following steps:

1. Open an existing OilFlow2D project.
2. Create *ObservationPoints*, *CrossSections* and *Profiles* layers, and draw the output controls.
3. Generate the mesh.
4. Running the model.
5. Review output files.

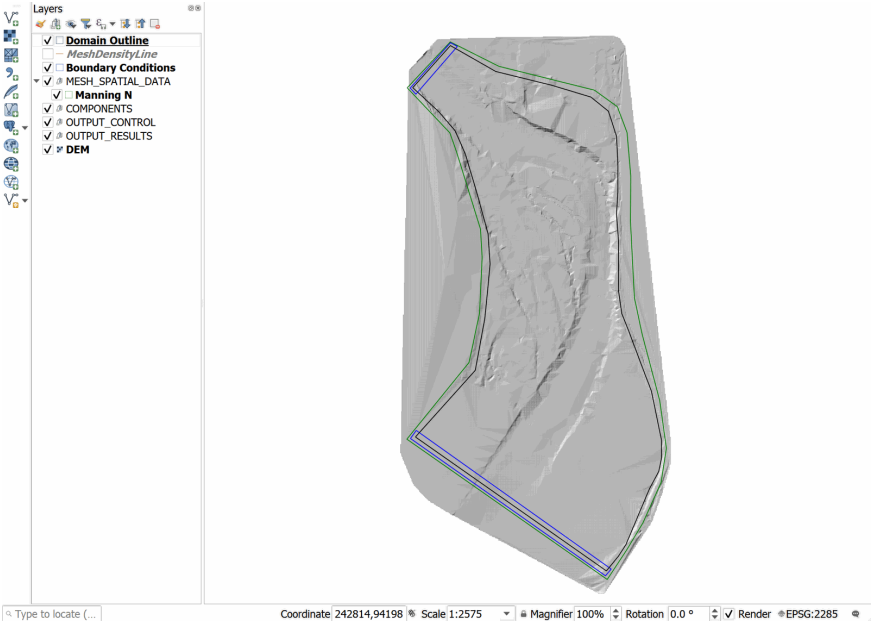
The files required to follow this tutorial can be extracted from the 'ExampleProjects' zip file under the 'OutControlTutorial' folder. This zip file is downloaded separately from your installation materials.

13.1 Open an existing project

1. Open QGIS
2. On the *Project* menu click *Open…* and browse to the existing project: 'OutControlTutorial.qgz'.

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

the project you will have an image of the project loaded in QGIS as shown in Figure 17.1.



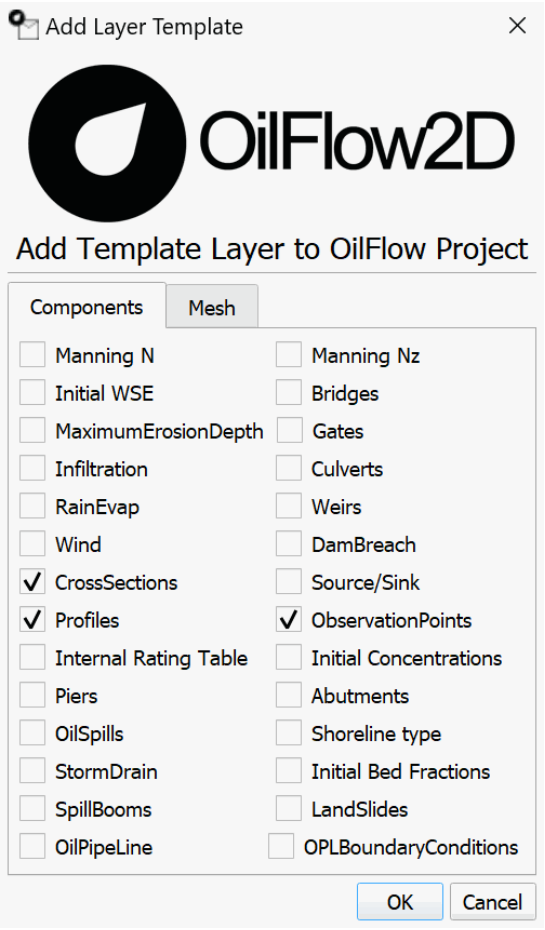
13.2 Create a template of the layers *ObservationPoints*, *CrossSections* and *Profiles* and draw the output controls

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

1. Create the templates of the layers *ObservationPoints*, *CrossSections* and *Profiles*: for this in the OilFlow2D toolbar click on the *New Template Layer* button



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



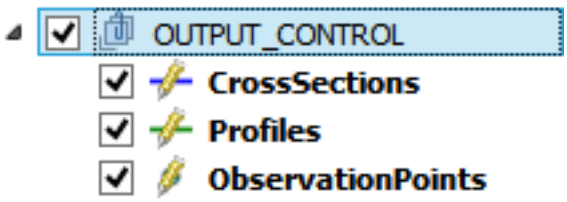
Plugin to add the new templates layer.

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

4. Click on the *Toggle Editing* button:



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



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

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

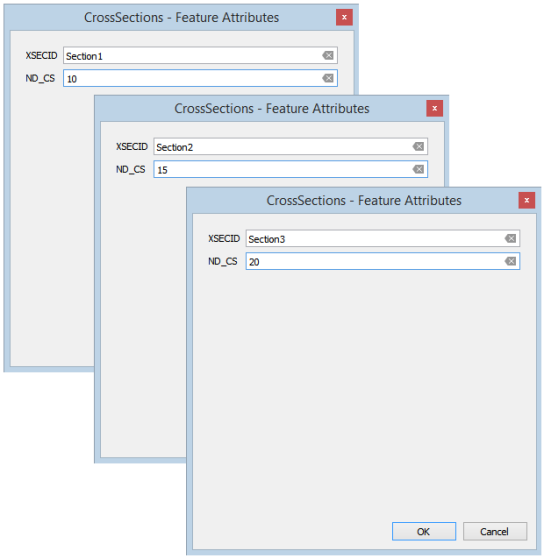


in the case of a point layer like *ObservationPoints*, the icon is

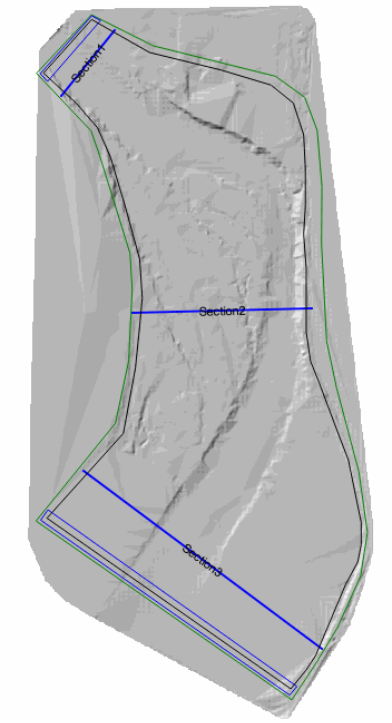


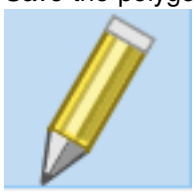
6. Drawing the cross sections: Select the *CrossSection* layer, and activate the *Add Feature* button.

7. Proceed to draw three sections: One at the beginning of the channel, another in the middle and the third almost at the end of the channel, identify (XSECID) as: Section1, Section2 and Section3, with intervals (ND_CS) of 10, 15 and 20 respectively. The attribute tables of the sections will be as shown in Figure 17.3 and at the end of the drawing a similar image should

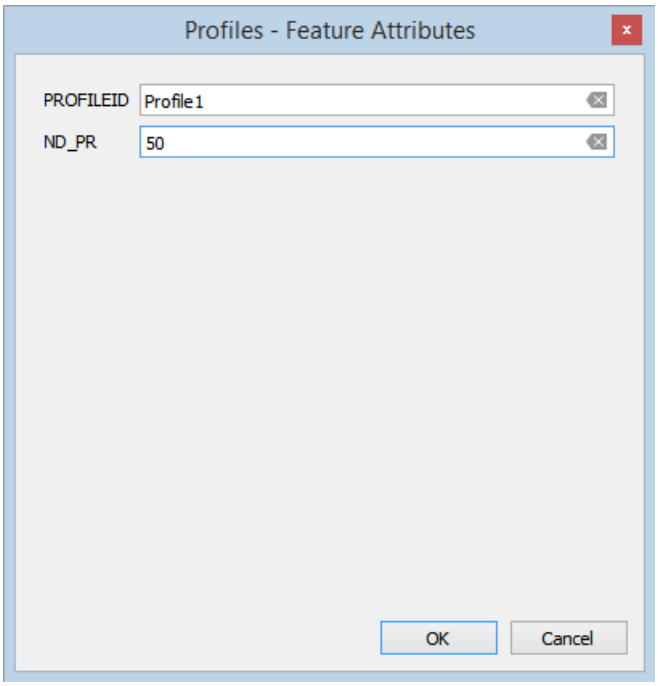


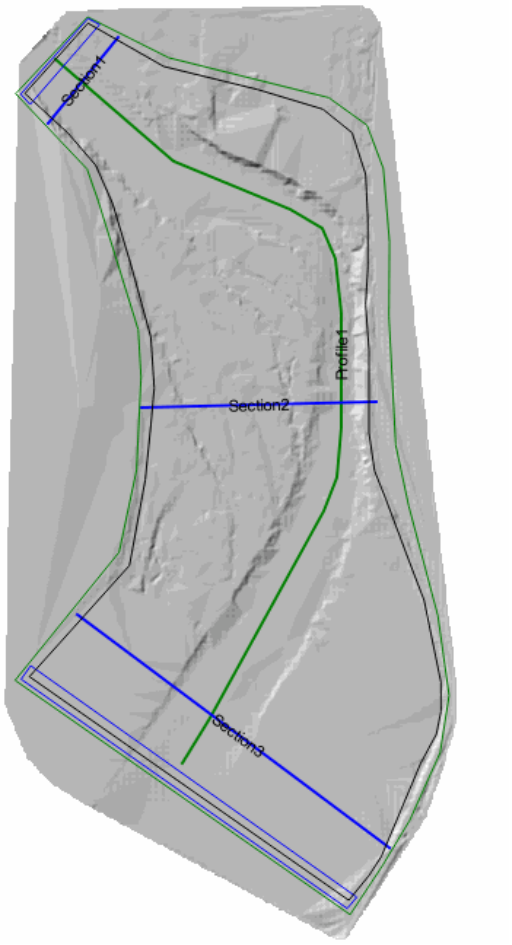
appear as shown in the following Figure 17.4.



8. Save the polygon by clicking the Save button  and click on the *Toggle Editing* button  to deactivate Edit mode on the CrossSections layer.

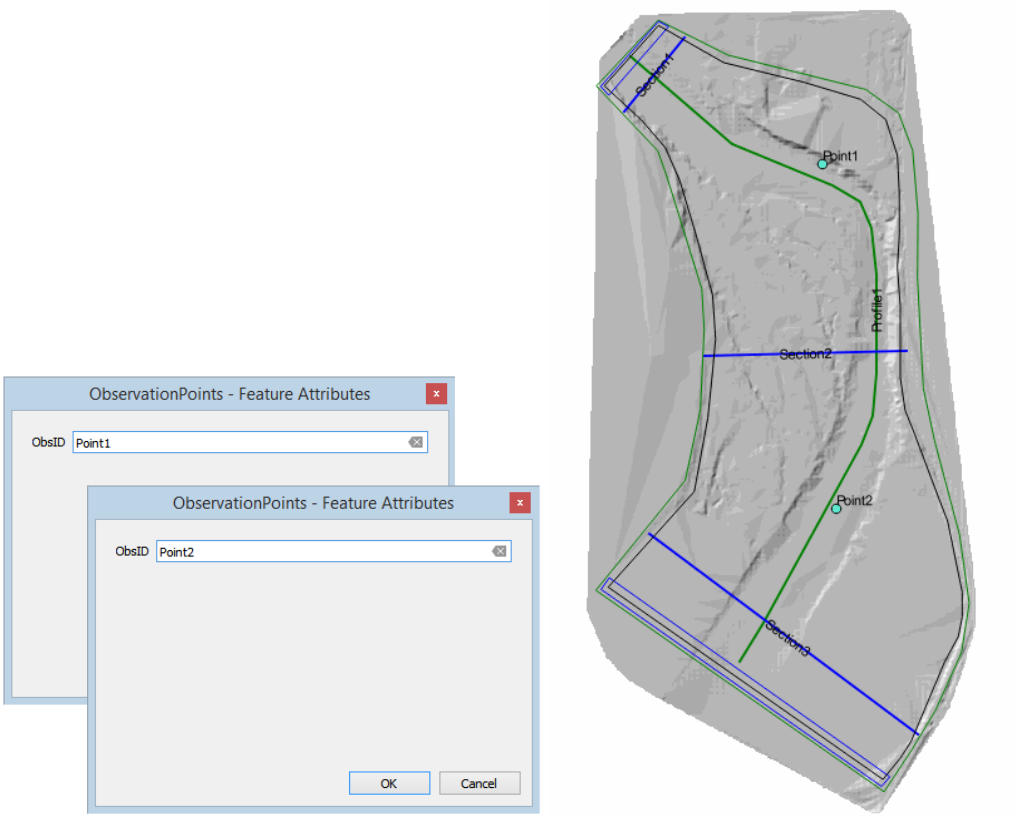
9. Drawing the Profile: Select the Profile layer and activate the Add Feature button, we proceed to draw the profile along the channel central axis, identifier (PROFILEID) is Profile1 and the number of intervals (ND_PR) equal to 50. The attribute table will be as shown in Figure 17.5. Once finished drawing, it should appear like the one shown in the following Figure 17.6.





10. To finalize the profile drawing, save the polygon by clicking the Save button  and click on *Toggle Editing* button  to deactivate Edit mode on the Profile layer.

11. Drawing the observation points: Select the *ObservationPoints* layer, and activate the *Add Feature* button, proceed to draw two observation points, the first between sections 1 and 2 and the second between sections 2 and 3. The identifier (ObsID) is assigned as Point1 and Point2, respectively. The attribute tables will be as shown in Figure 17.7 and at the end of the drawing you should have an image similar to the one shown in the following Figure 17.8.

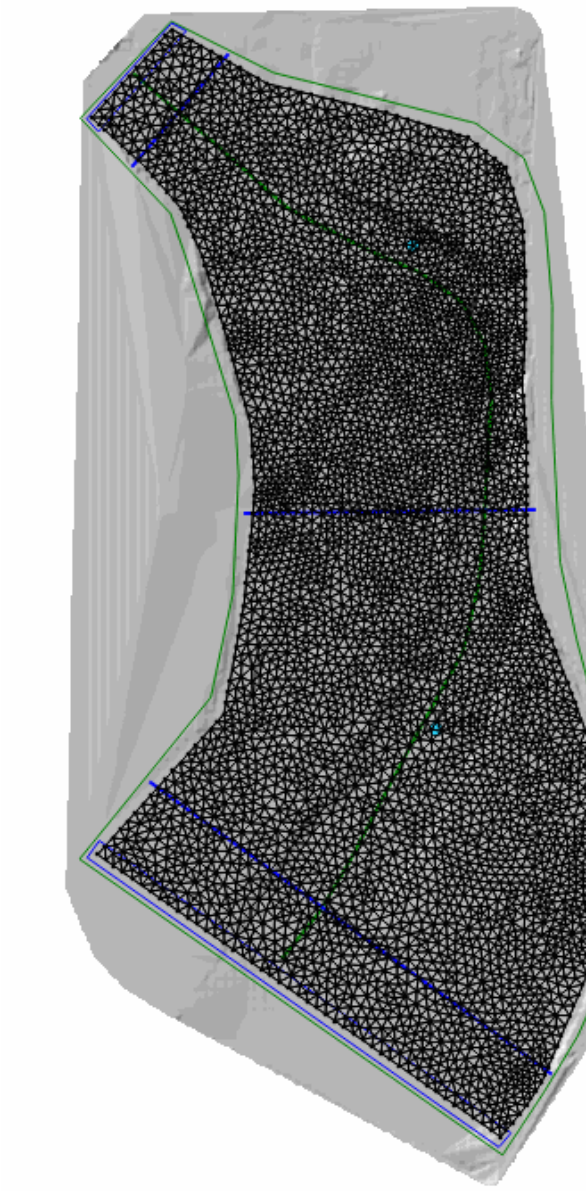


12. To finish the drawing of the observation points, you click again on the *Toggle Editing* button to disable the editing mode of the *ObservationPoints* layer.

13.3 Generate the mesh

The mesh is generated with the *Generate TriMesh* button





The result is a mesh of approximately 9,000 cells, as shown in Figure 17.9.

13.3.1 Exporting files to OilFlow2D

Now that the mesh is generated and the other layers are ready with the necessary data, export the files in the format required by OilFlow2D.

1. Click on the *Export OilFlow2D* button



2. When run the plugin a window is displayed, select the raster layer that contains the Digital Elevation Model (DEM) and the name of the project to be exported.

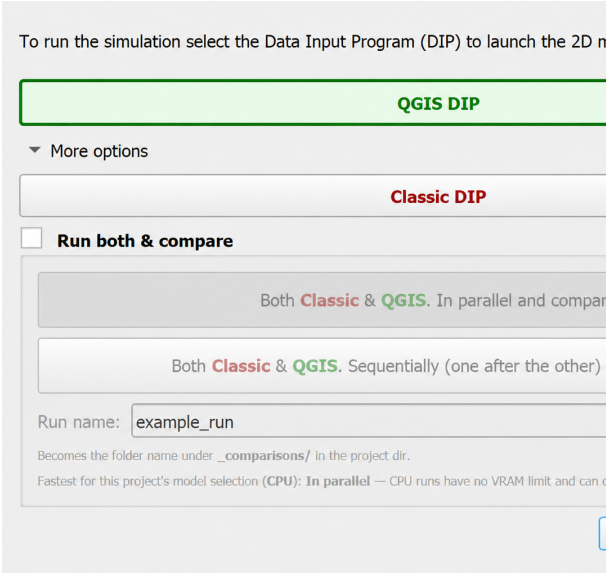
3. Before executing the plugin, activate the layer with the DEM (if it is deactivated).

   Once the plugin is executed, a window will be shown (Figure 17.10), as it should be for our example.

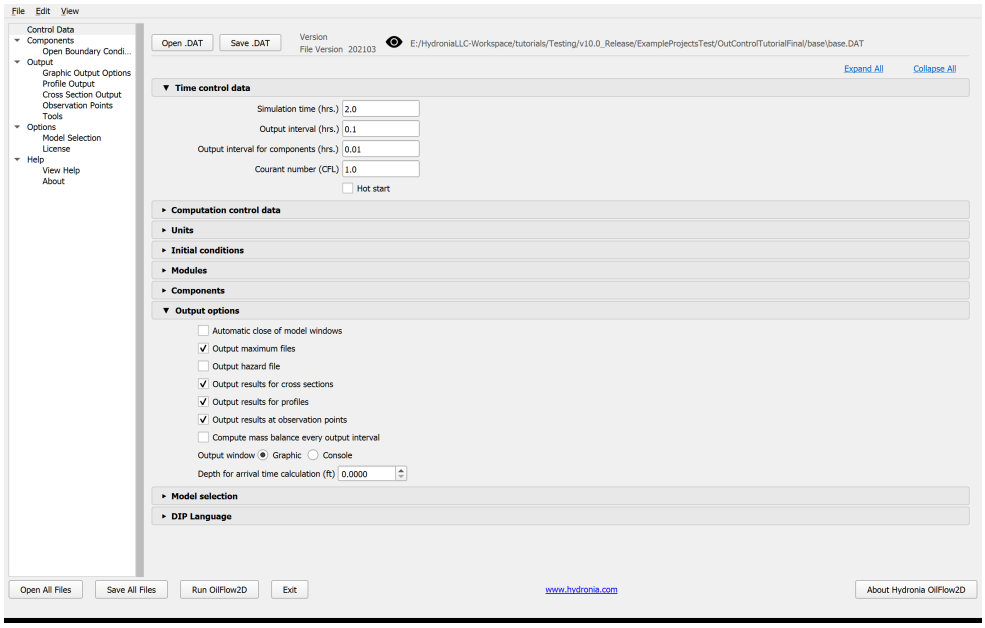


4. Once finished inputting the information, click the [OK] button to export the files to the model.

   After export completes, version 10 and later users see the Open DIP dialog. QGIS DIP is selected by default; click OK to open it inside QGIS. To use the historical standalone option, expand More options, select Classic DIP, and then click OK.



   With QGIS DIP selected, the current interface is shown below:



QGIS DIP Control Data panel for the OutControl tutorial.

QGIS DIP and Classic DIP use the same project settings, but their interfaces look different. Older standalone or non-maintenance versions may open Classic DIP automatically. The project-specific Classic DIP settings and Run steps below are retained.

Once it is finished, OilFlow2D will be loaded with the ‘base.DAT’ file.

13.4 Running the Model

After exporting the files, OilFlow2D opens with the project file of the ‘OutControl.DAT’ sample and shows the *Control Data* panel to it as illustrated in Figure 17.11. The following Classic DIP Control Data panel is retained as a legacy values reference:

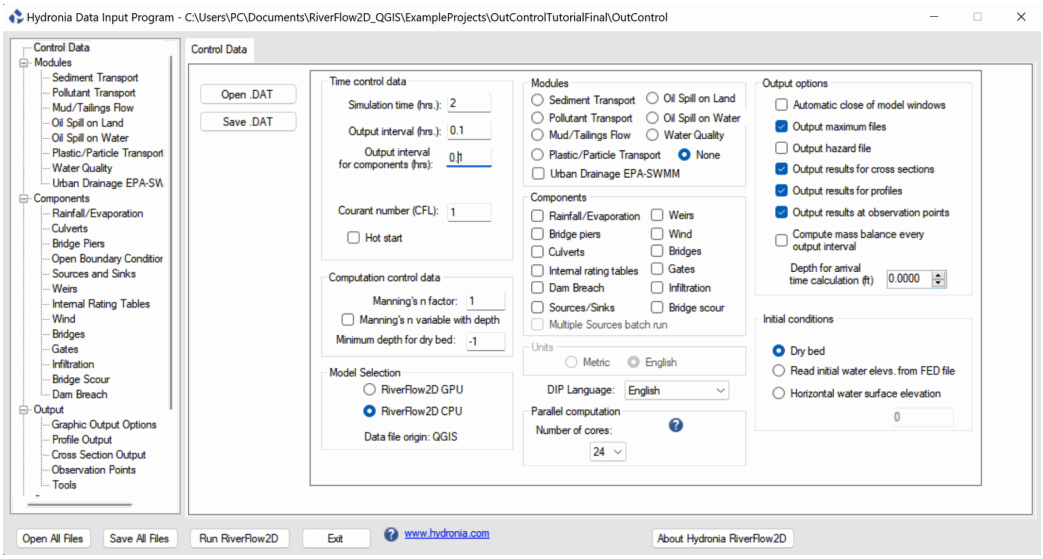
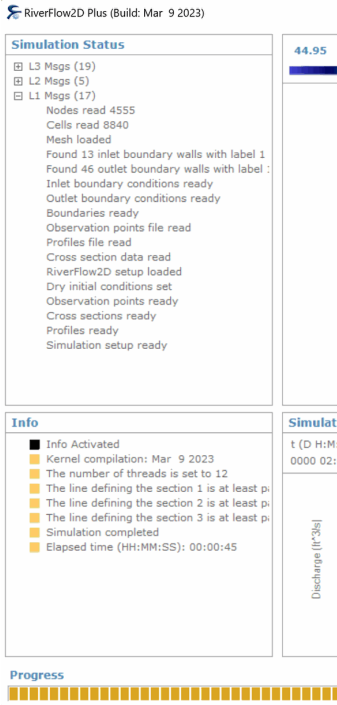


Figure 13.1 – Legacy Classic DIP Control Data panel.

You can observe in the control panel in Output Options the outputs of results for *Cross Sections*, *Profiles* and *Observation Points* are selected.

Leave all other parameters at their default values.

To run the model, click on the *Run OilFlow2D* button. The window that OilFlow2D presents while running the model shows: simulation time information, volume conservation error, total discharge of inflow in and outflow, as well as other parameters as execution progresses (Figure 17.12).



13.5 Review the output files

OilFlow2D generates the files with the extensions '.xseci' (metric units) and '.xsece' (English units) which report the results along the cross sections. it generates files with extensions '.prfi' (metric units) and '.prfe' (English units) which report the results along the profiles and generates the files with the extensions '.outi' (metric units) and '.oute' (English units) that report the results in the observation points.

Figure 17.13 shows an extract of the 'OutControl.xsece' file with results at the cross sections:

```
================================================================================
================================================================================
                          RiverFlow2D
                          Build Mar  9 2023
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                       (R) TRADEMARK 2009-2022 Hydronia, LLC.
                                   ALL RIGHTS RESERVED
                                   RUN DATE: 27/Mar/2023
================================================================================

 CROSS SECTION RESULTS IN ENGLISH UNITS

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


 CROSS SECTION NO.:   1 CROSS SECTION ID: XSEC_1

(     243444.89,       94253.80), (     243571.18,       94402.64)
ELEM    STATION    BEDEL     DEPTH     WSEL     VELOCITY  FROUDE     QS
          (ft)      (ft)      (ft)      (ft)      (ft/s)           (ft2/s)
 6351     13.80     50.16      0.00     50.16      0.00      0.00  0.000000
  587     23.66     49.49      0.00     49.49      0.00      0.00  0.000000
  359     35.49     49.52      0.00     49.52      0.00      0.00  0.000000
 2325     41.41     49.33      0.00     49.33      0.00      0.00  0.000000
 1551     57.18     49.22      0.00     49.22      0.00      0.00  0.000000
  293     67.04     49.10      0.00     49.10      0.00      0.00  0.000000
 6321     72.95     48.87      0.00     48.87      0.00      0.00  0.000000
 4106     80.84     48.82      0.00     48.82      0.00      0.00  0.000000
 1836     90.70     48.70      0.00     48.70      0.00      0.00  0.000000
 1597     94.64     49.04      0.00     49.04      0.00      0.00  0.000000
 3138    102.53     49.23      0.00     49.23      0.00      0.00  0.000000
  809    106.47     49.85      0.00     49.85      0.00      0.00  0.000000
   95    110.42     50.70      0.00     50.70      0.00      0.00  0.000000
 4592    124.22     51.52      0.00     51.52      0.00      0.00  0.000000
 2008    130.13     51.58      0.00     51.58      0.00      0.00  0.000000
 6910    139.99     51.70      0.00     51.70      0.00      0.00  0.000000
 1814    143.94     51.98      0.00     51.98      0.00      0.00  0.000000
 1411    149.85     52.02      0.00     52.02      0.00      0.00  0.000000
 2062    159.71     52.13      0.00     52.13      0.00      0.00  0.000000
  356    165.62     52.07      0.00     52.07      0.00      0.00  0.000000
 3167    175.48     52.21      0.00     52.21      0.00      0.00  0.000000
 6739    185.34     52.39      0.00     52.39      0.00      0.00  0.000000
Q =     -0.000 cfs.


 CROSS SECTION NO.:   2 CROSS SECTION ID: XSEC_2

(     243995.16,       93802.76), (     243602.76,       93800.51)
ELEM    STATION    BEDEL     DEPTH     WSEL     VELOCITY  FROUDE     QS
          (ft)      (ft)      (ft)      (ft)      (ft/s)           (ft2/s)
 7097     19.82     54.90      0.00     54.90      0.00      0.00  0.000000
 6346     23.78     54.22      0.00     54.22      0.00      0.00  0.000000
 6157     27.75     51.32      0.00     51.32      0.00      0.00  0.000000
  121     35.67     48.75      0.00     48.75      0.00      0.00  0.000000
  184     39.64     46.82      0.00     46.82      0.00      0.00  0.000000
 2632     47.56     46.00      0.00     46.00      0.00      0.00  0.000000
 2968     51.53     45.98      0.00     45.98      0.00      0.00  0.000000
 4400     55.49     46.02      0.00     46.02      0.00      0.00  0.000000
  682     59.46     46.51      0.00     46.51      0.00      0.00  0.000000
```

Figure 17.14 shows an extract of the ‘OutControl.prfe’ file with the report of the profile results:

```
================================================================================
================================================================================
                          RiverFlow2D
                          Build Mar  9 2023
================================================================================
             TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                       (R) TRADEMARK 2009-2022 Hydronia, LLC.
                                   ALL RIGHTS RESERVED
                                   RUN DATE: 27/Mar/2023
================================================================================

 PROFILE RESULTS IN ENGLISH UNITS

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

 PROFILE NO.:   1 PROFILE ID: PROFILE_1

 ELEM   DISTANCE    BEDEL     DEPTH      WSEL    VELOCITY  FROUDE
          (ft)      (ft)       (ft)      (ft)     (ft/s)
  6406      0.00     48.55     15.61     64.17      0.00      0.00
  4446     15.34     48.69      0.00     48.69      0.00      0.00
  3736     30.68     48.61      0.00     48.61      0.00      0.00
  1927     46.02     48.56      0.00     48.56      0.00      0.00
  2357     61.36     48.76      0.00     48.76      0.00      0.00
  1836     76.69     48.70      0.00     48.70      0.00      0.00
  8006     92.03     48.82      0.00     48.82      0.00      0.00
  6142    107.37     48.81      0.00     48.81      0.00      0.00
    80    122.71     48.78      0.00     48.78      0.00      0.00
  1878    138.05     48.69      0.00     48.69      0.00      0.00
  1622    153.39     48.63      0.00     48.63      0.00      0.00
   978    168.73     48.65      0.00     48.65      0.00      0.00
  6342    184.07     48.71      0.00     48.71      0.00      0.00
  5985    199.41     48.76      0.00     48.76      0.00      0.00
   843    214.74     48.78      0.00     48.78      0.00      0.00
  6360    230.08     48.89      0.00     48.89      0.00      0.00
   575    245.42     48.92      0.00     48.92      0.00      0.00
  4957    260.76     49.01      0.00     49.01      0.00      0.00
  4201    276.10     49.02      0.00     49.02      0.00      0.00
  2430    291.76     48.84      0.00     48.84      0.00      0.00
   734    307.41     48.79      0.00     48.79      0.00      0.00
  3495    323.07     48.91      0.00     48.91      0.00      0.00
  6854    338.72     49.05      0.00     49.05      0.00      0.00
  6256    354.38     49.03      0.00     49.03      0.00      0.00
  3401    370.03     48.65      0.00     48.65      0.00      0.00
  3600    385.69     48.45      0.00     48.45      0.00      0.00
  5619    401.34     48.52      0.00     48.52      0.00      0.00
  3586    417.00     48.77      0.00     48.77      0.00      0.00
   822    432.66     48.39      0.00     48.39      0.00      0.00
  4197    448.31     48.17      0.00     48.17      0.00      0.00
  5100    463.97     48.00      0.00     48.00      0.00      0.00
   194    479.62     47.93      0.00     47.93      0.00      0.00
  3785    495.28     47.54      0.00     47.54      0.00      0.00
  4267    510.93     46.81      0.00     46.81      0.00      0.00
  7358    526.59     46.52      0.00     46.52      0.00      0.00
  2123    540.71     46.84      0.00     46.84      0.00      0.00
  8723    554.84     46.76      0.00     46.76      0.00      0.00
  4474    568.96     47.17      0.00     47.17      0.00      0.00
  5572    583.08     45.98      0.00     45.98      0.00      0.00
  6970    597.21     46.20      0.00     46.20      0.00      0.00
  2493    611.33     46.19      0.00     46.19      0.00      0.00
```

Figure 17.15 shows an extract of the 'RESvsT_Point1.oute' file with the report of the results of the ob-

```
================================================================================
================================================================================
                          RiverFlow2D
                          Build Mar  9 2023
================================================================================
            TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                      (R) TRADEMARK 2009-2022 Hydronia, LLC.
                                  ALL RIGHTS RESERVED
                                  RUN DATE: 27/Mar/2023
================================================================================

 RESULTS FOR CELL:      840 OBSERVATION POINT ID: Point2
   LOCATED AT COORDINATE: (   243832.78),(    93473.50)

      TIME         U         V     VELOCITY    DEPTH      WSEL    BEDEL_ORI   BEDEL    DELTA_BED  FROUDE     QSX         QSY         QS
     (hours)    (ft/s)    (ft/s)    (ft/s)      (ft)      (ft)       (ft)      (ft)       (ft)      (-)     (ft2/s)     (ft2/s)     (ft2/s)
     0.10000    -2.897    -7.137     7.703     4.806    52.126     47.319    47.319     0.000     0.619     0.000      0.000      0.000
     0.20000    -2.736    -6.929     7.450     5.019    52.338     47.319    47.319     0.000     0.586     0.000      0.000      0.000
     0.30000    -2.720    -6.901     7.418     5.035    52.355     47.319    47.319     0.000     0.583     0.000      0.000      0.000
     0.40000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
     0.50000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
     0.60000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
     0.70000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
     0.80000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
     0.90000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
     1.00000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
     1.10000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
     1.20000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
     1.30000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
     1.40000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
     1.50000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
     1.60000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
     1.70000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
     1.80000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
     1.90000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
     2.00000    -2.718    -6.898     7.414     5.038    52.357     47.319    47.319     0.000     0.582     0.000      0.000      0.000
```

servation point Point1:

14

Advanced digitization/snapping

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

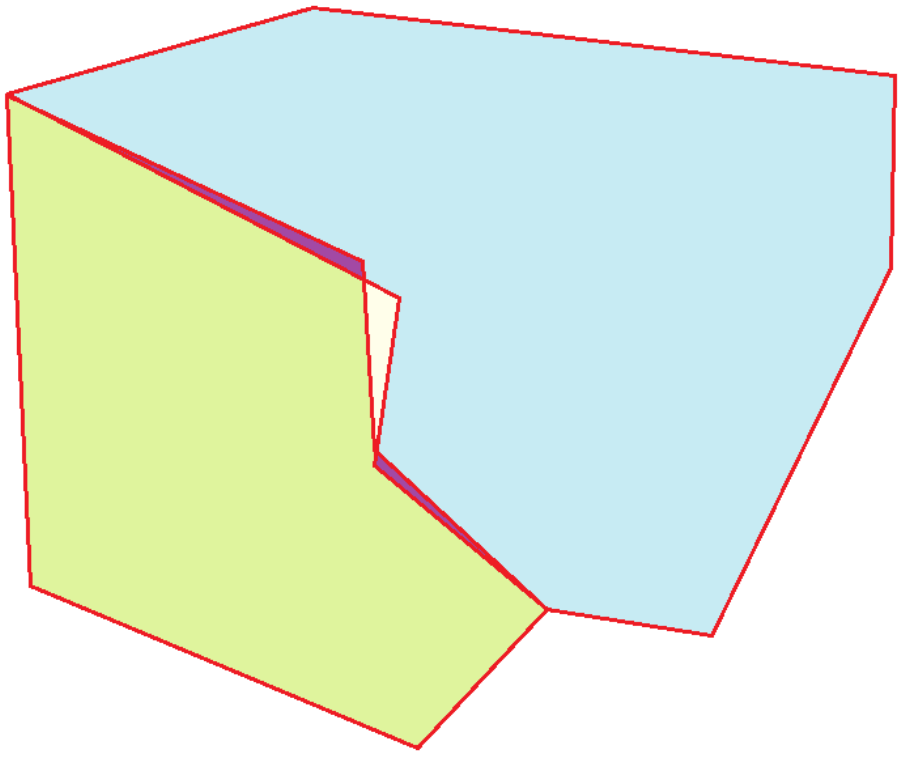


Figure 14.1 – Areas with overlap and empty spaces in manual digitization of adjacent polygons.

This tutorial illustrates how to use the QGIS Snapping tool to facilitate the scanning of adjacent

polygons. The procedure includes the following steps:

1. Open an existing OilFlow2D project.
2. Activate the *Snapping* tool.
3. Configure the *Snapping* tool
4. Draw contiguous or adjacent polygons using the *Snapping* tool.

The files required to follow this tutorial can be extracted from the 'ExampleProjects' zip file under the 'SnappingTutorial' folder. This zip file is downloaded separately from your installation materials.

14.1 Open an existing project

1. Open QGIS
2. In the main menu go to *Project* > *Open…* and browse to the existing project: 'SnappingTutorial.qgz'.

This project contains the basic templates of a OilFlow2D project and a raster with 4 areas with different Manning's n coefficients derived from land cover and land use maps. From this raster, the polygons of the *Manning N* layer will be drawn. When the project is opened, a project image will be loaded in QGIS as shown in Figure 18.2.

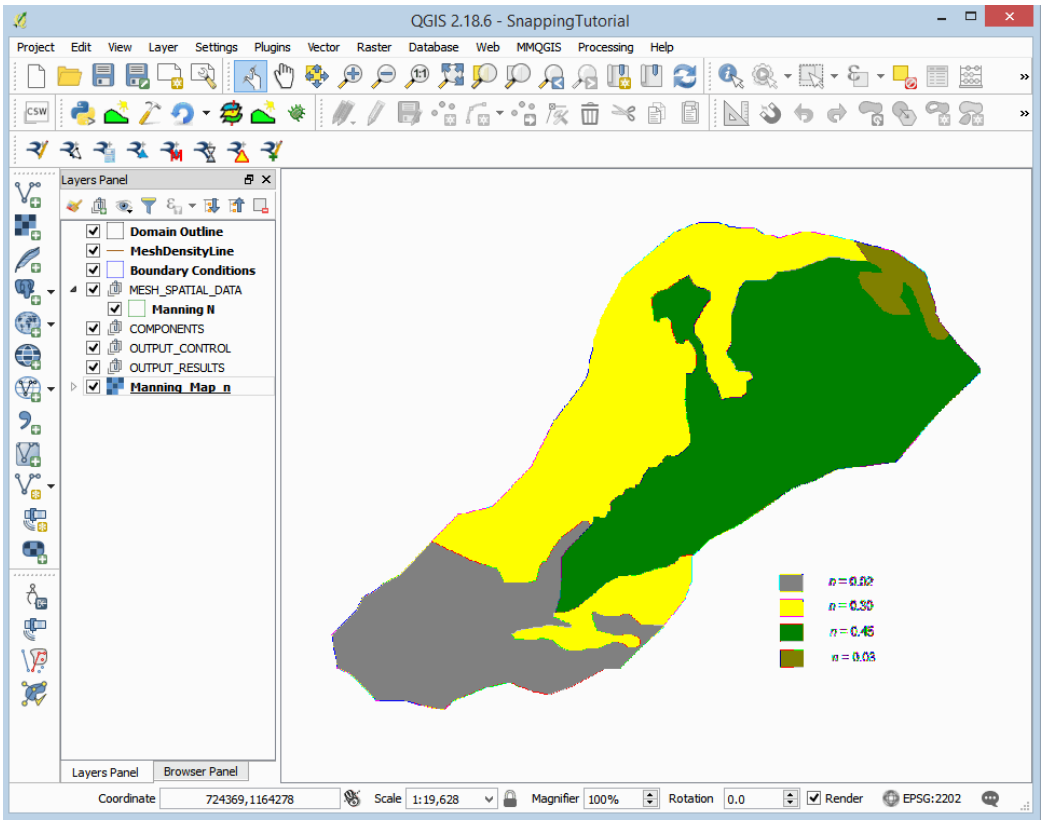
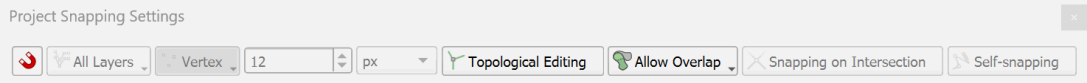


Figure 14.2 – Project screen loaded in QGIS.

14.2 Activate and configure the Snapping tool

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

1. Open the configuration options window: In the Project menu, select *Snapping Options…*



Project Snapping Settings window.

2. In the *Project Snapping Settings* window, click on the *Enable Snapping* button.



3. Next to the *Enable Snapping* button the dropdown menu for *All Layers* is activated. Click on the dropdown and select *Advanced Configuration*.
4. In the list of layers, make sure only *Manning N* layer is selected.
5. Select the checkbox to *Avoid Overlap* on the *Manning N* layer.

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

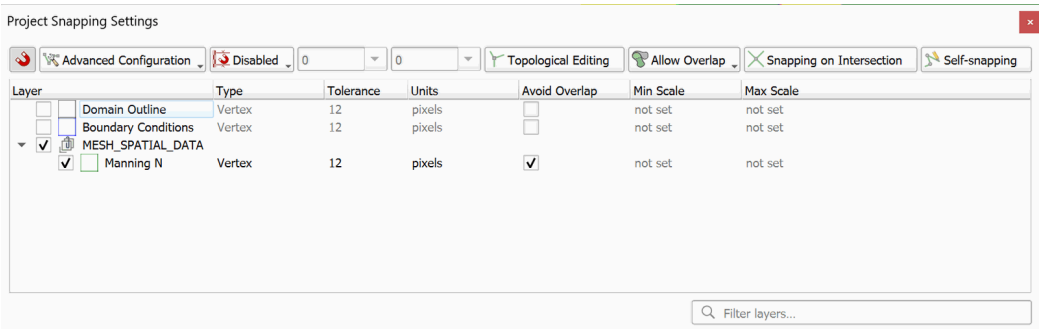


Figure 14.3 – Snapping configuration panel.

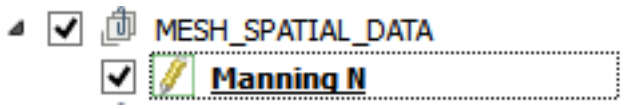
14.3 Draw contiguous or adjacent polygons using the Snapping tool

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

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



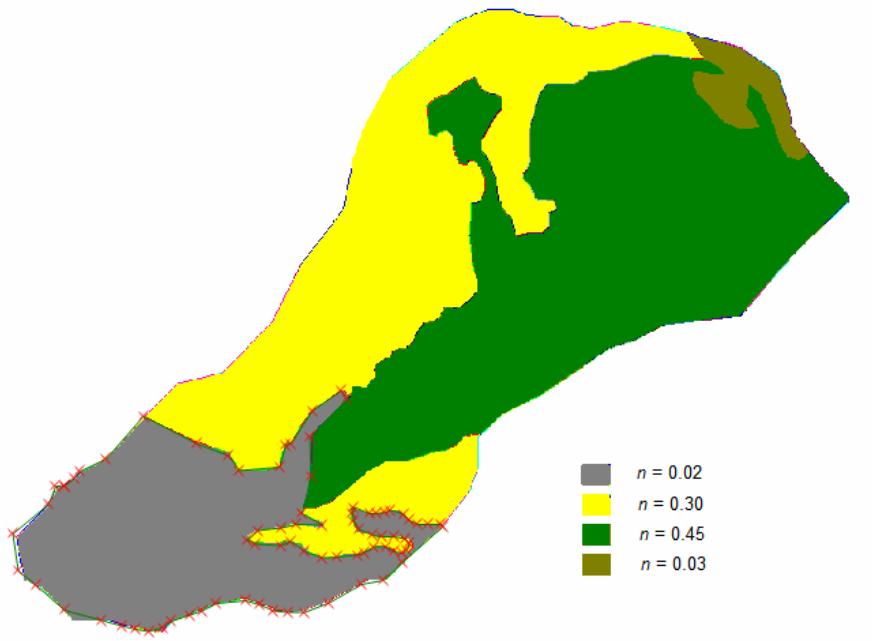
A pencil icon will appear in the *Manning N* layer indicating that the layer is in edit mode:



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

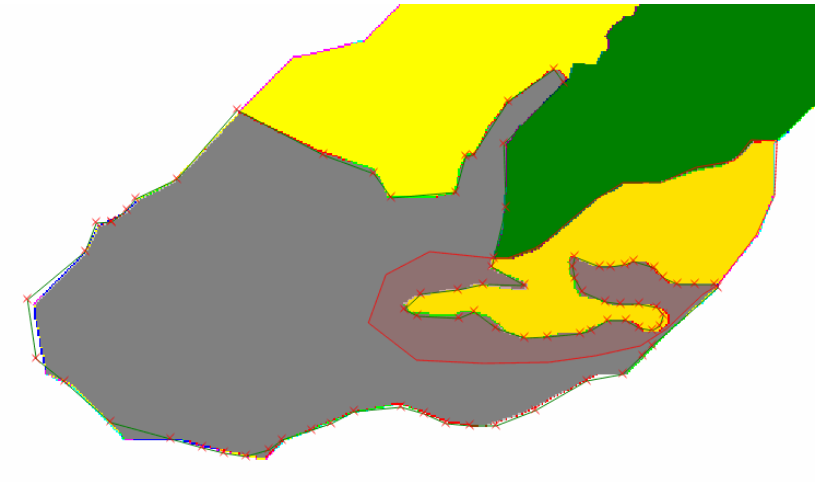


proceed to draw the polygons. Start by drawing the polygon in the bottom of the watershed shown in gray which corresponds to n = 0.02. After completing the drawing you should have an image similar to the one shown in the following figure:



The first polygon sketch.

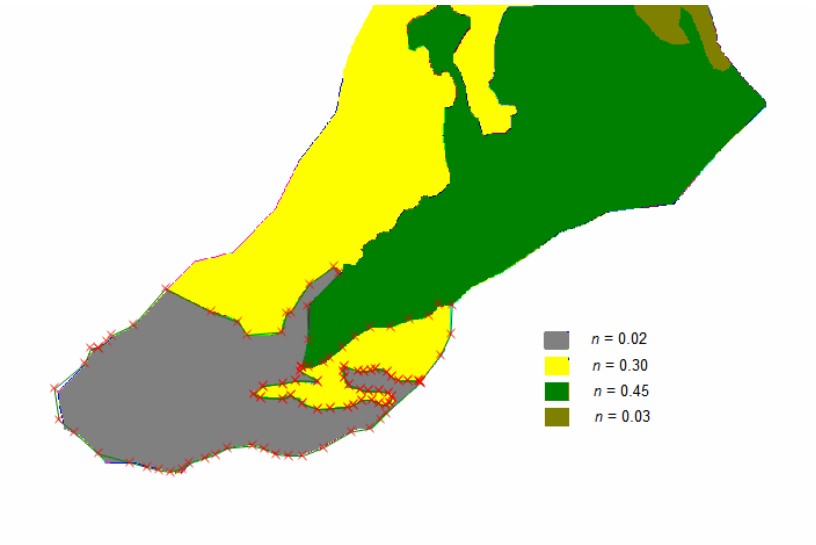
4. Proceed to draw the second polygon. This will be the small yellow polygon to which corresponds to n = 0.30. To draw the polygon, follow the boundary with the green polygon as indicated by the raster, but in the boundary with the polygon already drawn (the gray) an overlap will be made so that the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



Drawn of the second polygon.

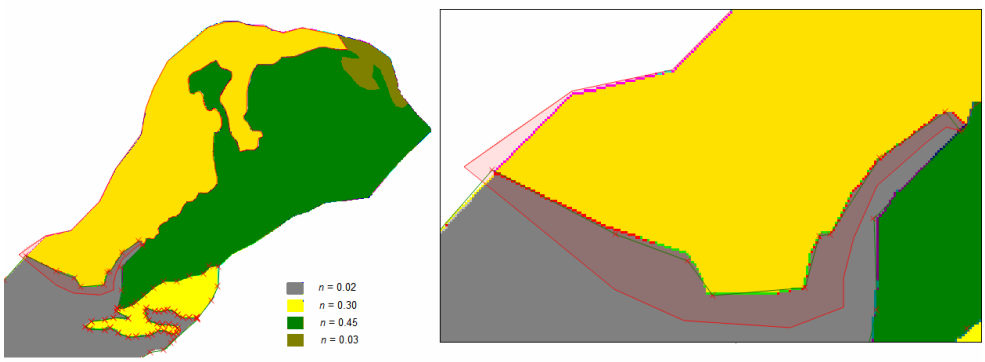
5. Right-click to finish the drawing.

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



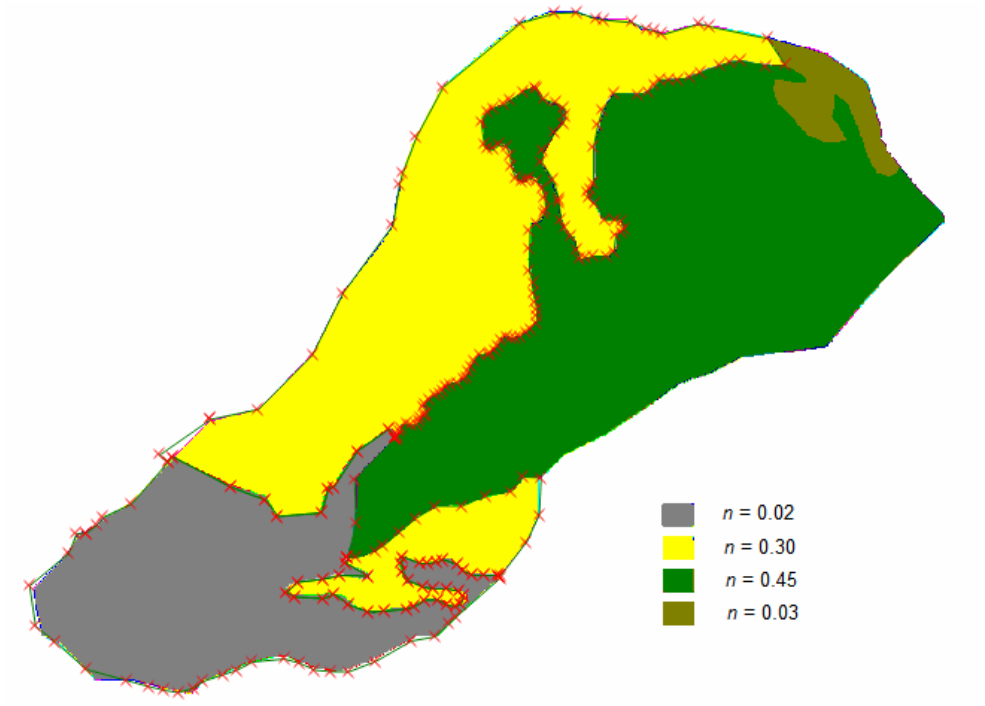
Final draw of the second polygon.

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



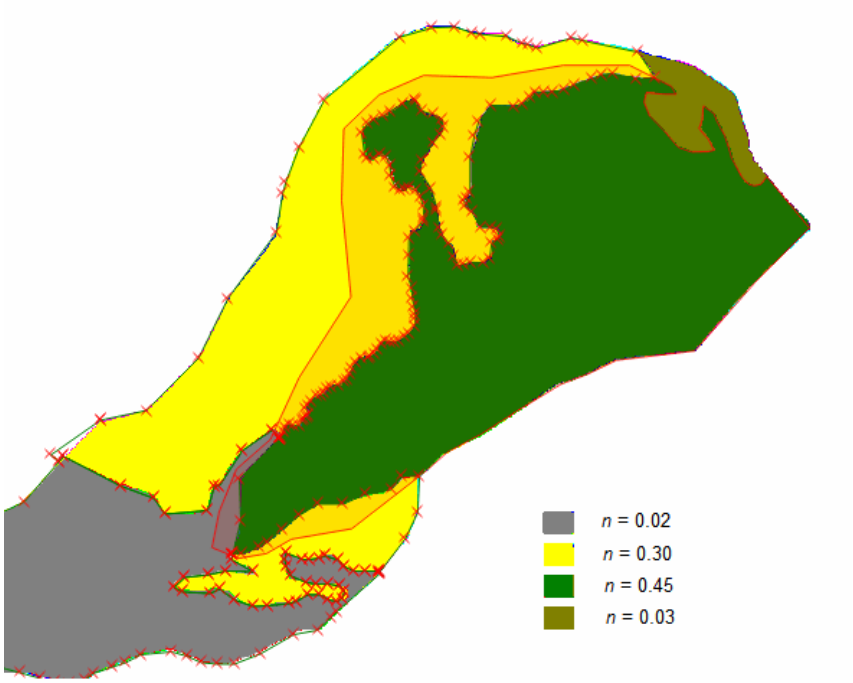
Preliminary draw of the third polygon.

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



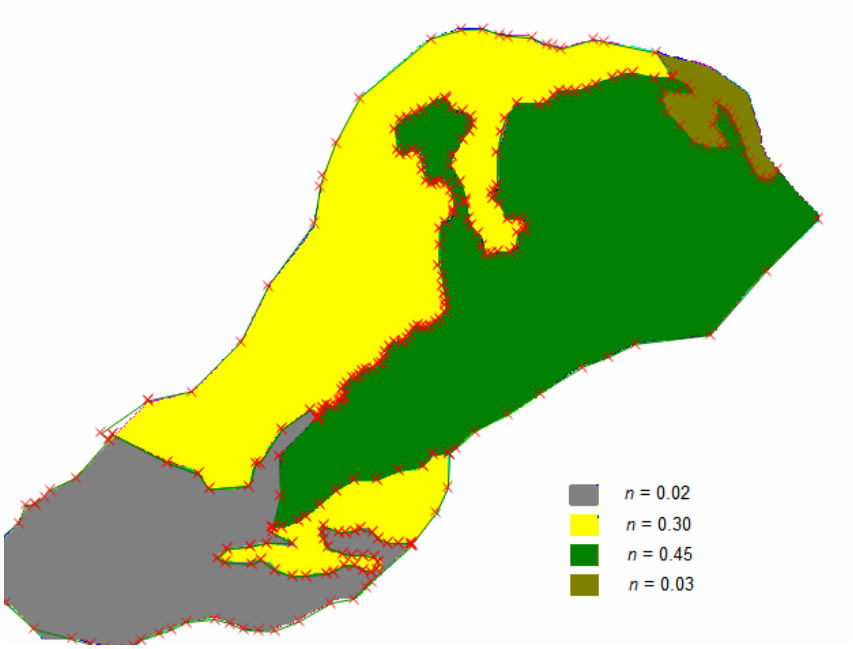
Final draw of the third polygon.

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



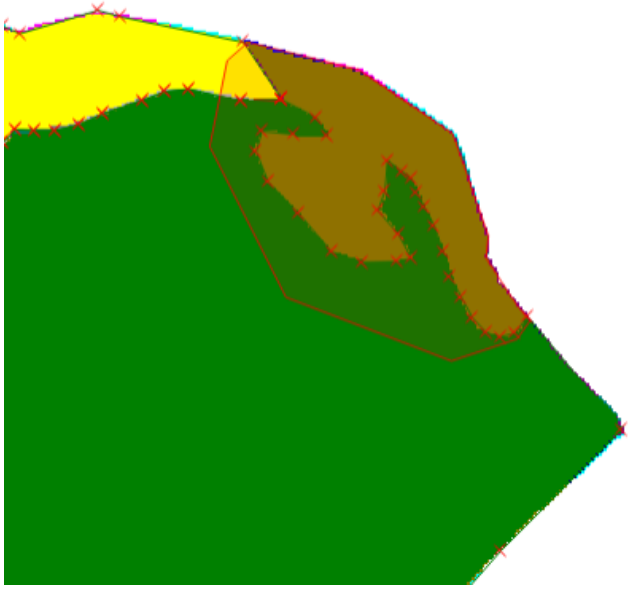
Preliminary draw of the fourth polygon.

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



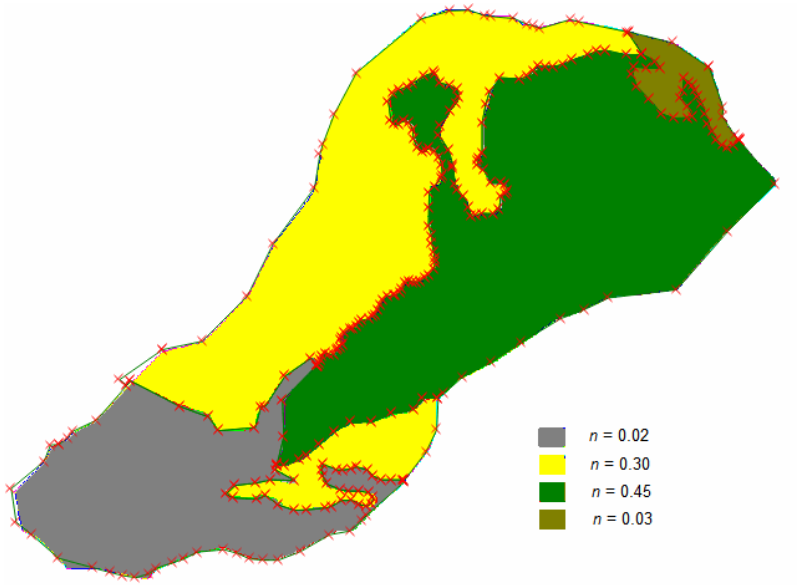
Final draw of the fourth polygon.

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



Preliminary draw of the fifth polygon.

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

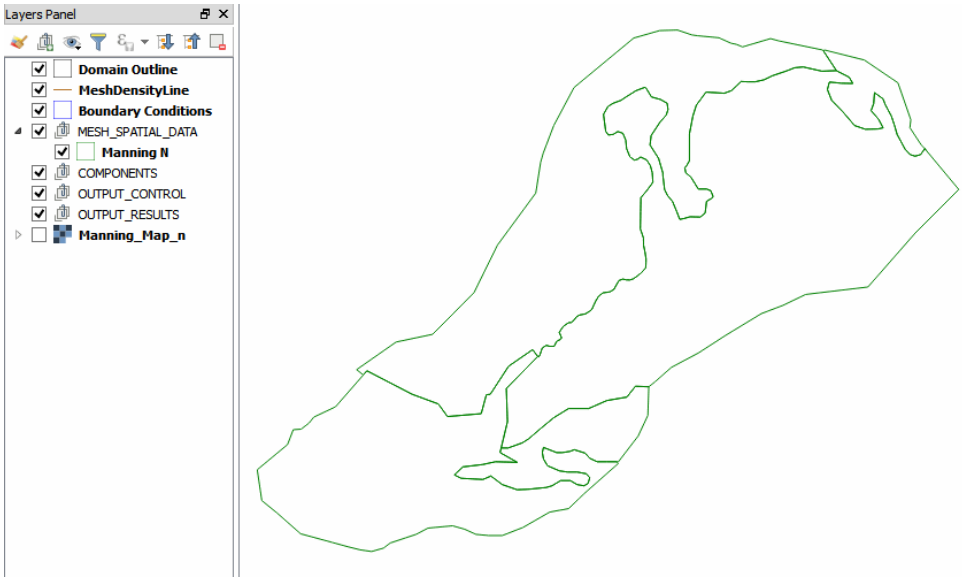


Final draw of the fifth polygon.

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



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



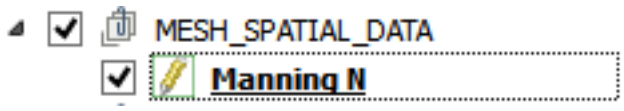
Final draw of the Manning N layer.

14.4 Delete a polygon

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

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

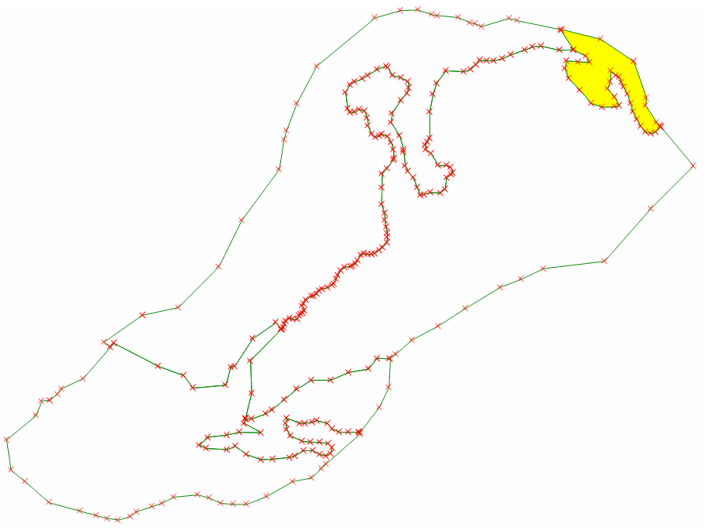




2. Then the polygon to be eliminated is selected using the *Select Feature* button



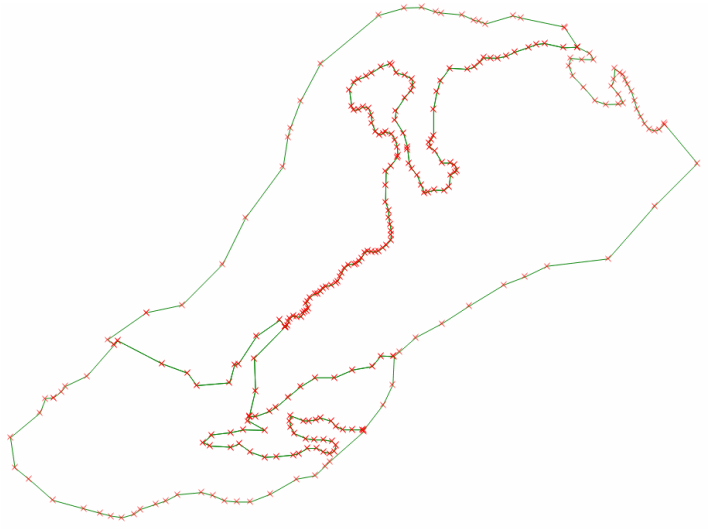
3. With the active select Feature tool, click on the polygon to be deleted and it will be highlighted in yellow to indicate that it is selected (Figure 18.15).



Polygon to be deleted selected.

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





Manning N layer with the polygon removed.

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



This concludes the Snapping tutorial.

15

Creating raster elevations from X Y Z data sets

The files required to follow this tutorial can be extracted from the 'ExampleProjects' zip file under the 'InterpolatingRasterTutorial' folder. This zip file is downloaded separately from your installation materials.

OilFlow2D uses elevation data in raster format. For instance, to load an ASCII grid file, from the *Layer* menu, click *Add Layer*, and then click *Add Raster Layer…* You may also click the *Add Raster Layer* button:



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

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

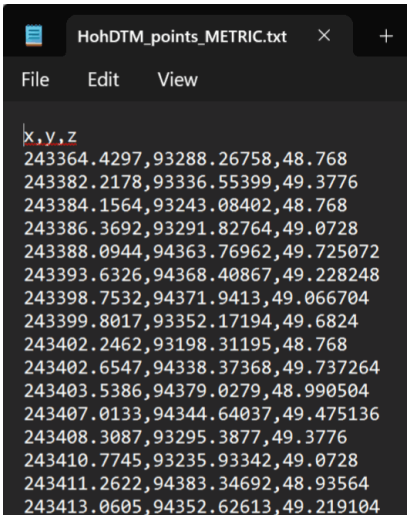


Figure 15.1 – File containing terrain elevation points.

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

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

2. Open the elevations file: Click the Browse […] button and go to

   '\ExampleProjects\InterpolatingRasterTutorial\base\HohDTM_points_xyz.txt'.

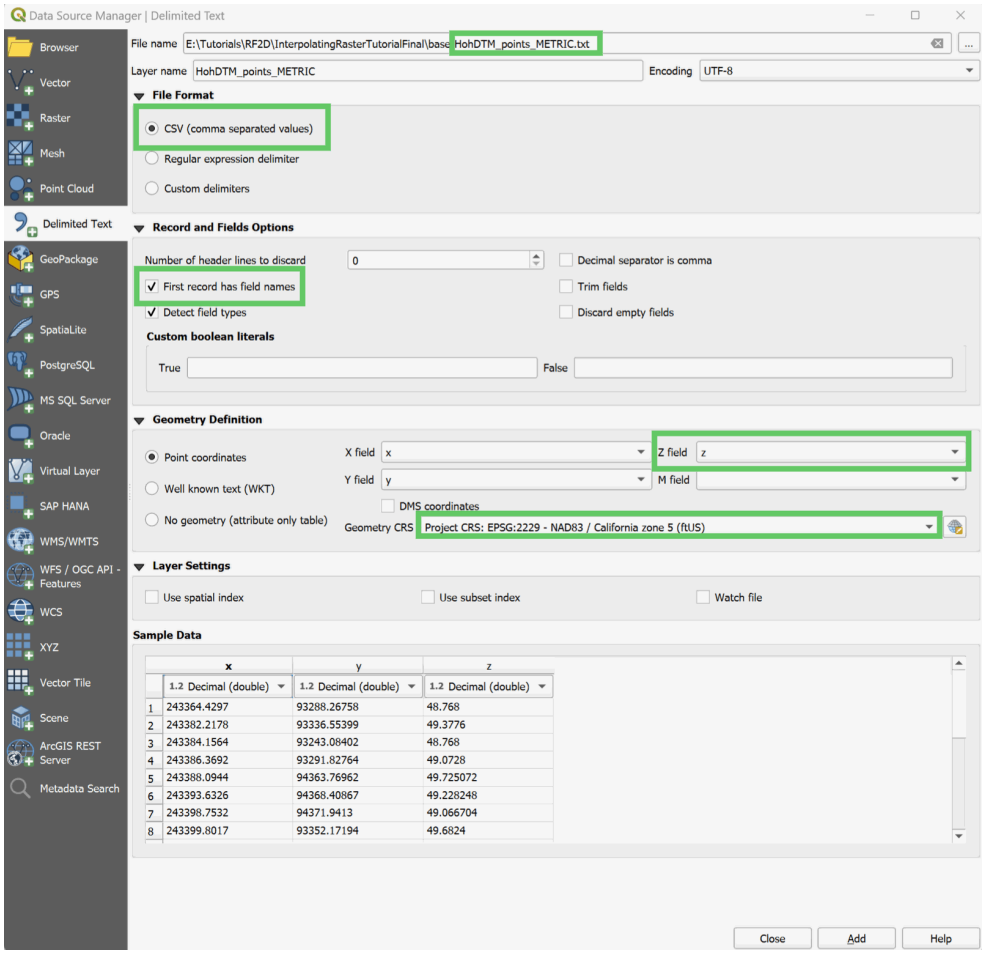
3. Under *File format*, *Comma Separated Values* should be selected.

4. Under *Record and Fields Options* the checkboxes *First row contains field names* should be selected.

5. Under *Geometry Definition*, click on the *Z field* dropdown and select *z*.

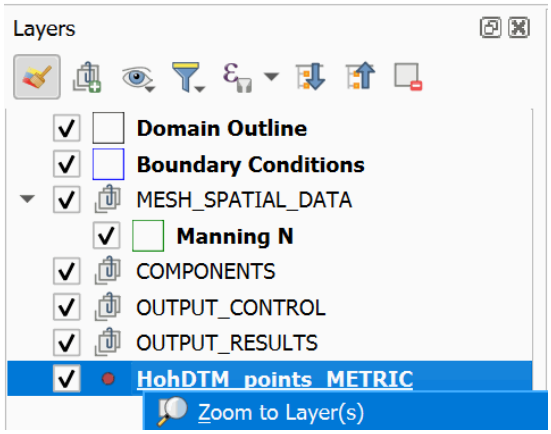
6. The *Geometry CRS* dropdown should be set to *Project CRS: EPSG:2229*.

   The dialog should look like the following figure:



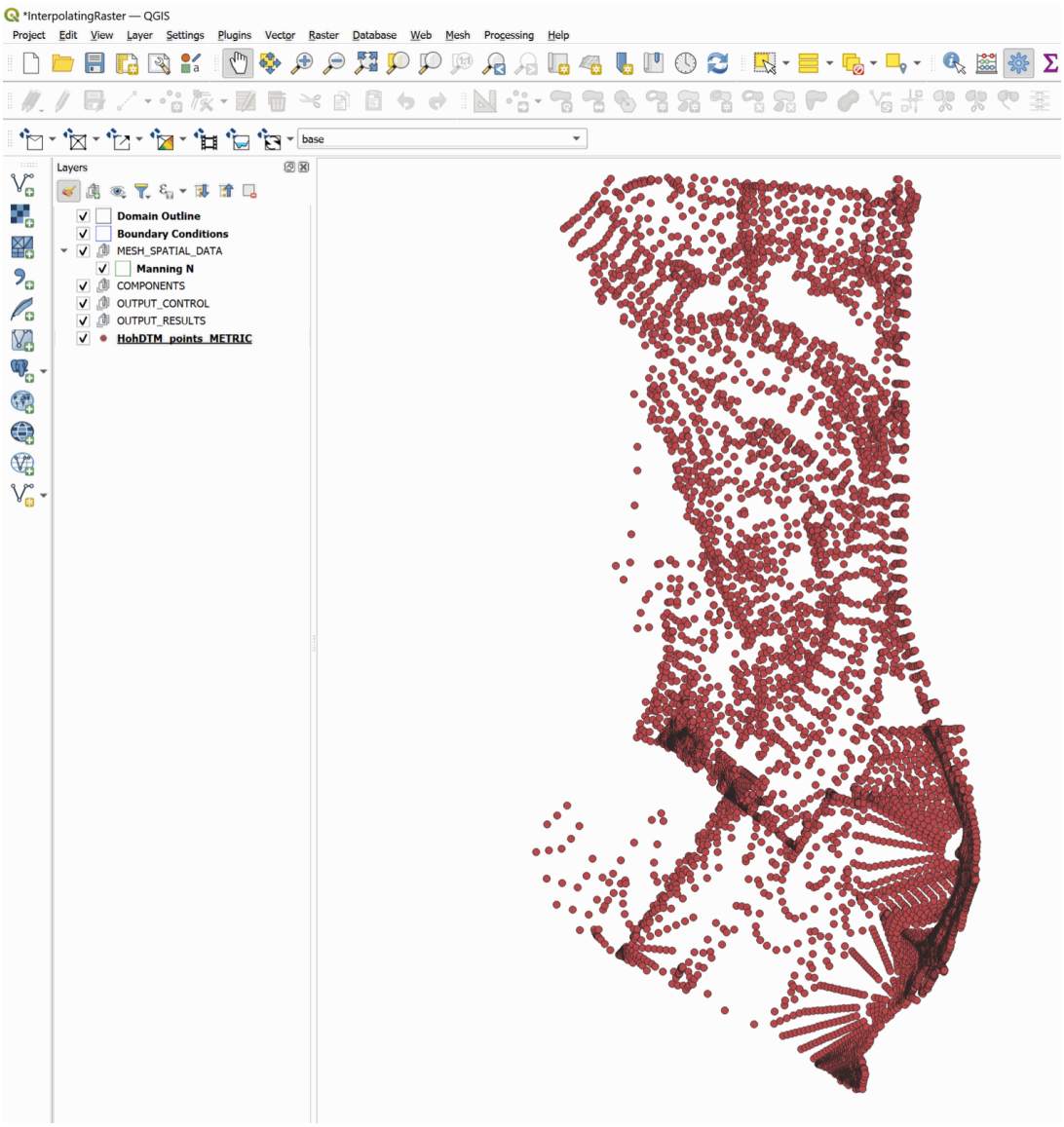
Dialog to create a layer from delimited data text file.

7. To go to the area where the points are located, you will have to right-click on the label of the created layer and select the option Zoom to the layer.



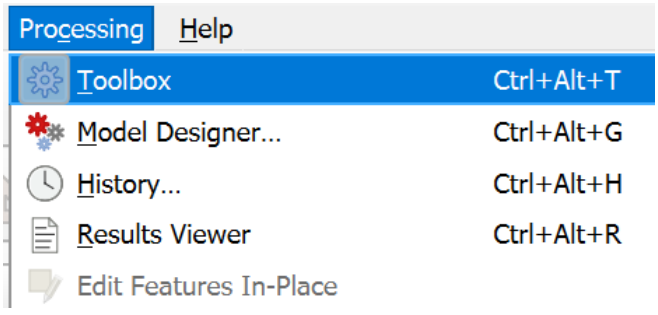
How to zoom a layer.

8. The points should appear in the project window:

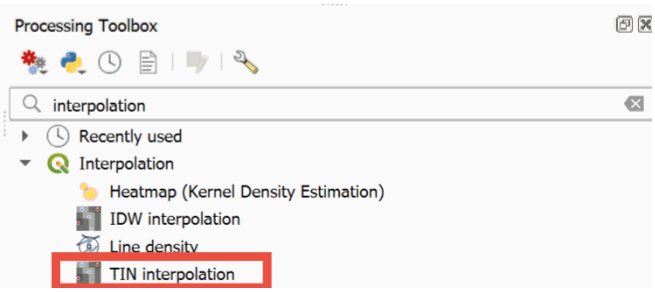


Layer of points created from delimited data file.

9. The raster is created interpolating from the data in the point layer. For that we will use the *Processing Toolbox* to load the panel that will allow us to search for the *TIN interpolation* command.



Opening the Processing Toolbox.



Loading to the QGIS TIN interpolation tool.

10. In this window select the layer to be interpolated to create the raster, in this example we use the point layer that we just created named *HohDTM_points_METRIC*. Change it by selecting the dropdown for *Vector layer*.

11. We select the field that has the attribute that was used as the value to interpolate, in this case the z field. This is done by clicking the dropdown for *Interpolation Attribute*, and select *z*.

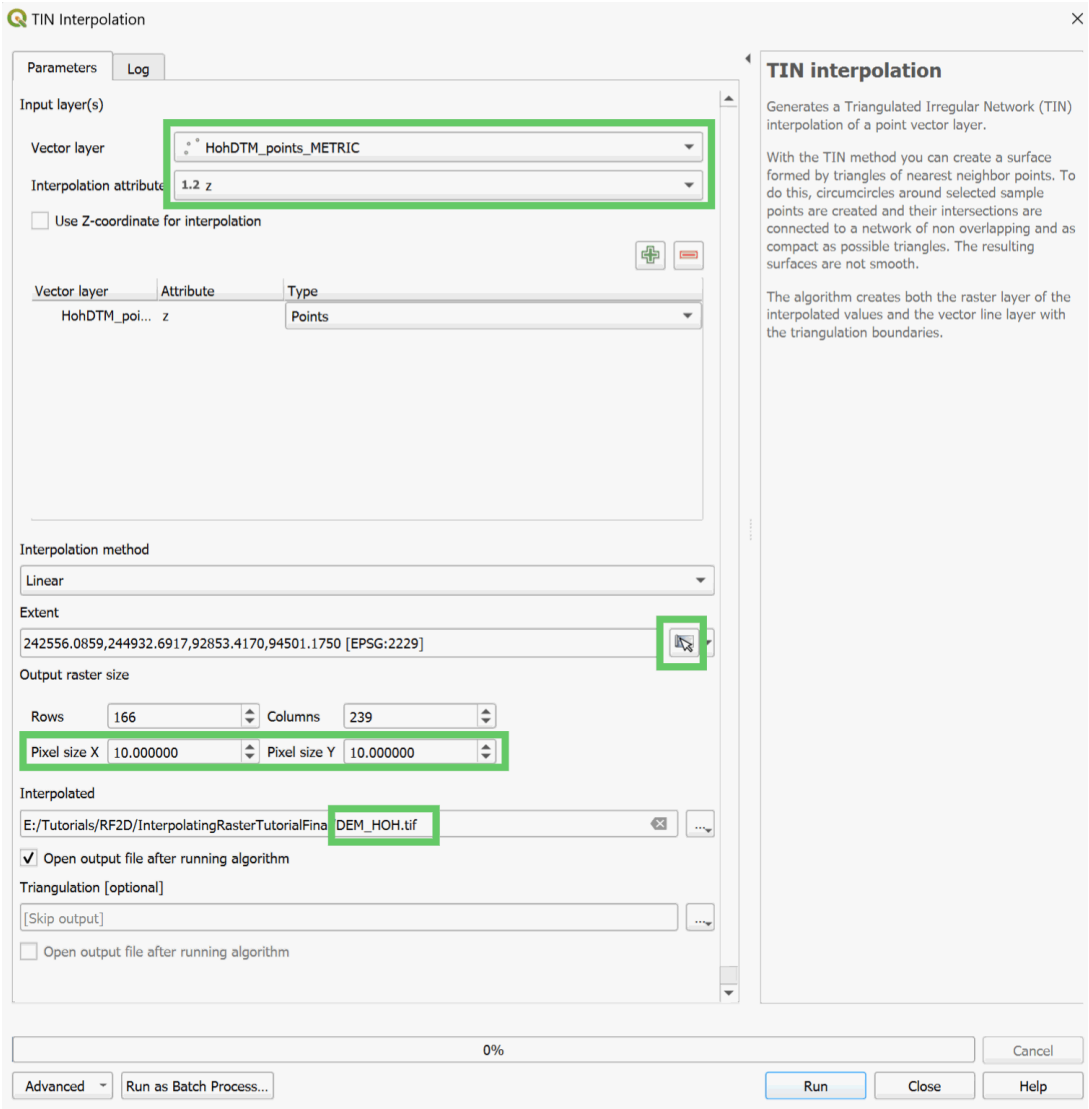
12. Click the green + button to add the vector layer into the list.

13. Click on the *Set to current map canvas extent* button and it shoudl automatically fill in the extent information.

14. In the *Pixel size* textboxes, enter *10*. You will notice the *Rows* and *Columns* textboxes are automatically filled in based on the pixel size.

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

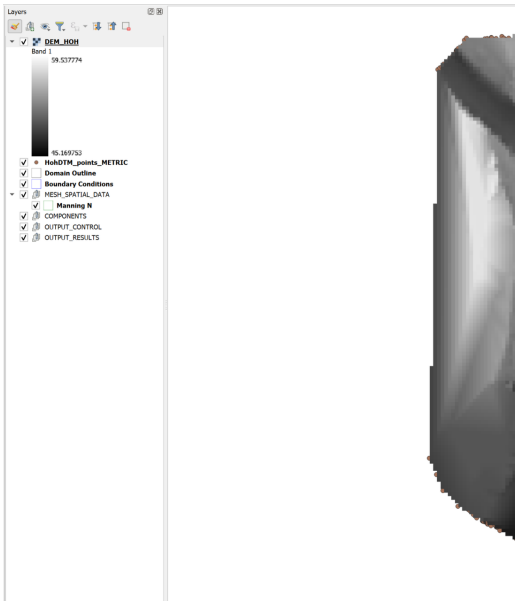
    The panel should look like the following figure:



Interpolation plugin window.

16. Click OK to start the interpolation process.

Once the process is completed, the raster resulting from the interpolation will be displayed on the



screen, by default it is rendered in gray gradient as shown in Figure 19.8.

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

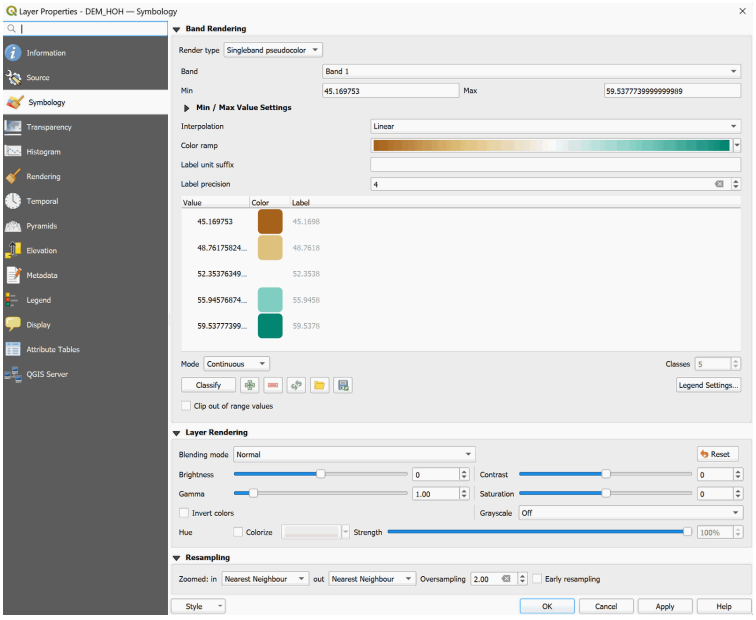


Figure 15.2 – Window to change the render style of a raster layer.

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

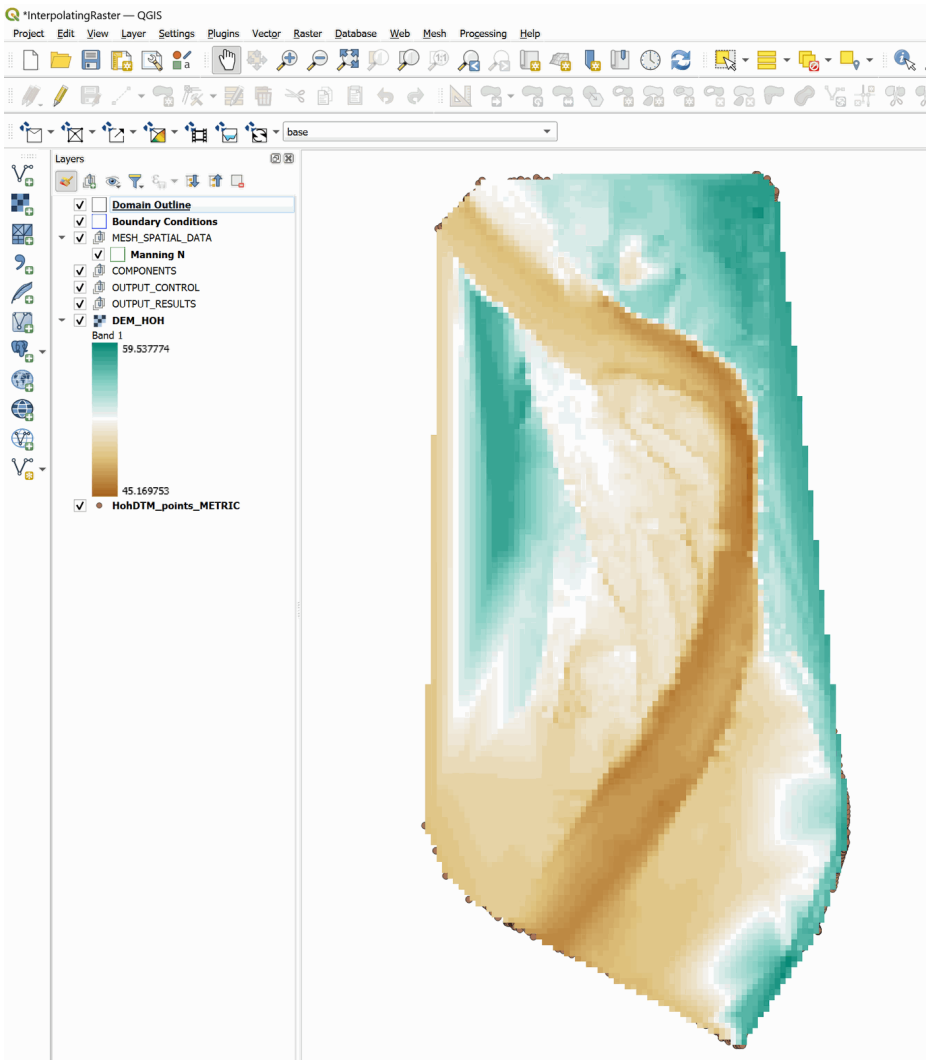


Figure 15.3 – Digital elevation model with color render.

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

This concludes the Interpolating Raster tutorial.

16

Setting up a Google Cloud VM for GPU-Accelerated Simulations

This tutorial provides a step-by-step guide for setting up a powerful virtual machine (VM) on Google Cloud Platform (GCP) equipped with an NVIDIA Tesla V100 GPU. Such a setup is ideal for running computationally intensive OilFlow2D or OilFlow2D simulations that can leverage GPU acceleration, significantly reducing model run times.
Note: Running a VM with a GPU on Google Cloud will incur costs. Windows Server images and GPUs are premium resources. Please review GCP pricing and ensure billing is enabled for your project. The Google Cloud Free Trial does not typically cover GPU usage. See the official documentation for more details: https://cloud.google.com/compute/docs/create-windows-server-vm-insta
The process involves these main steps:

1. Configure and create a new Compute Engine VM instance.
2. Specify the machine type (4 vCPUs and 32 GB Memory).
3. Add and configure an NVIDIA Tesla V100 GPU.
4. Select a Windows Server boot disk.
5. Connect to the newly created VM.

16.1 Before you begin

1. You must have a Google Cloud account with a project created and billing enabled.
2. Ensure your account has the necessary IAM permissions. At a minimum, you will need the 'Compute Instance Admin (v1)' and 'Service Account User' roles for the project.

3. Enable the Compute Engine API for your project. This can be done from the APIs & Services dashboard in the Google Cloud Console.

4. You will need an RDP client on your local machine to connect to the Windows VM. Microsoft's Remote Desktop client is available for Windows, macOS, iOS, and Android.

16.1.1 Checking and Requesting GPU Quotas

By default, Google Cloud projects have a GPU quota of zero for most regions to prevent accidental usage. Before you can create a VM with a GPU, you must request and be granted a quota for the specific type of GPU (e.g., NVIDIA Tesla V100) in your desired region.

1. In the Google Cloud Console, navigate to **IAM & Admin > Quotas**.

2. In the **Filter** bar, select the **Metric** property and search for NVIDIA V100 GPUs. Select it from the list. If it does not appear, you may need to clear the filter and first filter by **Service** for Compute Engine API.

3. This view will show your current GPU quota for all regions. Most will likely be at 0.

4. Find the region where you plan to create your VM (e.g., 'us-central1') and select the checkbox next to it.

5. Click the **EDIT QUOTAS** button at the top of the page.

6. A panel will open on the right. Enter your desired new quota limit (e.g., 1 for one V100 GPU) and provide a brief justification for your request (e.g., "Required for running GPU-accelerated scientific modeling software").

7. Submit the request. Quota increase requests can take some time to be reviewed and approved by Google. You will receive an email notification once the process is complete. **You cannot proceed with creating the VM until your quota request is approved.**

16.2 Creating the VM Instance

1. In the Google Cloud Console, navigate to the **Compute Engine** service and select **VM instances**.

2. Click the **CREATE INSTANCE** button at the top of the page. This will open the instance creation form.

3. **Name your instance**: Enter a descriptive name, for example, 'hydronia-gpu-workstation'.

4. **Region and Zone**: Select a region (e.g., 'us-central1'). For the **Zone**, select **Any (recommended)**. This allows Google to pick an available zone, which is necessary to trigger the quota request workflow if your quota is insufficient.

16.3 Configuring Machine and GPU

This is the most critical step, where you define the computational resources for your VM. The following steps correspond to the selections shown in Figure 20.1.

1. In the main **Machine configuration** section, first click the **GPUs** button. This filters the available options to those that support graphics processors.

2. A new **Graphics processing units** subsection will appear. Configure it as follows:

    - **GPU type**: Select **NVIDIA V100**.
    - **Number of GPUs**: Select **1**.
    - Leave the **Enable Virtual Workstation (NVIDIA GRID)** checkbox *unchecked*.

3. Next, configure the CPU and memory for the VM itself:

    - **Series**: Ensure **N1** is selected.
    - **Machine type**: Click the **Custom** button.
    - **Cores**: Drag the slider or type **4** into the vCPU box.
    - **Memory**: Drag the slider or type **32** into the GB box. Ensure the **Extend Memory** checkbox is active.

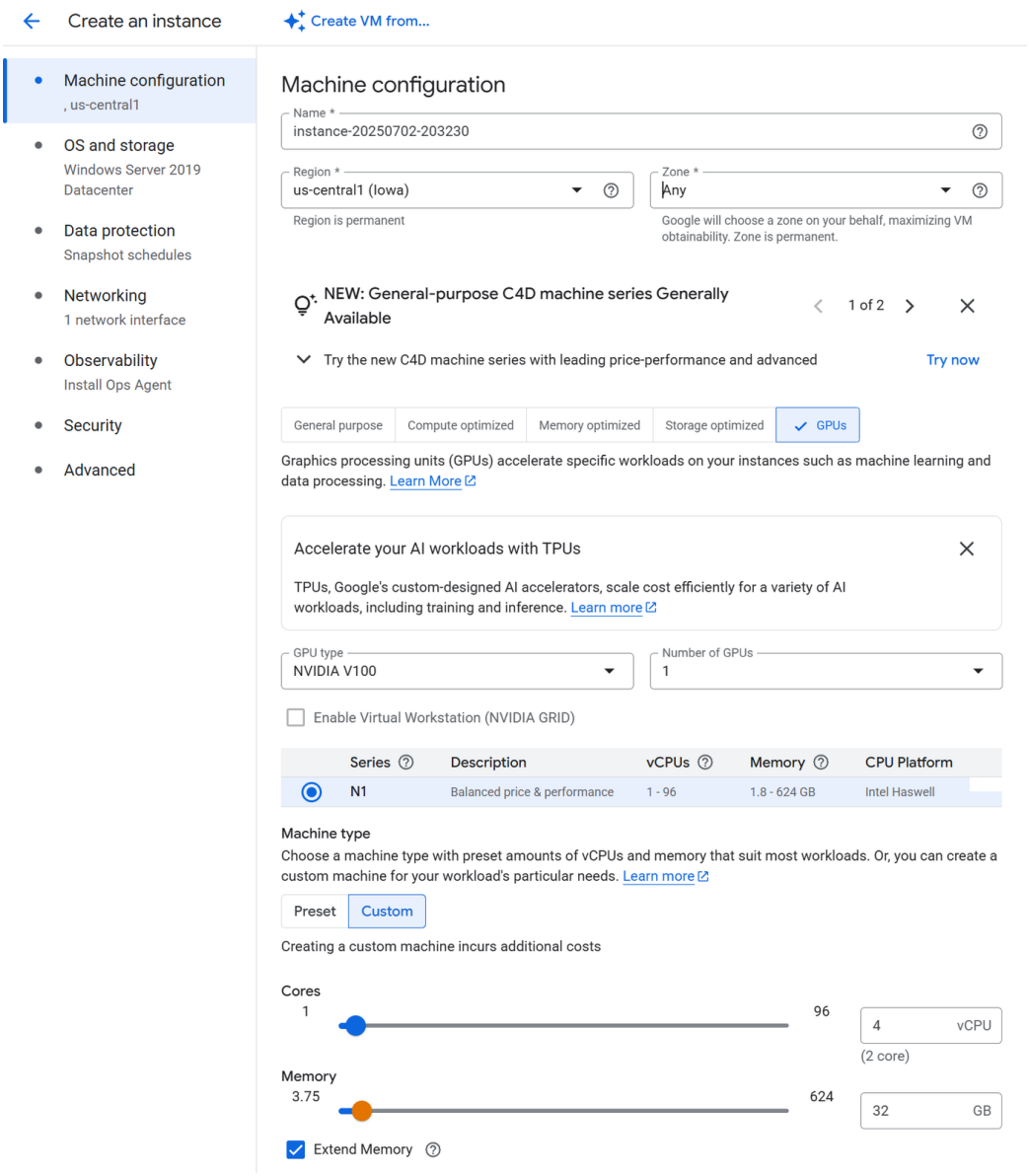


Figure 16.1 – Configuring the machine with a custom N1 type, 4 vCPUs, 32 GB of memory, and one NVIDIA V100 GPU.

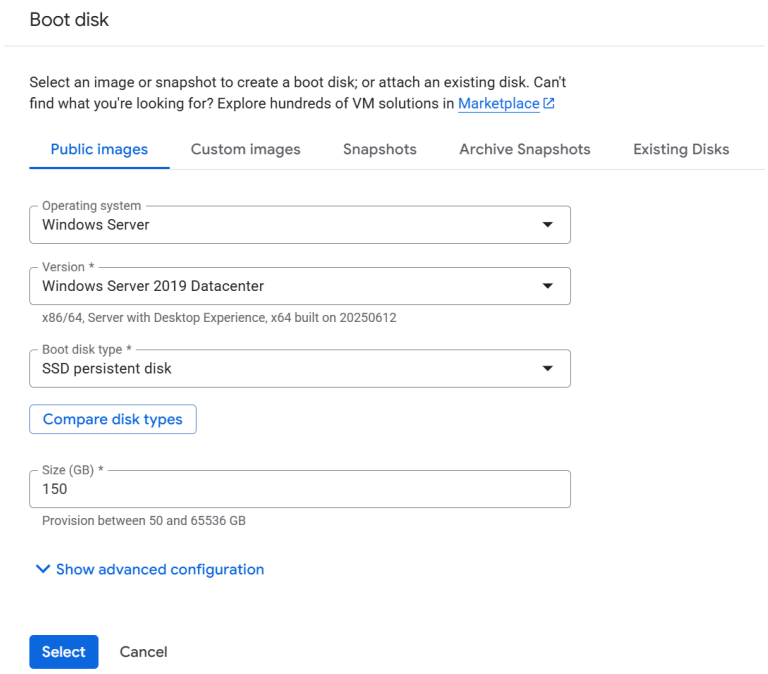
16.4 Configuring the Boot Disk and Firewall

1. In the **OS and Storage** section, click the **Change** button.

2. In the fly-out menu, make the following selections:

   - **Operating system**: Select **Windows Server**.
   - **Version**: Select **Windows Server 2019 Datacenter** or a newer version if available.
   - **Boot disk type**: Select **SSD persistent disk** for better performance.
   - **Size (GB)**: Set a size appropriate for your needs, for example, **150** GB.

3. Click **Select** to confirm the boot disk configuration.



Configuring the Windows Server 2019 boot disk.

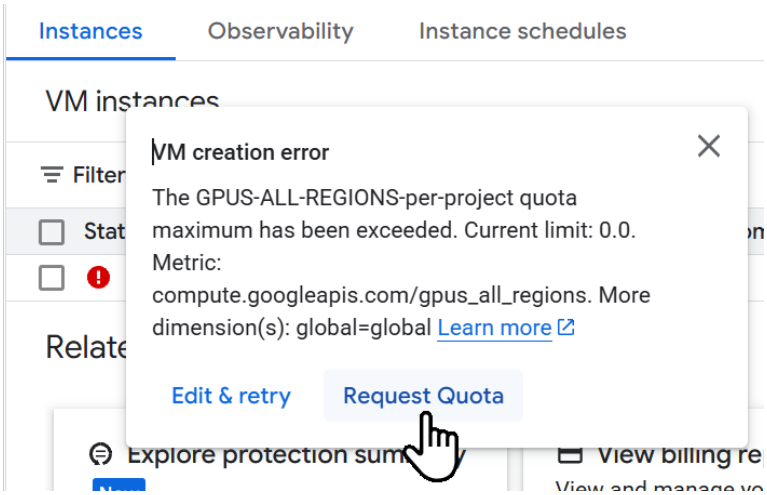
4. In the **Firewall** section of the main creation page, check both **Allow HTTP traffic** and **Allow HTTPS traffic**. RDP access is enabled by default for Windows instances.

16.5 Create the VM and Handle Quota Requests

1. Review all the settings one last time. At the right side of the page, you will see a monthly cost estimate.

2. Click the **Create** button.

3. If this is your first time creating a GPU-enabled VM, the creation will likely fail with a “VM creation error” pop-up, as shown in Figure 20.3. This is expected and is the easiest way to request the needed quota.

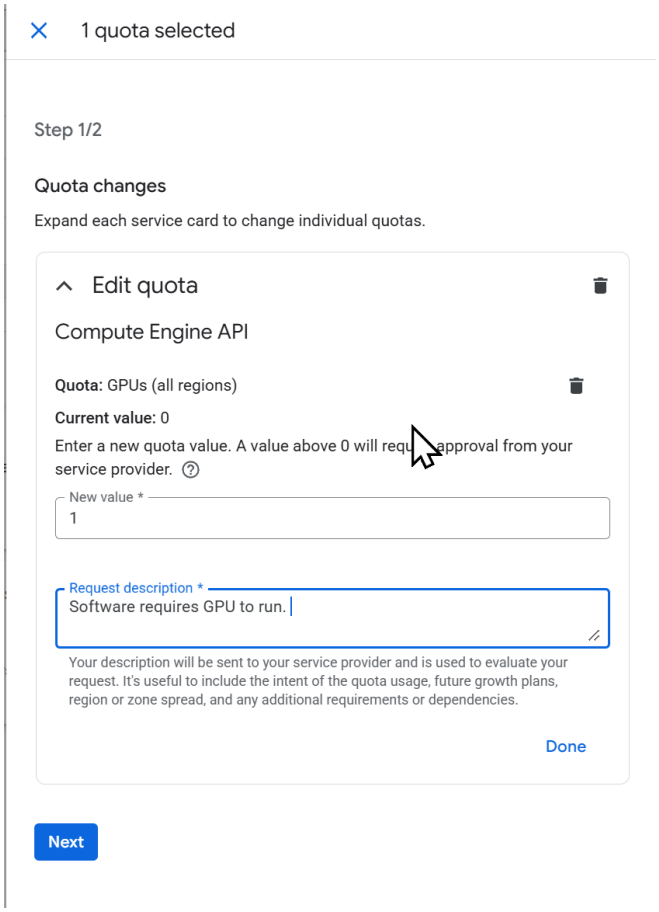


VM creation error due to GPU quota.

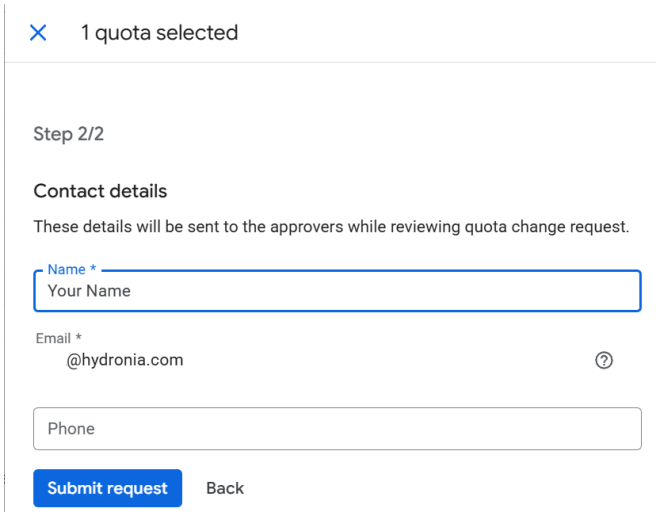
4. Click the **Request Quota** button directly within the error dialog.

5. This will open the "Quota changes" request form. You will proceed in two steps:

   - **Step 1/2: Edit Quota** (Figure 20.4)
     - In the New value box, enter 1.
     - In the Request description box, provide a brief justification, such as "Software requires GPU to run for scientific modeling."
     - Click Done, then click Next.
   - **Step 2/2: Contact Details** (Figure 20.5)
     - Fill in your contact information.
     - Click Submit request.



Requesting a quota increase.

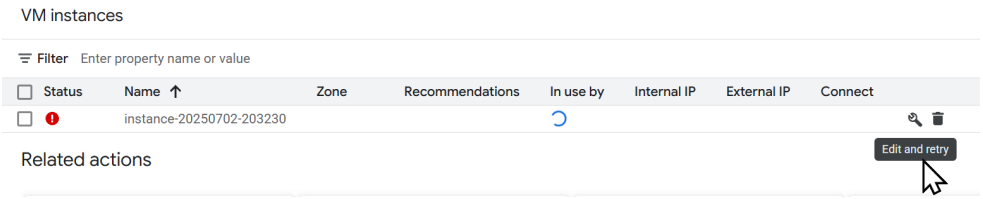


Providing contact details for the quota request.

6. The quota request will be sent to Google for approval. This can take anywhere from a few minutes to a few business days. You will receive an email once the request has been approved. **You must wait for the quota approval before you can successfully create the VM.** Once approved, return to the **VM instances** page in the Google Cloud Console.

7. You will see your failed instance with a red error icon. On the far right of the instance row, click

the wrench icon to **Edit and retry** (Figure 20.6).



Using the Edit and retry option on a failed VM instance.

8. This will take you back to the instance creation page with all of your previous settings pre-filled. Scroll to the bottom and click **Create**. The VM should now be provisioned successfully.

16.6 Connect to the VM

Once your quota request is approved and the VM has been successfully created (indicated by a green checkmark icon on the **VM instances** page), you can set your password and connect.

1. Click on the name of your new VM instance to go to its details page.
2. Under the **Remote access** section, click the **SET WINDOWS PASSWORD** button.
3. A default username (your Google account username) will be shown. You can keep it or change it. Click **SET**. A new password will be generated. **Copy this password and store it securely**.
4. You can now connect to your VM. Return to the VM instance details page, click the **RDP** button, and choose **Download the RDP file**. Open this file with your RDP client.
5. When prompted, enter the username and the password you just copied. You are now connected to your new Windows Server VM with a powerful GPU, ready for your simulation work.

This concludes the Google Cloud VM Setup tutorial.

17

Creating high-impact graphics and animations using Paraview

ParaView is an open-source, multi-platform data analysis and visualization application. You can quickly build visualizations to analyze model results using qualitative and quantitative techniques. The data exploration can be done interactively in 2D and 3D, or using ParaView's batch processing capabilities.

This tutorial will demonstrate the use of Parview to generate high quality graphics, including depth maps, velocity fields, 3D visualizations, and animations of OilFlow2D results.

> **Note**
>
> This tutorial requires ParaView version 5.8.1 or later. You can download and install ParaView version 5.8.1 or later from the website www.paraview.org. To create the ParaView graphs, the OilFlow2D model needs to generate during runtime the '.vtk' files using the *Create graphic output files* option in the Hydronia Data Input Program *Graphic Output Options* panel.

The files required to follow this tutorial can be extracted from the 'ExampleProjects' zip file under the 'ParaviewTutorial' folder. This zip file is downloaded separately from your installation materials.

17.1 Paraview basics

After loading Paraview, the following components can be identified in the main window (see Figure 21.1):

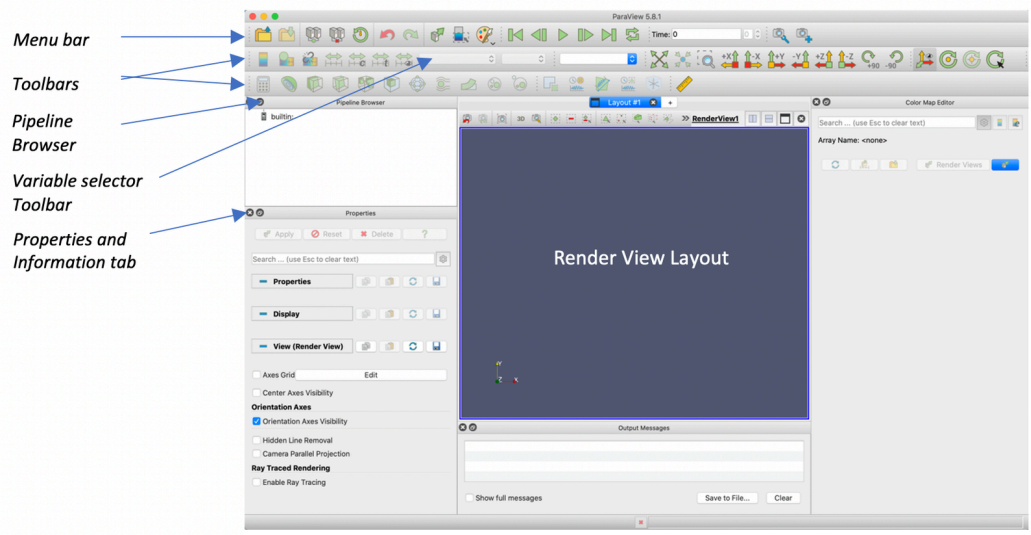


Figure 17.1 – Main Paraview window.

To open the tutorial example group file, click  in the Menu bar and double-click the file group 'BridgeTutorial..vtk' in the 'ParaviewTutorial' folder. The group contains multiple '.vtk' files, where each file corresponds to a specific simulation time.

Make it visible clicking *Apply* in the *Properties* tab. The following graphic in the *Render View layout* will look as follows:

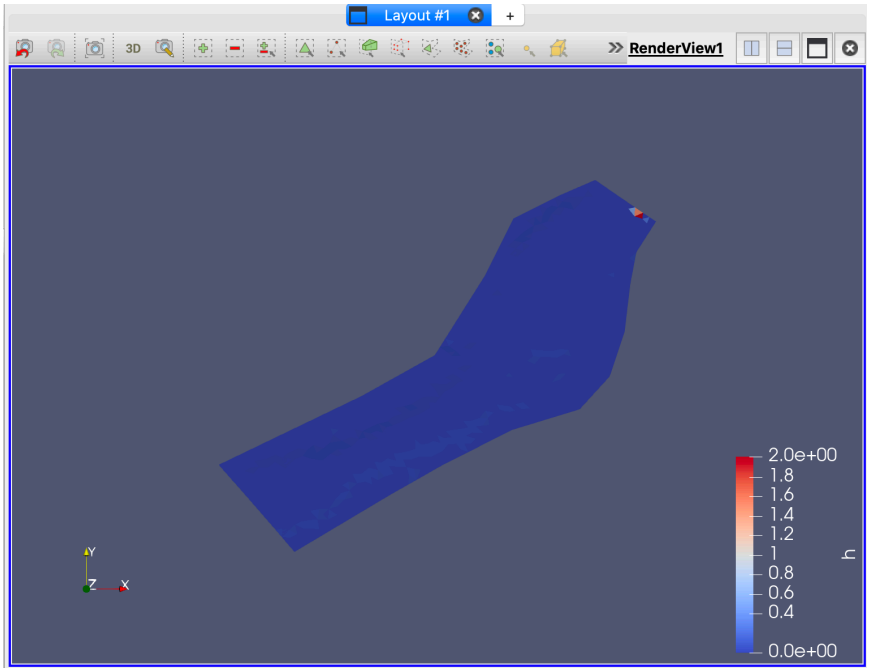


Figure 17.2 – View after opening the 'BridgeTutorial..vtk' file.

Using the Time control toolbar (see Figure 21.3), select a time to visualize the selected variable.



Figure 17.3 – VCR and Time Control toolbars.

The graphic will change according with the selected simulation time. For example, for *Time = 2* it should look similar to this:

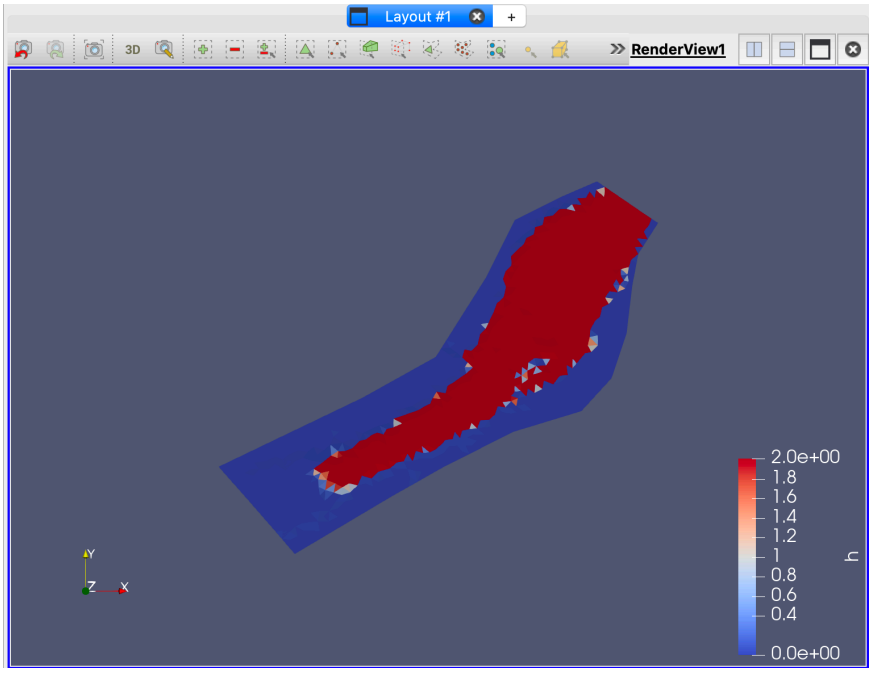


Figure 17.4 – View for Time = 2.

An adequate visualization depends on the selection of a color scale of a defined variable. In the previous figure, the default color scale was selected associated with the depth variable (h). The next section shows how to customize color rendering.

17.2 Two-dimensional (2D) visualizations

17.2.1 Create a 2D bed elevation map

Paraview has many filters for full visualization of 1D, 2D and 3D data. In this part of the tutorial we will create a 2D bed elevation graphic. Select the bed elevation variable z in the *Variable selector*:

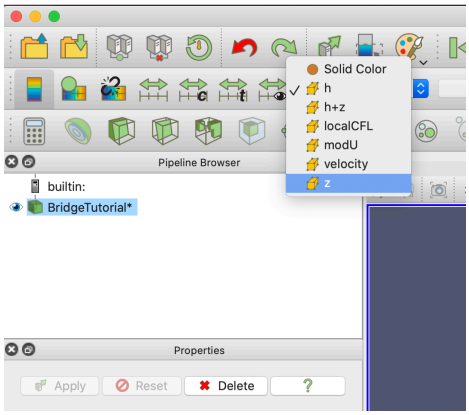


Figure 17.5 – Variable selector.

The representation of the bed elevation z will look like this:

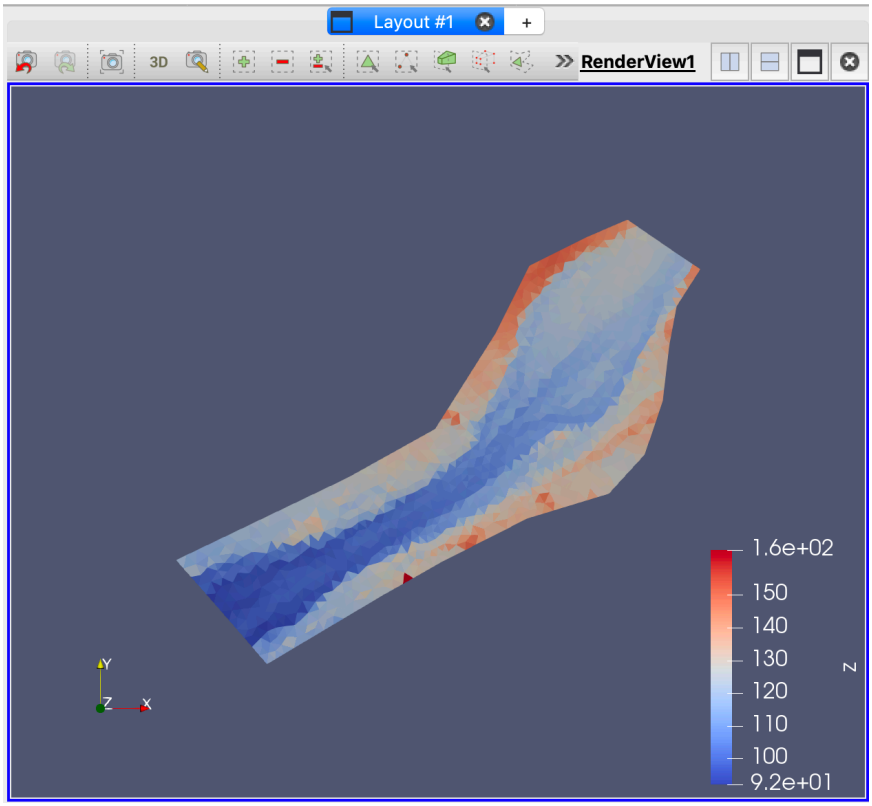


Figure 17.6 – Bed elevation z representation.

Although it is possible to customize the color maps, we will see how to do it later in this tutorial, for this example we will use a predefined color.

Click  in the *Properties* tab to pick up the Rainbow Desaturated color map in the *Choose Preset* dialog box. Once selected, *Apply/Close*.

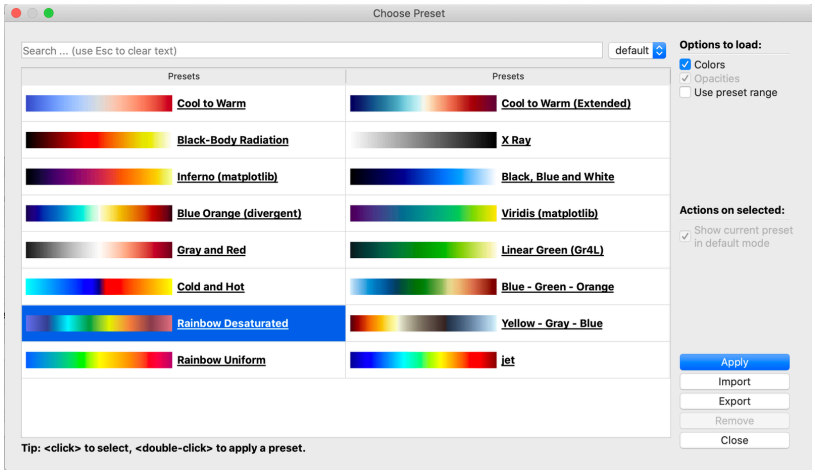


Figure 17.7 – Predefined color maps.

The result graph visualizes of the bed elevation z.

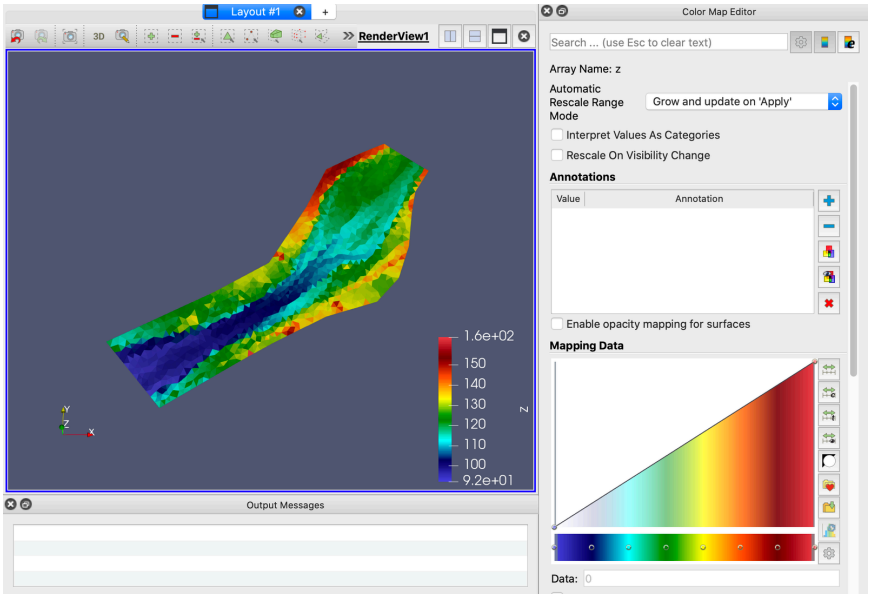


Figure 17.8 – Bed elevation z representation.

17.2.2 Creating 2D velocity vector fields

In order to create a 2D visualization of the flow and its magnitude with our project data, we will start with a water depth h representation in a white-to-blue color scale. The first step is to select h in the *Variable selector*. To change the color map for water depth h to a white-to-blue, click  and choose the *XRay* preset color then *Apply/Close*. The *Mapping Data* should look as:

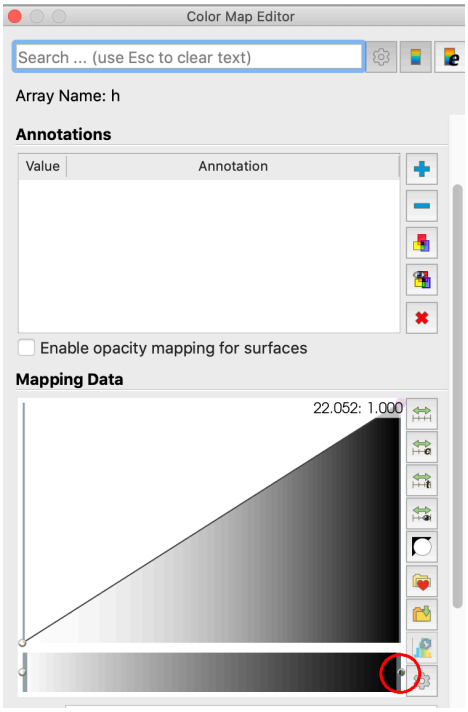


Figure 17.9 – XRay Color Map.

Double click in the right circle of the range marker bar and select any blue color from the *Select Color* dialog box. This example uses *Red = 0*, *Green = 85* and *Blue = 255* and Click *OK*

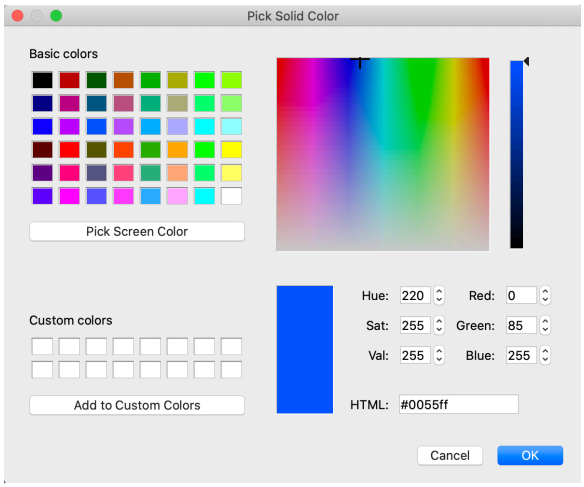


Figure 17.10 – White-to-blue color map.

The *Mapping Data* will look as follows:

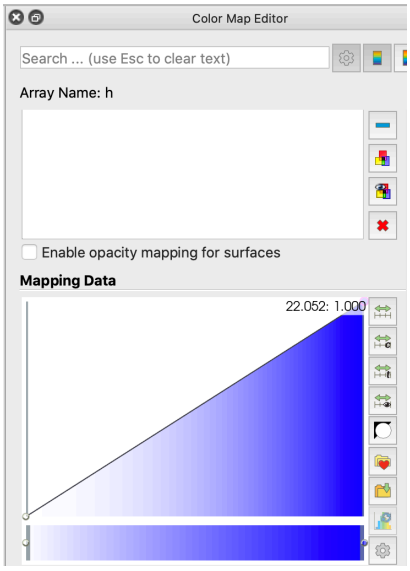


Figure 17.11 – White-to-blue color map.

Set the *Time = 20* and the result should look as follows:

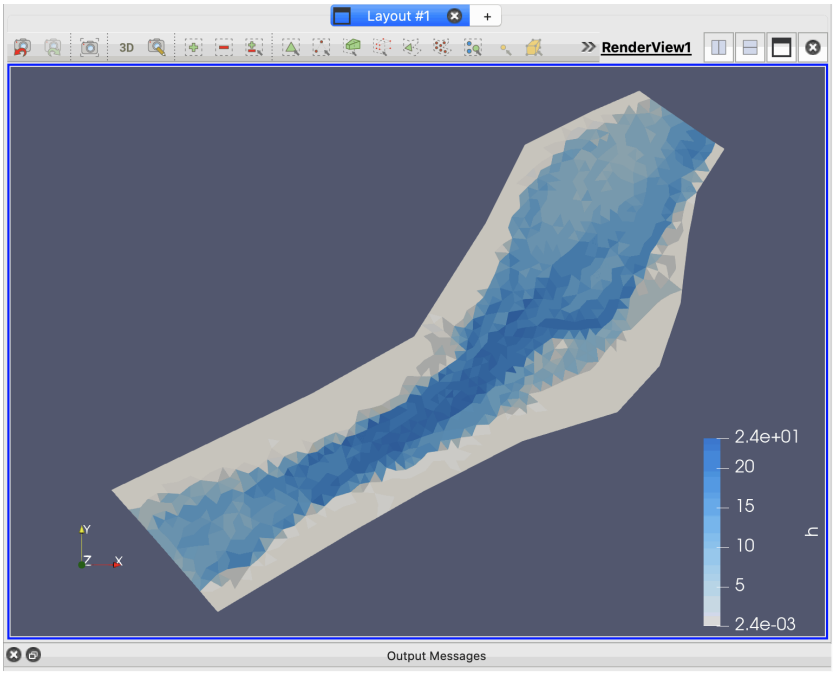


Figure 17.12 – Water depth (h).

To create a 2D velocity vector map for this example, follow these steps:

- Select *BridgeTutorial** in the *Pipelines Browser*
- In the *Filters* menu select *Common/Glyph*
- Click *Apply* in the *Properties* panel. You will adjust the big arrows after
- Configure the Vector field configuration in the *Properties* panel as follows:

- Glyph type: *2D Glyph*
- Scale Array: modU (velocity modulus)
- Scale Factor: 35
- Glyph mode: *Every Nth Point*
- Click *Apply* to visualize the the filter setup. The user can press this button each time changes are made
- Coloring: modU

- Choose the Rainbow Desaturated color scale clicking in  and *Apply/Close*.

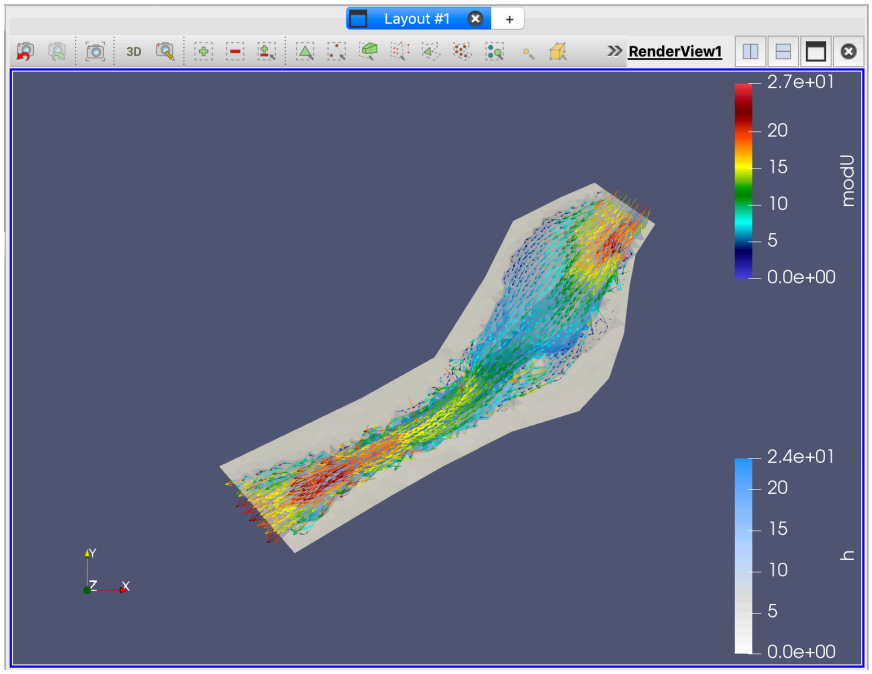


Figure 17.13 – Velocity vector field on water depth.

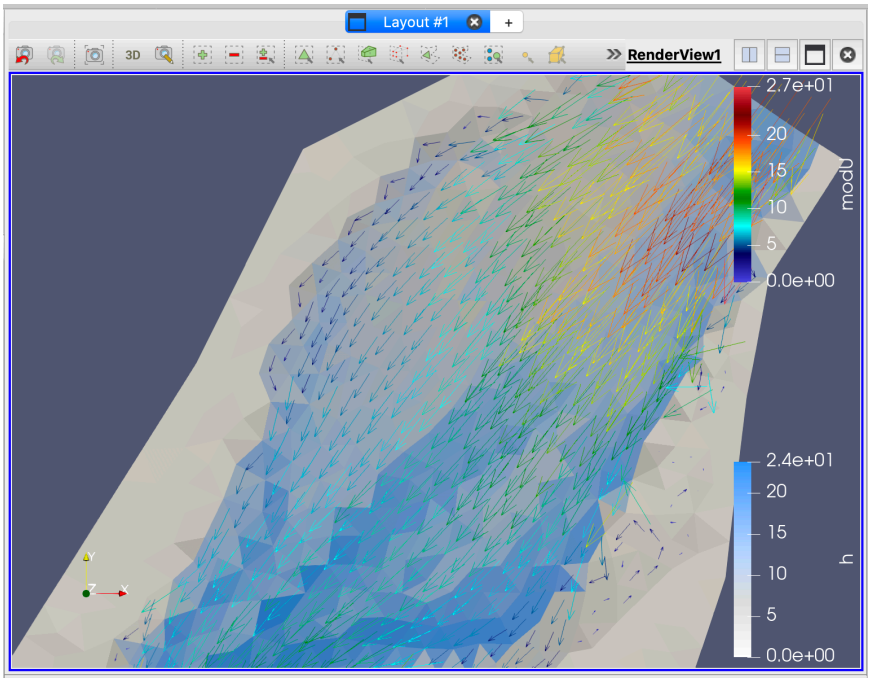


Figure 17.14 – Velocity vector field detail.

In order to save the Paraview project, select *File/Save State*. This will save a '.pvsm' file.

17.3 Three-dimensional (3D) visualizations

In this part of the tutorial we will explain the steps to create a 3D visualization using Paraview. We start where section 21.2 ends. This assumes that we have a visualization of the bed elevation (z) variable using the *Rainbow Desaturated* color map.

Open the 'BridgeTutorial..vtk' group as explained in section 21.1 and select the bed elevation variable z in the *Variable selector*. The render view should look as the following figure:

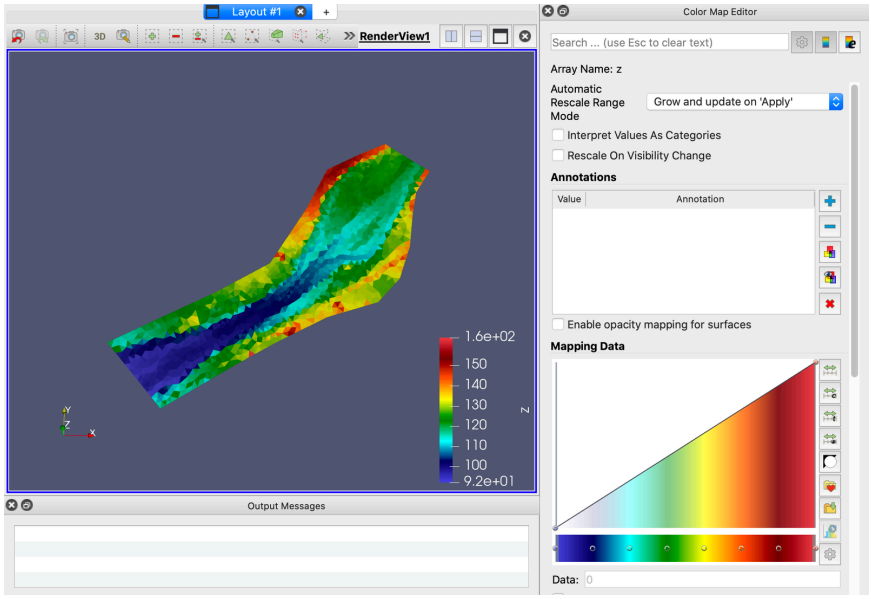
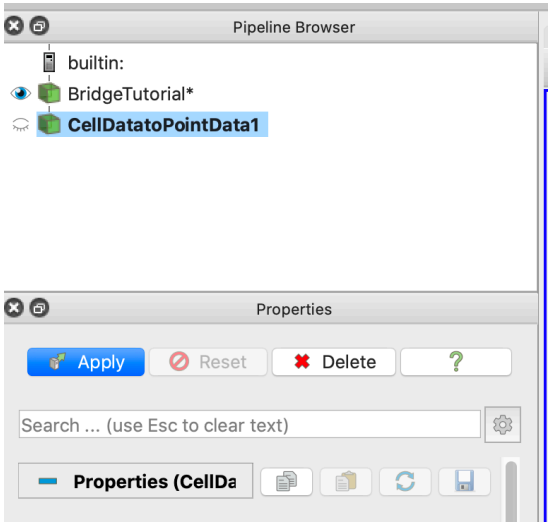


Figure 17.15 – Bed elevation z.

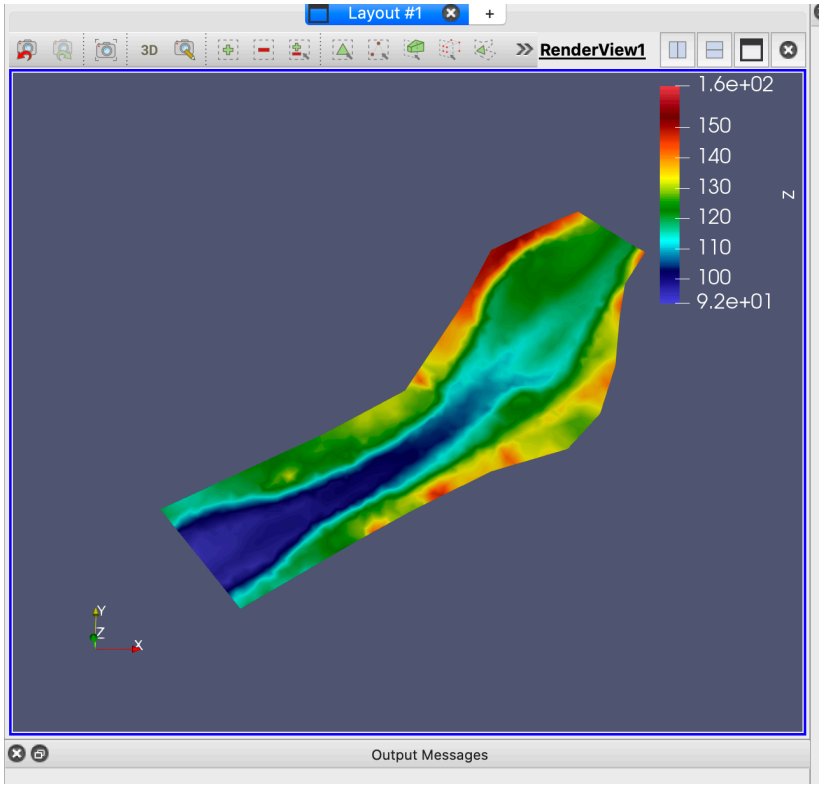
Generating a 3D visualization in ParaView requires an interpolation from *Cell Data to Node Data* as follows.

- Select *Cell Data to Point Data* in the *Filters/Alphabetical* menu. Then, press the *Apply* button in the *Properties* tab.



Cell Data to Point Data filter.

- Select again z in the *Variable selector*. Note that the color view is smoother due to the interpolation. The result should look as follows:



Cell Data to Point Data interpolation.

The 3D appearance is obtained by extruding one of the data variables z or (h+z).

- With the *CellDatatoPoint* selected in the *Pipeline Browser*, select *Warp by Scalar* in the *Filters/Alphabetical* menu to do the extrusion and then click *Apply*.

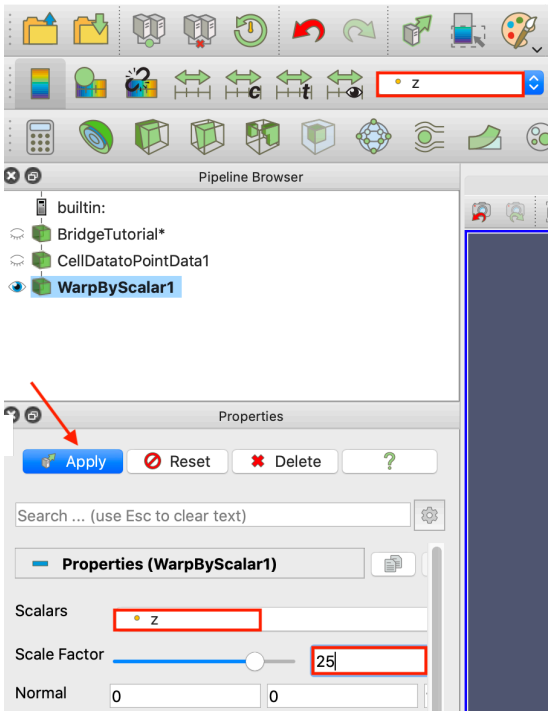
- Configure the *Properties* tab as follows:

  - Scalars = z

  - Scale Factor = 25

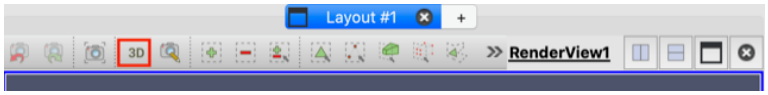
- Click *Apply*.

- Choose z in the *Variable selector*.



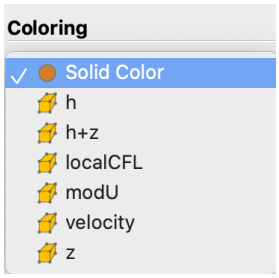
Wrap By Scalar filter.

- Switch from 2D to 3D visualization in the *Layout Render view* to manipulate the project in 3D.



3D option.

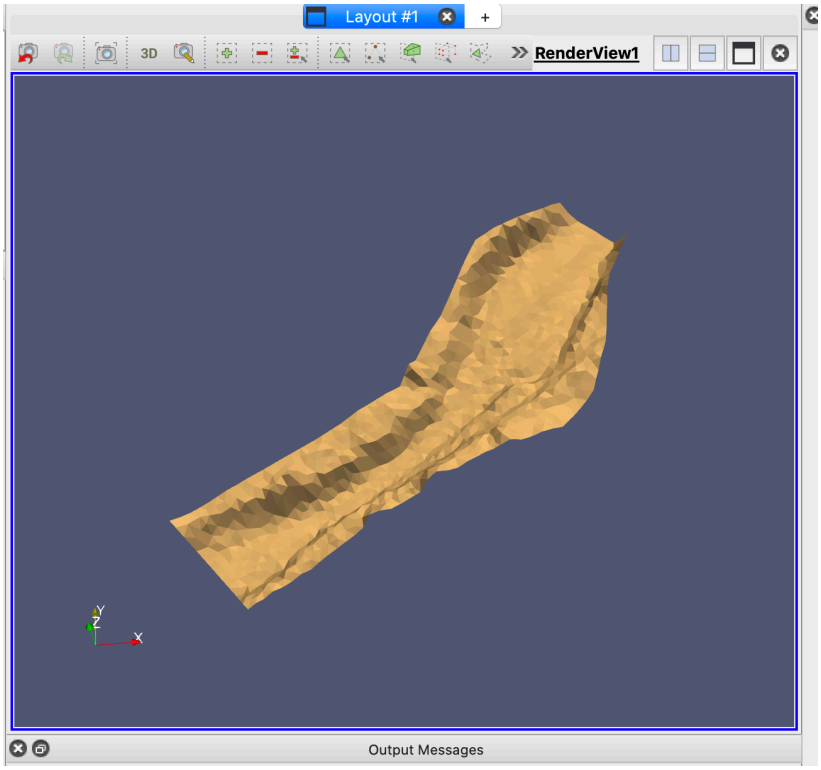
- Remove the color map of the bed elevation z by selecting *Solid Color* in the *Coloring* drop-down menu of the *Properties* tab:



Coloring parameter.

- The *Solid Color* can be customized clicking the Edit color map icon 

in the *Properties* tab. Once the desired color has been chosen at the *Pick Solid Color tab* (this example uses HTML=#ecb57d), the *Render view* should look as:



3D bed elevation representation.

To view the terrain from different points, use the left mouse button to rotate, press down the mouse wheel to translate, and scroll the mouse wheel to zoom in or out the image.

Using the *Viewpoint Toolbar* (Figure 21.22), you can add different view points of your graphic clicking  .



Figure 17.16 – Viewpoint Toolbar.

The following image was generated using an alternative view point.

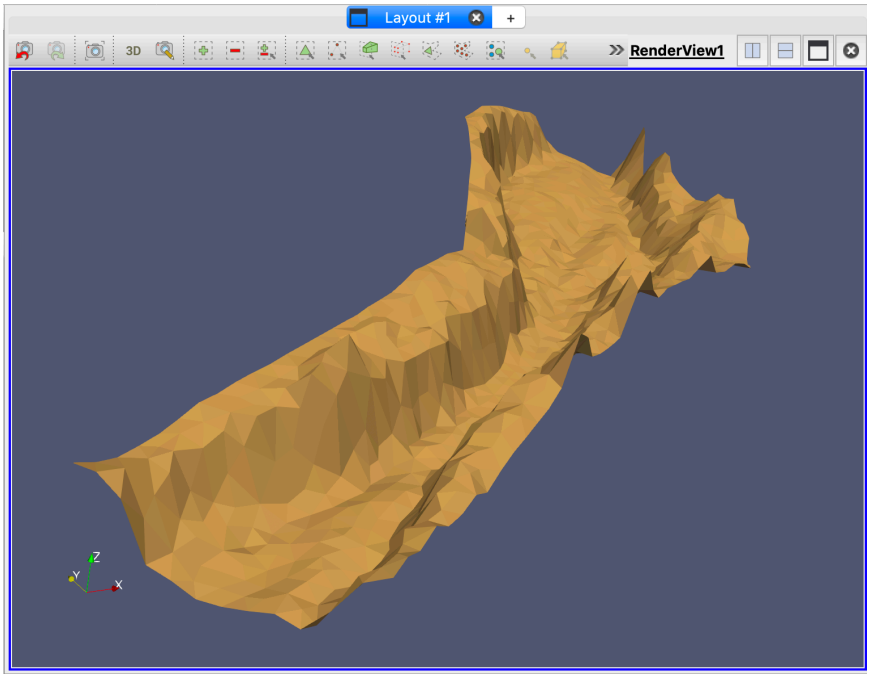


Figure 17.17 – Alternative 3D bed elevation representation.

To return to prearranged views use the *Camera control* toolbar.



Figure 17.18 – Camera Control Toolbar.

A more sophisticated visualization can include additional layers. In the next section we will add two more layers: the water elevation (h+z) and the velocity vector field.

17.3.1 Create a 3D water elevation graphic adding a (h+z) layer

Follow the following steps to create the (h+z) layer:

- Click on the *CellDataToPointData* item in the *Pipeline Browser* and select *Warp by Scalar* in *Filter/Alphabetical*. A *WrapByScalar2* item will be created in the *Pipeline Browser*.
- Select a time $\neq 0$ in the *Time control toolbar*, e.g.



- Configure the *Properties* panel as follows:
  - Scalars = (h+z)
  - Scale factor = 25 (very important)

- Click the *Apply* button. Check that h is in the *Variable selector*.

- Click  and choose the *Rainbow Desaturated* color map, then *Apply/Close*.

The resulting Render view should look similar to:

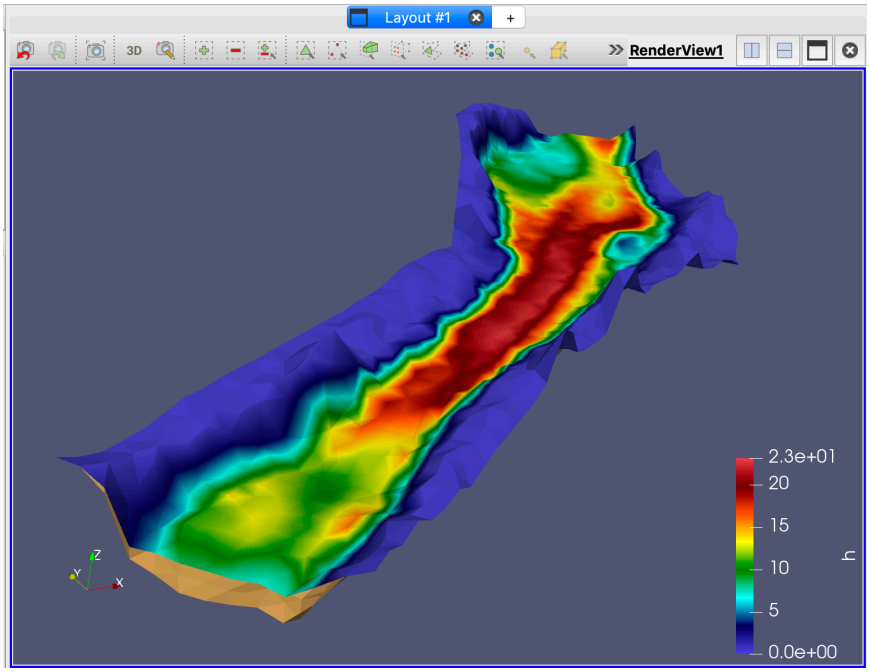


Figure 17.19 – Water surface elevation representation.

As seen in this figure, the layers corresponding to z and (h+z) variables are overlapped in the dry areas of the domain, generating a confusing presentation. One easy way to fix this issue is to remove the water depth h values below a user-defined value to avoid overlapping, doing as follows:

- Click *WrapbyScalar2* in the *Pipeline Browser* and select *Threshold* in *Filter/Common* in the main menu

- Click *Apply* in the *Properties* panel, and check that h is in the *Variable selector*.

- Set the minimum value equal to 0.01 m. Depths (h) below this minimum will not appear in the color representation.

- Click *Apply* and the result should be as follows:

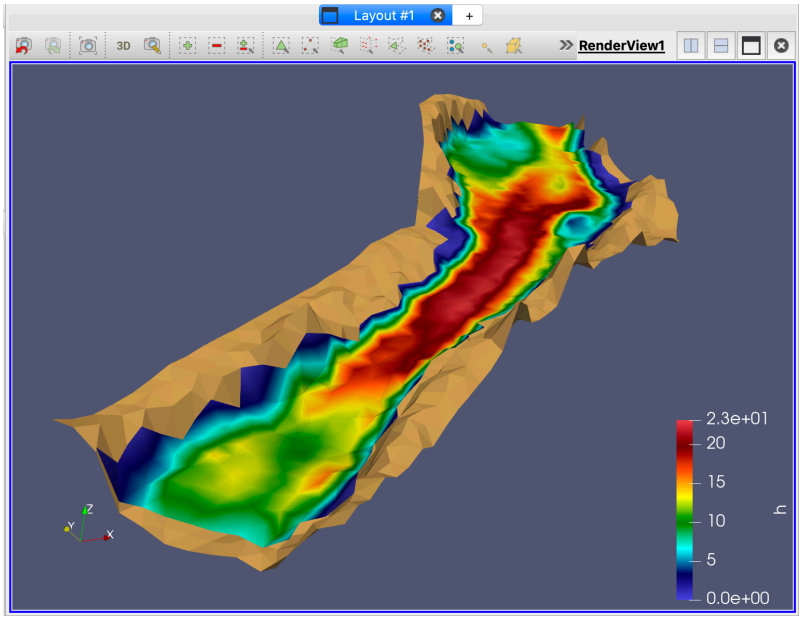


Figure 17.20 – Threshold Filter application.

With the *Threshold* layer selected, change the color map for water depth h to a white-to-blue as it was explained in Section 21.2.2 of this tutorial. The result graph should look as follows:

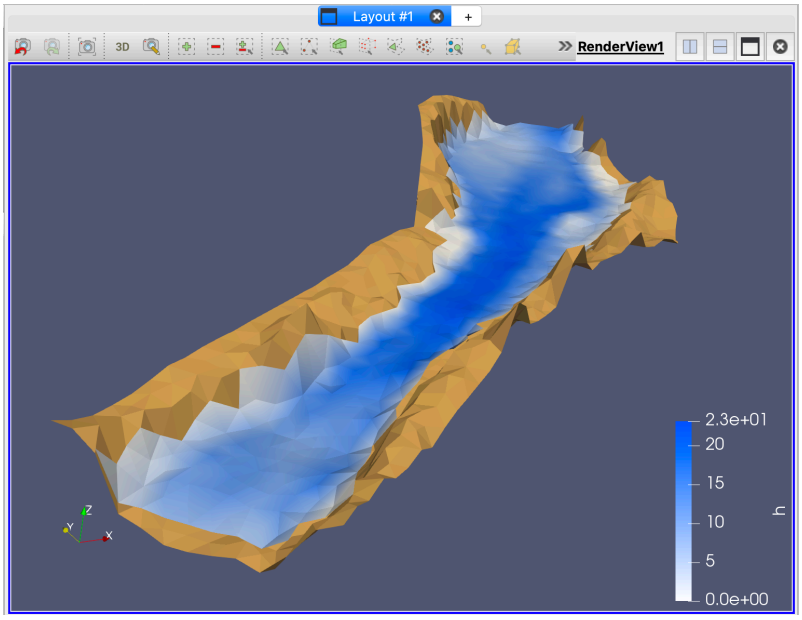


Figure 17.21 – Three dimensional water depth h representation.

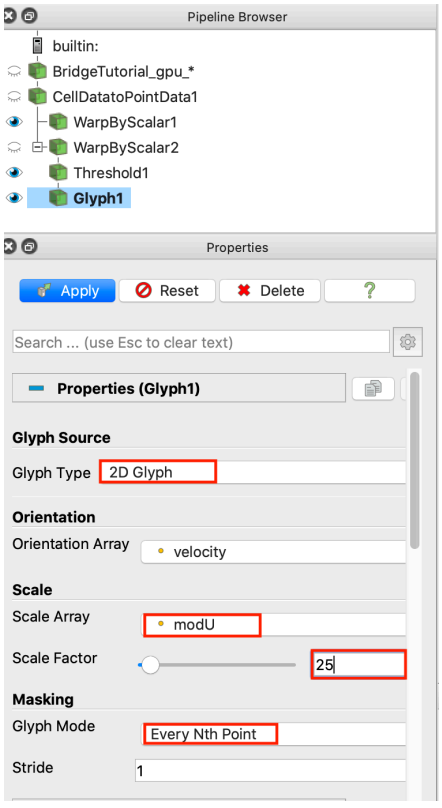
17.3.2 Create a velocity field graphic

Follow the following steps to create a velocity field graphic:

- Select the *Threshold* item in the*Pipeline Browser*.
- In the Filters menu select *Common/Glyph* and click the *Apply*.

- Configure the *Properties* panel as follows:

  - Glyph type = *2D Glyph*
  - Orientation Array = velocity
  - Scale Array = modU (velocity modulus)
  - Scale Factor = 25
  - Glyph mode = *Every Nth Point* (to prevent a saturation of arrows in the visualization)
  - Click the *Apply* button to visualize immediately the filter setup, the user can press this button after each of the previous steps
  - Set Coloring as (modU) for the color map representation and choose the preset color Rainbow desaturated then *Apply/Close*



- Select Time $\neq 0$ as the current time control tool bar. In this example chose for instance *Time=3*.

After these steps, the graphic should look like this one:

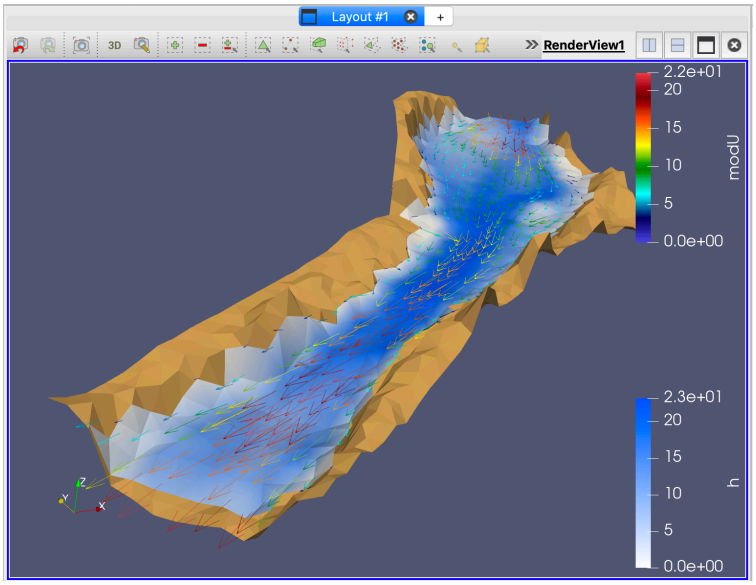


Figure 17.22 – Three-dimensional velocity field representation.

In order to save this Paraview project select *File/Save State* and create a '.pvsm' file. To load any of the saved projects select *File/Load State* and select a '.pvsm' file.

17.4 Generating animations

With your Paraview project loaded in your computer to create an animation or movie go to *File/Save Animation* and create a file type '.avi'. The *Save Animation Options* dialog will be displayed:

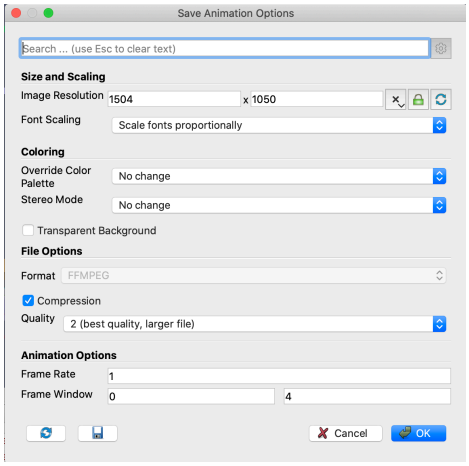


Figure 17.23 – Animation.

In this dialog, the user can configure the video frame rate, the number of frames per time step, the resolution (in pixels) and the range of time steps. In order to reduce the size of the output video file, a compression mode is also available for the video. Note that this will also reduce the video quality of the animation. As an example, the next figures show three frames of the output video generated in this way, corresponding to six different simulation times.
Animation time frames.

17.5 Streamlines representation

Streamlines represent the instantaneous direction of the velocity and is drawn as unending lines that may converge or diverge from one another. They are drawn at roughly even intervals to capture the flow in all areas.

To create Streamlines with Paraview, follow these steps:

- Open the Paraview application in your computer
- Load the plugin *StreamLinesRepresentation* in *Tools/Manage Plugins* in the main menu
- Open the 'BridgeTutorial..vtk' Group file and Select *Apply*
- Select a time different than 0 in the time control tool bar
- Select velocity in the *Variable selector*
- In the *Properties* panel select:
  - Representation = StreamLines
  - Step Length = 1
  - Number of Particles = 1000
  - Max Time to Live = 600
- To have Streamlines in one color, select in the Property panel Coloring = *Solid* and then click on 

and select a color.

The next figure shows how the StreamLines look:

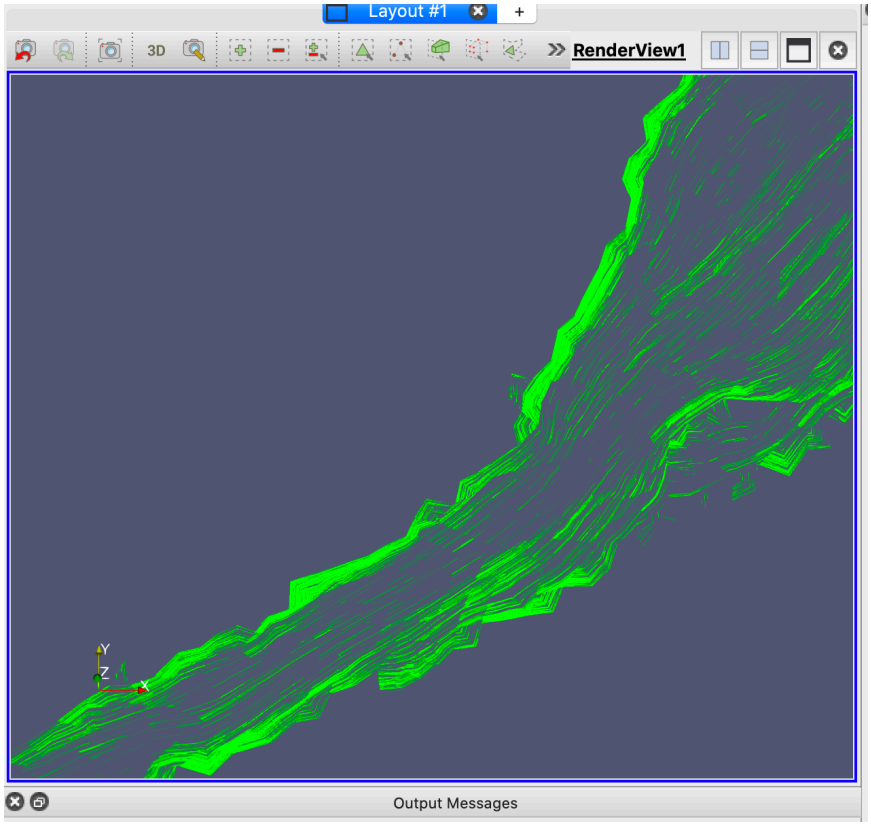


Figure 17.24 – StreamLines representation.

If you load a Paraview project '.pvsm' file, you have to deactivate the filters in the *Pipeline Browser* by clicking the open eyes so that only the '*.vtk' file layer is open.