OilFlow2D QGIS Reference Manual

Oil Spill Modeling for Land and Water

May 2026

Hydronia LLC

Last document modification date: May 2026.
Technical Support: support@hydronia.com
Web site: www.hydronia.com

Content

List of Figures

1

OilFlow2D

Oil spills on land and water.
OilFlow2D is Hydronia's 2D oil-spill simulator for environmental-assessment, contingency-planning, and spill-response studies. Built on the same 2D hydraulic core as RiverFlow2D, OilFlow2D adds oil-specific physics: multi-phase transport, shoreline interaction, and spill-containment features such as boom and skimming.

1.1 Plugin reference

Chapters follow the order of the QGIS plugin toolbar.

1. New Project / Scenario Tool - create projects and scenarios, configure initial layers.
2. Export Tools - hydrodynamic export plus the OilFlow2D .OILW export.
3. Results vs Time Maps - time-series, maximum, hazard, economic, and oil-on-land / oil-on-water maps.
4. Animation Tool - animate results and export images, videos, and KMZ files.
5. Cross Sections Tool - extract cross-section, hydrograph, and tabular results.
6. Tools - EPA-SWMM, TriMesh utilities, pipeline break model, HCA impact analysis, and project migration.
7. Hydronia Tools Context Menus - right-click hydrograph and tool menus attached to each plugin layer.
8. Appendix - Layer Attributes Reference - full attribute tables for every plugin layer.

2

New Project / Scenario Tool

Figure 2.1 – New Project Tool Icon for OilFlow2D

This chapter describes the tools related to creating and managing new projects and scenarios. Within the *New OilFlow2D Project* menu, the following tools are available:

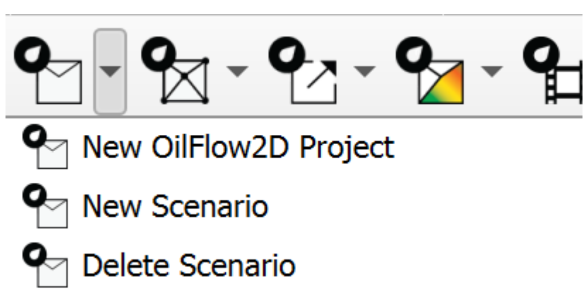


Figure 2.2 – New OilFlow2D Project Menu

2.1 New OilFlow2D Project

This tool initializes a new *OilFlow2D* project. It sets up the necessary directory structure, creates a QGIS project, and generates the required initial shapefile layers based on the components selected by the user. This establishes the foundation for defining model geometry, components, and simulation parameters within a multi-scenario context.

2.1.1 Dialog Window

The *New OilFlow2D Project* dialog allows the user to specify the project name, initial scenario name, project location, and select the components to be included in the initial setup.

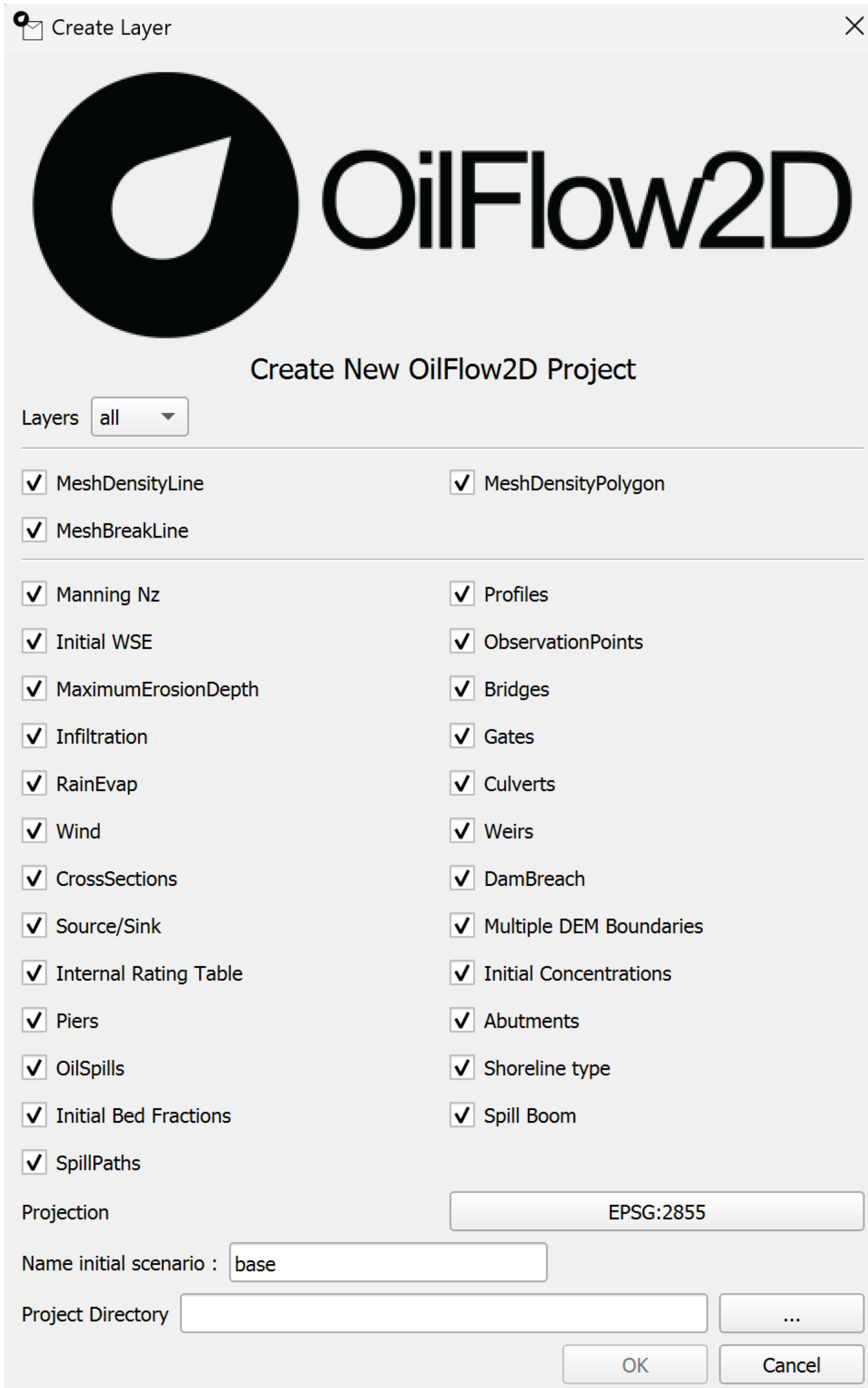


Figure 2.3 – Create New Project Dialog for OilFlow2D

2.1.2 Dialog Controls

The dialog contains the following controls:

Control	Type	Description
Set Current Project Projection	*Button*	Button to open the QGIS Project Properties dialog to select or verify the Coordinate Reference System (CRS) for the project.
Projection Display	*Read-only field*	Read-only field showing the selected project CRS (e.g., EPSG code).
Project Name	*Text input field*	Text input field to specify the name for the new project. This name will be used for the main project folder and the QGIS project file.
Name Initial Scenario	*Text input field*	Text input field to specify the name for the first scenario within the project. A subfolder with this name will be created.
Project Directory	*Text input field*	Text input field displaying the path to the directory where the project folder will be created.
Browse Button (...)	*Button*	Button next to Project Directory that opens a directory selection dialog.
Components Checkboxes	*Checkbox Group*	A series of checkboxes allowing the user to select which model components and data layers should be initialized. Options include common hydrodynamic features (MeshDensityLine, MeshBreakLine, Manning Nz, Initial WSE, Bridges, Gates, Culverts, Weirs, DamBreach, Sources/Sinks, etc.) and OilFlow2D-specific features like: SpillPaths, ShoreLine, OilSpills, SpillBooms, OilPipelines, OilPipeLineBoundCond, OilRetDepth, VegTrapp. (All are checked by default).
Ok	*Button*	Button to confirm the settings and create the new project.
Cancel	*Button*	Button to close the dialog without creating a project.

2.1.3 Initial Layers

The tool creates several initial shapefile layers within the shape subdirectory of the new scenario, based on the components selected in the dialog. The following table describes the layers created by default:

Layer Name	Type	Description
MeshDensityLine	Line	Defines lines along which mesh density is controlled.
MeshDensityPolygon	Polygon	Defines areas within which mesh density is controlled.
MeshBreakLine	Line	Enforces mesh element edges along these lines (e.g., levees, walls).
Domain Outline	Polygon	Defines the outer boundary of the 2D model domain.
Boundary Conditions	Line	Defines locations where boundary conditions (inflow, outflow, WSE) are applied.
Manning Nz	Polygon	Defines zones with specific anisotropic Manning's roughness coefficients.
Initial WSE	Polygon	Defines zones with specific initial water surface elevations.
Observation Points	Point	Defines locations where time series output results will be generated.
Sources Sinks	Point	Defines locations where flow is added (source) or removed (sink).
Wind	Polygon	Defines zones where wind shear stress is applied.
RainEvap	Polygon	Defines zones where rainfall and/or evaporation rates are applied.
Infiltration	Polygon	Defines zones with specific soil infiltration parameters.
Weirs	Line	Represents weir structures controlling flow.
Gates	Point	Represents gate structures controlling flow.
Culverts	Line	Represents culverts (pipes) conveying flow.
Bridges	Line	Represents the overall alignment or deck of bridge structures.
Piers	Polygon	Represents bridge pier obstructions.
Abutments	Polygon	Represents bridge abutment obstructions.
DamBreach	Polygon	Defines parameters for simulating a dam breach failure.
Spill Paths	Line	Defines preferential paths for oil spill movement.
ShoreLine	Line	Defines the shoreline boundary with specific oil interaction properties.
OilSpills	Point	Defines the location, type, and parameters of oil spill sources.
SpillBooms	Line	Represents physical booms used for oil containment.
OilPipelines	Line	Represents the alignment of oil pipelines.

Layer Name	Type	Description
OilPipeLineBoundCond	Point	Defines boundary conditions (e.g., leaks) for oil pipelines.
OilRetDepth	Polygon	Defines zones with specific oil retention depth characteristics.
VegTrapp	Polygon	Defines zones where vegetation characteristics affect oil trapping.
CrossSections	Line	Defines lines along which cross-sectional results are extracted.
Profiles	Line	Defines lines along which longitudinal profile results are extracted.

2.1.4 Layer Attributes

Please refer to the 8.1 section for detailed information on the attributes for the default layers created by the New Project tool.

2.1.5 Workflow

The typical workflow for using the New Project tool is as follows:

1. Open QGIS and click on the  icon to open the *New OilFlow2D Project* dialog.
2. The *Create New OilFlow2D Project* dialog appears.
3. By default, all common components are selected. Click the Layers dropdown menu and click None to deselect all layers if needed, then select the desired initial components.
4. Set the project's Coordinate Reference System (CRS) using the **Projection** button. Ensure a projected CRS is selected.
5. Enter a unique name for the **Name Initial Scenario** or leave the default.
6. Click the ... button to select a path for the **Project Directory**.
7. Click **Ok**.
8. The tool validates the inputs (directory, names, CRS).
9. If valid, it creates the project directory structure: Project Directory/Scenario Name/shape/.
10. Essential project metadata (e.g., marking it as a Multi-Scenario project, storing the current scenario name) is written into the QGIS project properties.
11. The newly created layers (based on selected components) are loaded into the QGIS Layers Panel.
12. Default styling and labeling are applied to the layers.
13. The project's CRS is set according to the user's selection.

2.1.6 Requirements

- A valid output directory must be selected where the user has write permissions.
- A unique Project Name must be provided. The tool will check if a folder with this name already exists in the selected Project Directory.
- A unique Scenario Name must be provided.
- A valid projected Coordinate Reference System (CRS) must be selected for the QGIS project. Geographic coordinate systems are not valid for hydrodynamic modeling because the unit of length is the arc degree.

2.1.7 Technical Details

- **Directory Structure**: The tool always creates a nested directory structure: Project Name/Scenario Name/shap
- **QGIS Project File**: Creating a new project does not automatically save a .qgz file. Project-specific settings are stored within this file. Ensure that the qgz file is outside of the scenario directory, and in the main project directory.

2.2 Create New Scenario

This tool allows users to create a new scenario within an existing *OilFlow2D* project. It essentially duplicates the directory structure and essential data files from a selected existing scenario (source scenario) to create a new, independent scenario ready for modification.

2.2.1 Dialog Window

The *New Scenario* dialog prompts the user for the name of the new scenario and allows selection of an existing scenario to use as a template.

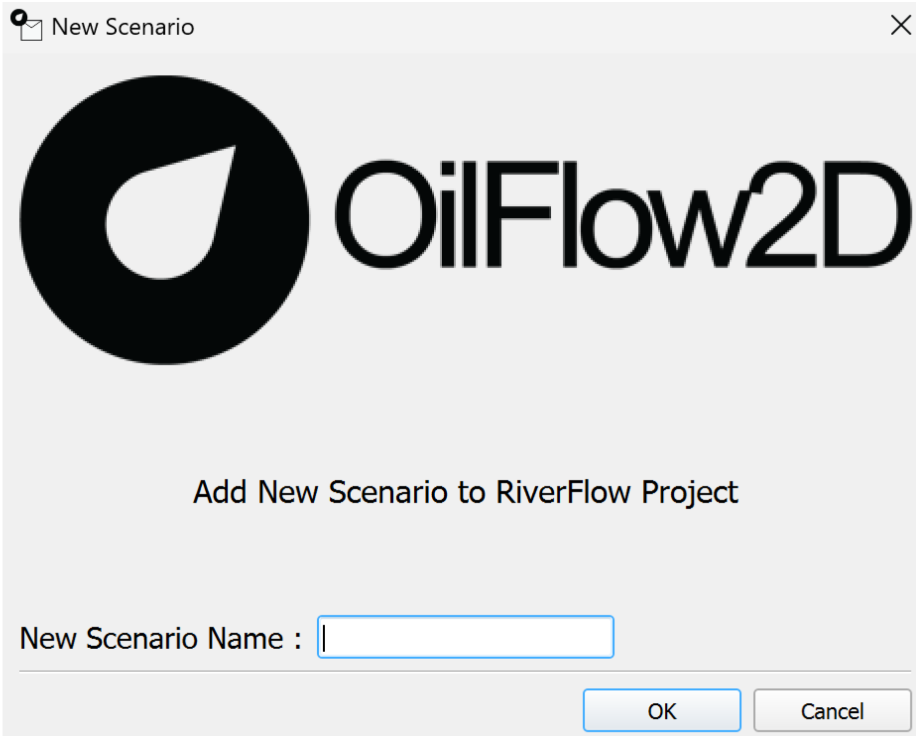


Figure 2.4 – The New Scenario dialog window for OilFlow2D.

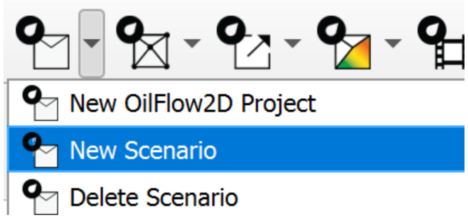
2.2.2 Dialog Controls

Control	Type	Description
Base Scenario	*Combo box*	Populated with the names of existing scenarios in the current project. The user selects the scenario to use as the template for the new one.
New Scenario Name	*Text input field*	Text input field where the user enters the desired name for the new scenario.
Ok	*Button*	Button to confirm the settings and create the new scenario.
Cancel	*Button*	Button to close the dialog without creating a new scenario.

2.2.3 Workflow

1. Open QGIS and click on the

    icon and click on the *New Scenario* menu item.



2. The *Add New New Scenario to Project* dialog appears.

3. Enter a unique name for the **New Scenario Name**.

4. Click **Ok**.

5. A new scenario directory is created at the same level as the existing scenario directories: project_folder \scenario_name\shape.

6. The tool determines which files need to be copied.

7. Essential project files (shapefile layers, mesh files, and related scenario files), are copied from the Base Scenario's and /shape directories to the corresponding locations in the New Scenario directory.

8. The plugin updates its internal list of scenarios (reflected in a scenario selection dropdown in the main interface).

9. The plugin switches the active scenario to the newly created one, unloading layers from the previous scenario and loading the corresponding layers from the new scenario's shape directory.

2.2.4 Requirements

- A *RiverFlow2D MS* project must be currently open in QGIS.
- The project must contain at least one existing scenario to serve as the base.
- A unique New Scenario Name must be provided. It cannot be empty or match an existing scenario name.
- The user must have write permissions in the project directory.

2.2.5 Technical Details

- **Scenario Identification**: Scenarios are identified by their folder names within the main project directory.
- **Data Duplication**: The core action is file-based copying. It duplicates the structure, content, and common scenario data files, including shapefile layer sidecar files. Does **not** copy any model output files or produced maps.
- **Active Scenario Switching**: Involves unloading the layers associated with the previous scenario and loading the corresponding layers from the new scenario's directory by updating their data source paths.

2.3 Delete Scenario

This tool removes an existing scenario from the *OilFlow2D* project. This action is irreversible but the files are not permanently deleted from the file system.

2.3.1 Dialog Window

The *Delete Scenario* dialog allows the user to select which existing scenario to remove.

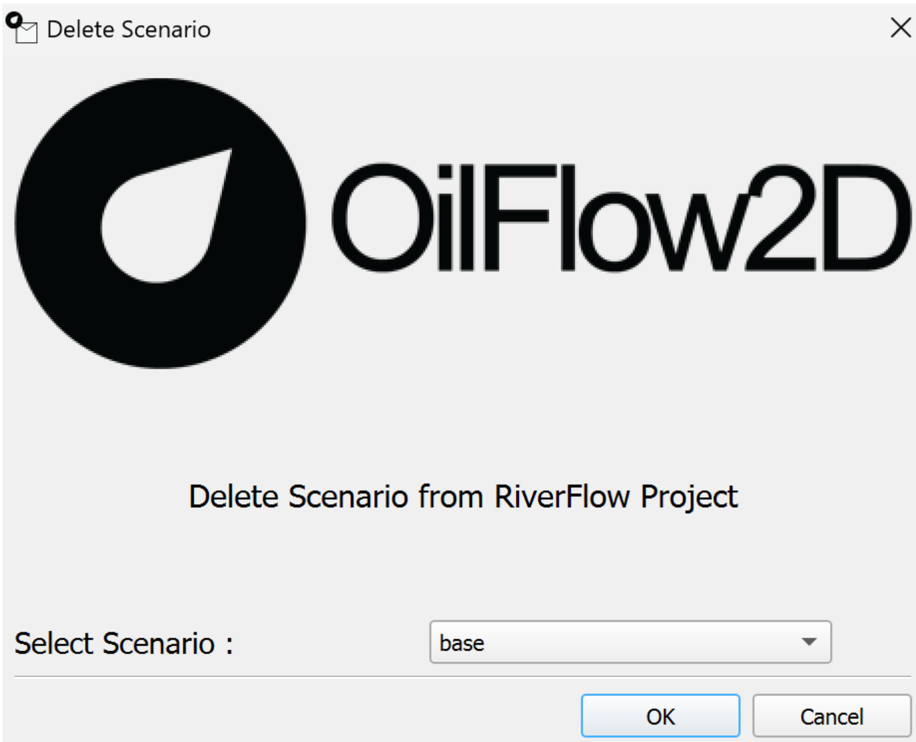


Figure 2.5 – The Delete Scenario dialog window.

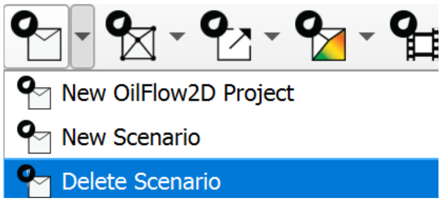
2.3.2 Dialog Controls

Control	Type	Description
Select Scenario	*Combo box*	Populated with the names of existing scenarios in the current project. The user selects the scenario to be deleted.
Ok	*Button*	Button to confirm the selection and remove the chosen scenario from the project.
Cancel	*Button*	Button to close the dialog without removing any scenario.

2.3.3 Workflow

1. Ensure a *OilFlow2D* project is open and active.

2. Activate the tool, click on the  icon and click on the *Delete Scenario* menu item.



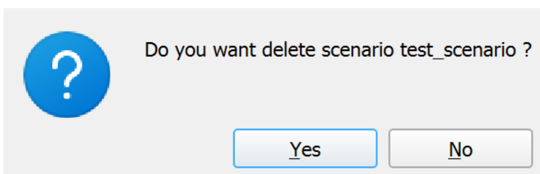
3. The *Delete Scenario from OilFlow2D Project* dialog appears.

4. The **Select Scenario** dropdown list is populated with all existing scenarios.

5. Select the scenario you wish to remove from the dropdown list.

6. Click **Ok**.

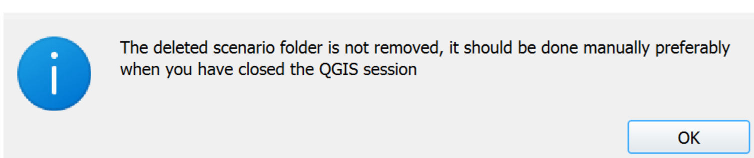
7. The user will receive a confirmation message asking them to confirm the removal.



8. Click **Ok** again to confirm the removal.

9. The tool identifies the directory path corresponding to the selected scenario, then removes the scenario from the project.

10. The user will receive a message indicating that the scenario has been removed from the project.



11. As the message indicates, this action is not permanently deleting files from the file system. If the user chooses, they can manually delete the entire directory in File Explorer. This will make it unrecoverable.

12. The plugin updates its internal list of scenarios, removing the deleted one (this is reflected in the scenario selection dropdown in the main interface).

13. Save your changes to the project to ensure that the changes are saved and no leftover artifacts are left behind.

2.3.4 Requirements

- A *RiverFlow2D MS* project must be currently open in QGIS.
- The project must contain at least two scenarios, as the currently active scenario cannot be deleted.
- The user must select a scenario from the list.
- The plugin will not allow the user to delete a scenario if it is the only one remaining in the project.

2.3.5 Manual Recovery of Deleted Scenario

If you need to recover a scenario that was deleted from the project but did not delete the old scenario directory, you can create a new scenario and manually copy the files back in.

- Follow the steps in the 1.2 section to create a new scenario.
- Save the project.
- Close QGIS.
- In File Explorer, navigate to the old scenario directory and copy the contents, not the directory itself.
- Paste the files into the new scenario directory, overwriting the existing files.
- Open QGIS and load the project.
- The new scenario will be loaded with the copied files.

2.3.6 Technical Details

- **Files are left intact**: The scenario directory and associated .qlr file is left intact. The user must manually delete the directory and .qlr file.
- **Cannot reuse deleted scenario name**: Unless the user deletes the associated .qlr file and saves the project, the scenario name cannot be reused.
- **Interface Update**: The scenario selection dropdown in the main plugin interface is updated to reflect the removal of the scenario.
- **Post-Deletion Active Scenario**: The plugin automatically switches the active QGIS view to another existing scenario after the deletion is complete.

3

Export Tools

This chapter describes the export options available for the active model and how output files are configured.

3.1 Export Hydrodynamic OilFlow2D

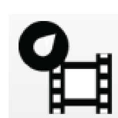


Figure 3.1 – Export Tool Icon for OilFlow2D

3.1.1 Dialog Window

The following dialog is the main interface for configuring and initiating the export process for the OilFlow2D model. It gathers information about input layers and specific components to be included in the export.

Figure 3.2 – Export Dialog for OilFlow2D

3.1.2 Dialog Controls

The following table describes the controls available in the Export dialog.

Control Name	Type	Description
Project Directory	*Text Field*	Displays the full path to the current scenario's output directory within the project. (Read-only)
Scenario Name	*Text Field*	Displays the name of the current scenario being exported. (Read-only)
DEM (Single Raster)	*Dropdown*	Select the single raster layer representing the bed topography/elevation. Enabled only if neither 'Using TriMesh Elevation' nor 'Get elevations from Multiple DEM Boundaries' is active. Populated with available raster layers.

Control Name	Type	Description
Using TriMesh Elevation (not resampling elevations)	*Checkbox*	Enable to use elevation data directly from the 'TriMesh' layer vertices/nodes instead of resampling from a DEM raster. Disables DEM selection options. Automatically checked if mesh was generated using "Generate TriMesh with Elevation" tool.
Get elevations from Multiple DEM Boundaries	*Checkbox*	Enable to use elevation data based on boundaries defined in a 'MultipleDemBoundaries' layer. Disables the 'DEM (Single Raster)' dropdown. Requires the 'MultipleDemBoundaries' layer to be present.
Using Manning N Raster Layer	*Checkbox*	Enable to use Manning's n values sampled from a selected raster layer instead of a vector layer. Enables the 'Manning N Raster Layer' dropdown.
Manning N Raster Layer	*Dropdown*	Select the raster layer containing Manning's n values. Enabled only when 'Using Manning N Raster Layer' is checked. Populated with available raster layers.
Using Initial WSE Raster Layer	*Checkbox*	Enable to use initial water surface elevation (WSE) values sampled from a selected raster layer. Enables the 'Initial WSE Raster Layer' dropdown.
Initial WSE Raster Layer	*Dropdown*	Select the raster layer containing initial WSE values. Enabled only when 'Using Initial WSE Raster Layer' is checked. Populated with available raster layers.
Rewrite .DAT file	*Checkbox*	Enable to force regeneration and overwriting of the main RiverFlow2D control file ('.dat').
Use external routine to sample cell elevations	*Checkbox*	Enable to use the external executable ('ASCIISamplingC.exe') for sampling elevation values for mesh cells/centroids. This might be faster or handle large datasets differently than the internal QGIS methods.
OK	*Button*	Confirms the selections and initiates the export process based on the chosen options and detected layers.
Cancel	*Button*	Closes the dialog without exporting any files.

3.1.3 Workflow

The typical workflow for using the Export Files to RiverFlow2D tool is as follows:

1. Ensure all required input layers (e.g., 'TriMesh', 'Manning N'/'Nr'/'Nz', optional layers like 'Boundary Conditions', 'Weirs', 'Bridges', DEM rasters etc.) are loaded into the QGIS project and meet the requirements (See Section 2.1.4). Make sure the 'Domain Outline' layer, if present, is not in editing mode.
2. Activate the tool from the OilFlow2D plugin menu or toolbar. This will open the Export Files to OilFlow2D dialog (Figure 2.2).
3. Verify that the Project Directory and Scenario Name are correctly displayed, reflecting the current project and active scenario.
4. Click **OK** to start the export process, or alternately if you need to set additional parameters, review the **Options** group box.
5. (Optional) Configure if Manning's roughness data should be sourced from a raster layer.
6. (Optional) Select the checkbox to use a raster for Initial Water Surface Elevation.
7. (Optional) Decide whether to use the external elevation sampling routine and whether to force rewriting the '.dat' file.
8. Click **OK**.
9. The tool performs requirement checks (e.g., layer existence, CRS compatibility, empty layers).
10. If checks pass, the tool proceeds with the export. It creates a centroids layer, samples necessary data (elevation, roughness, initial WSE) based on the selected options, and exports various component files ('.TGates', '.TWeirs', '.TBridges', '.TDams', '.THydnet', '.OBC', etc.) based on the presence of corresponding layers in the QGIS project (e.g., 'Gates', 'Weirs', 'Bridges', 'DamBreach', 'Channels1D', 'Boundary Conditions', etc.).
11. Finally, it generates the primary model input files: the geometry/mesh file ('.fed') and the control data file ('.dat'). Other auxiliary files like '.plt' and '.qgisunits' may also be created.
12. All exported files are saved in a subdirectory named after the current scenario within the main project directory. A confirmation message will appear upon successful completion, or error messages will be shown if issues occur.

3.1.4 Requirements

Before using the Export tool, ensure the following requirements are met:

- A QGIS project must be loaded.
- The project must have a defined current scene (scenario). This determines the output subdirectory and file naming.
- A mesh layer named 'TriMesh' must be present and active in the layer panel. This provides the core mesh geometry.
- At least one Manning's roughness source must be available and active: either a vector layer named 'Manning N' or raster layers named 'Manning Nr' or 'Manning Nz'.

  - Having both 'Manning N' (vector) and 'Manning Nz' (raster) active simultaneously is not allowed.
  - Having both 'Manning N' (vector) and 'Manning Nr' (raster) active simultaneously is not allowed.

- All layers must **not** be in editing mode. Save any edits and toggle editing off for this layer before exporting.

- If the option 'Get elevations from Multiple DEM Boundaries' is to be used, a layer named 'MultipleDemBoundaries' must be present.

- All relevant input layers (mesh, DEMs, roughness sources, component layers) must have compatible Coordinate Reference Systems (CRS).

- Active input layers should not be empty (contain no features or valid data).

Failure to meet these requirements will likely result in error messages displayed in the QGIS message bar and prevent the export from completing.

3.1.5 Technical Details

- The primary output files generated are the model's mesh/geometry file ('.fed') and the main input control data file ('.dat'). A plot data file ('.plt') and a units file ('.qgisunits') are also typically generated.

- Numerous other component-specific input files are generated conditionally, based on the presence of layers with specific names in the QGIS project. These include: .dOut ('Domain Outline'), .TGates ('Gates'), .IRT ('InternalRatingTable'), .TWeirs ('Weirs'), .TDams ('DamBreach'), .TBridges ('Bridges'), .THydnet ('Channels1D'), .OBC/.OBCP ('Boundary Conditions'), .OBS ('Observation Points'), .source ('Sources'), .scour ('ScourFromPiers', 'ScourFromAbutment'), .lswmm ('LSWMM'), .MannN/.MannN2 ('Manning N' vector/raster), .Linf/.Lrain/.Wind/.initConc (Related infiltration, rain, wind, initial concentration layers).

- All exported files are saved within a subdirectory named after the current project scene (scenario name), located within the main project directory identified in the dialog.

- A temporary centroids layer is created during the process using centroide_layer().

3.2 Export OILW File

This dialog allows the user to configure and export the necessary input file for running oil weathering and transport simulations within OilFlow2D.

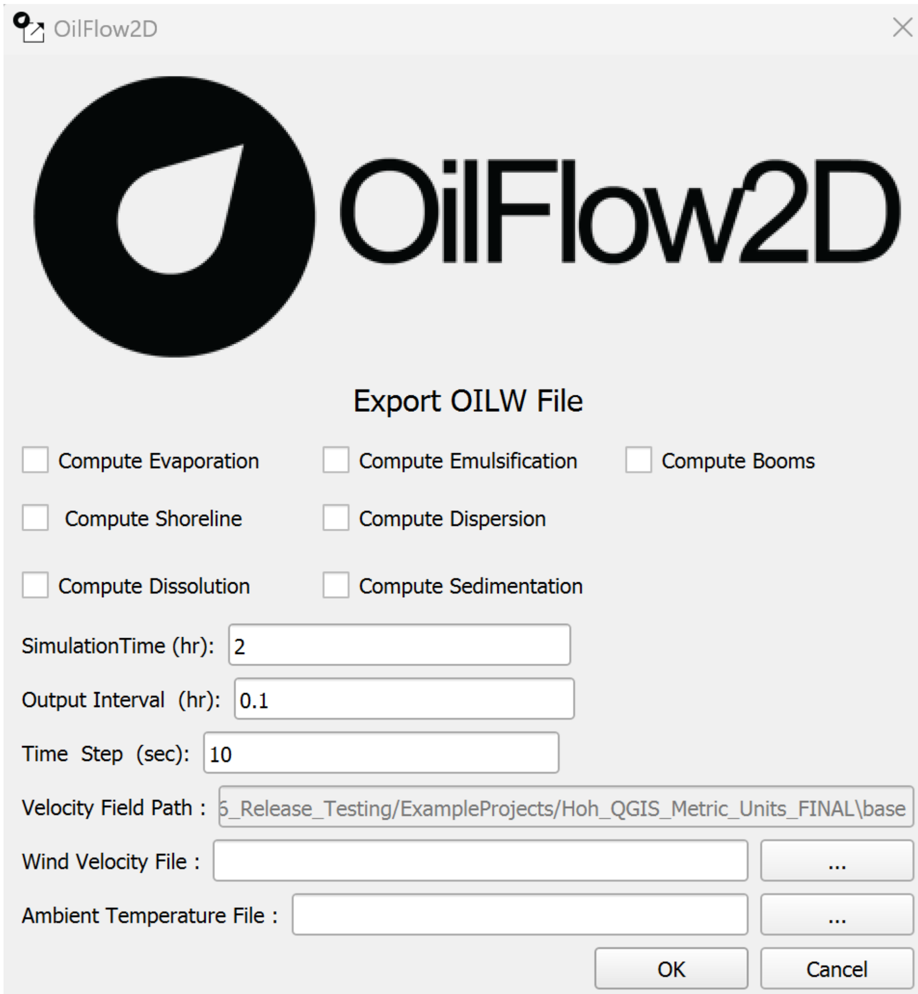


Figure 3.3 – Export OILW File Dialog.

3.2.1 Dialog Controls

The following table describes the controls available in the Export OILW File dialog.

Control Name	**Type**	**Description**
Compute Evaporation	*Checkbox*	Enable the evaporation process simulation.
Compute Emulsification	*Checkbox*	Enable the emulsification process simulation.
Compute Booms	*Checkbox*	Enable the simulation of containment booms. Requires a 'Booms' layer.
Compute Shoreline	*Checkbox*	Enable the shoreline interaction simulation.
Compute Dispersion	*Checkbox*	Enable the dispersion process simulation.
Compute Dissolution	*Checkbox*	Enable the dissolution process simulation.
Compute Sedimentation	*Checkbox*	Enable the sedimentation process simulation.
SimulationTime (hr)	*Text Field*	Enter the total simulation duration in hours.

Control Name	Type	Description
Output Interval (hr)	*Text Field*	Enter the time interval for writing output results in hours.
Time Step (sec)	*Text Field*	Enter the simulation time step in seconds.
Velocity Field Path	*Text Field (Read-only)*	Displays the path to the directory containing the velocity field data (typically the current scenario directory).
Wind Velocity File	*Text Field*	Enter the name of the file containing wind velocity data. Use the browse button to select.
… (Browse Wind)	*Button*	Opens a file dialog to select the Wind Velocity File.
Ambient Temperature File	*Text Field*	Enter the name of the file containing ambient temperature data. Use the browse button to select.
… (Browse Temp)	*Button*	Opens a file dialog to select the Ambient Temperature File.
OK	*Button*	Confirms the settings and initiates the export process.
Cancel	*Button*	Closes the dialog without exporting the file.

3.2.2 Workflow

The typical workflow for using the Export OILW File tool is as follows:

1. Ensure the required 'OilSpills' point layer and/or 'OilSpillsRelease' line layer are loaded into the QGIS project and meet the requirements (See Section 2.2.3). The attributes of these layers define the spill locations/sources and properties.
2. If simulating booms, ensure the required 'Booms' line layer is loaded.
3. Ensure the main OilFlow2D hydrodynamic simulation files (especially .dat) have already been exported for the current scenario using the main export tool (Section 2.1).
4. Activate the tool from the OilFlow2D plugin menu or toolbar (usually under a submenu like 'Modules' or similar).
5. The Export OILW File dialog opens (Figure 2.3).
6. Check or uncheck the various 'Compute…' options (Evaporation, Emulsification, Booms, Shoreline, Dispersion, Dissolution, Sedimentation) as required for the simulation.
7. Enter the desired 'SimulationTime', 'Output Interval', and 'Time Step'. These values may be pre-populated from a parameter file ('Oilwparamxport.txt') if it exists in the scenario directory.
8. Verify the 'Velocity Field Path' is correct (it should point to the current scenario directory).
9. Enter the filenames for the 'Wind Velocity File' and 'Ambient Temperature File' or use the browse buttons ('…') to select them. These files must exist in the scenario directory.

10. Click **OK**.
11. The tool performs requirement checks (e.g., layer existence, CRS compatibility, non-empty control fields, file existence within scenario folder).
12. If checks pass, the tool generates the Oil Weathering input file (texttt.OILW) in the current scenario directory using data from the dialog and the relevant spill/boom layer attributes.
13. If an 'OilSpillsRelease' layer is present, separate spill path files ('.pth') are generated for each feature using spillReleaseFile ().
14. If the 'Compute Booms' option is checked and a 'Booms' layer is present, a booms data file (texttt.booms) is generated using spillBoomFile().
15. The tool then reads the existing .dat file for the scenario, updates a specific flag to indicate oil spill simulation is active (flag set to 3), and rewrites the .dat file.
16. An oil parameters export file ('Oilwparamxport.txt') is created or updated in the scenario directory to store the dialog settings.
17. The OilFlow2D executable (specifically RiverFlow2DDIP.exe) is optionally launched automatically using the generated files.
18. A confirmation or error messages will be shown in the QGIS message bar.

3.2.3 Requirements

Before using the Export OILW File tool, ensure the following requirements are met:

- A QGIS project must be loaded with a defined current scene (scenario).
- The main OilFlow2D hydrodynamic export (Section 2.1) must have been run previously for the current scenario, creating the necessary base files like .dat and .OUTFILES.
- At least one oil source layer must be present and active: 'OilSpills' (point layer for instantaneous spills) or 'OilSpillsRelease' (line layer for continuous release paths). These layers must contain features with the specific attributes required by the fileOilSpills () function (e.g., density, viscosity, timing, dispersion, evaporation parameters, etc.).
- If the 'Compute Booms' option (texttcBox_booms) is checked, a line vector layer named 'Booms' must be present and active. It needs appropriate attributes for boom properties.
- The source layers ('OilSpills', 'OilSpillsRelease', 'Booms') must not be empty if they are present.
- All layers involved must have compatible Coordinate Reference Systems (CRS). The tool performs a check.
- The required input fields in the dialog (SimulationTime, Output Interval, Wind Velocity File, Ambient Temperature File) must not be empty.
- The selected Wind Velocity File and Ambient Temperature File must reside within the current scenario directory.

Failure to meet these requirements will result in error messages and prevent the export from completing.

3.2.4 Technical Details

- The primary output file generated by this tool is the Oil Weathering input file (texttt.OILW).
- If continuous spills are defined via an 'OilSpillsRelease' layer, corresponding path files (texttt.pth) are also generated for each release source.
- If booms are enabled and a 'Booms' layer exists, a texttt.booms file is generated.
- The tool modifies the existing main OilFlow2D control data file (texttt.dat) by reading it (texttt-treadDAT()) and rewriting it (textttrewriteDAT()) with a specific flag (index 15) set to 3 to enable the oil spill module in the OilFlow2D engine.
- The .OILW file is generated by the fileOilSpills () function, which reads control data from the dialog and detailed spill information from the attributes of the 'OilSpills' and/or 'OilSpillsRelease' layer features. It includes sections for [Trajectory], [Evaporation], [Emulsification], [Dissolution], [Dispersion], [Sedimentation], and [Shoreline] parameters based on the attributes and dialog settings.
- A parameter file ('Oilwparamxport.txt') is saved in the scenario directory by the oilwParameterFile() function (called from tools.py) to persist the dialog settings (time, output interval, wind file, temp file).
- The tool attempts to automatically launch the OilFlow2D executable (specifically RiverFlow2DDIP.exe, which handles oil spills) using the generated .OILW and modified .dat files via the callRiverFlow() function.

4 Results vs Time Mapping Tools

The plugin provides a comprehensive set of mapping tools for visualizing and analyzing simulation results. These tools allow users to create, display, and export various types of maps representing hydraulic parameters, sediment or tailings concentrations, time of arrival or to depth, hazard assessments, economic evaluations.

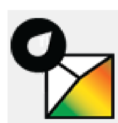


Figure 4.1 – Results vs Time Maps Icon for OilFlow2D

Users have the following options available in the mapping tool menu:

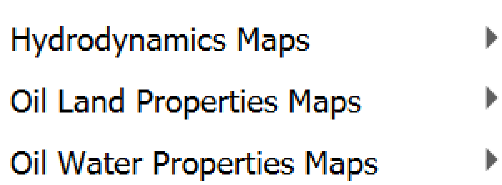


Figure 4.2 – Results vs Time Maps Menu for OilFlow2D

The mapping tools in the plugin enable users to visualize simulation results in both cell-based and raster formats. The plugin provides specialized tools for different types of analyses, including:

- Time-dependent maps for various hydraulic parameters (Results vs Time Maps)
- Maximum value maps showing peak conditions (Maximum Result Maps)
- Hazard intensity maps for risk assessment (Hazard Intensity Maps)
- Time-depth maps for flood progression analysis (Time-to-Depth Maps)

- Element concentration or fractions maps for pollutant/sediment/mud transport (Element Concentration Maps)
- Hydro-Economic Evaluation of Flood (HEEF) maps (Hydro-Economic Evaluation of Flood Map)

4.1 Results vs Time Maps

The Results vs Time Maps tool allows users to create and visualize time-dependent maps of various hydraulic parameters from model simulations.

4.1.1 Dialog Window

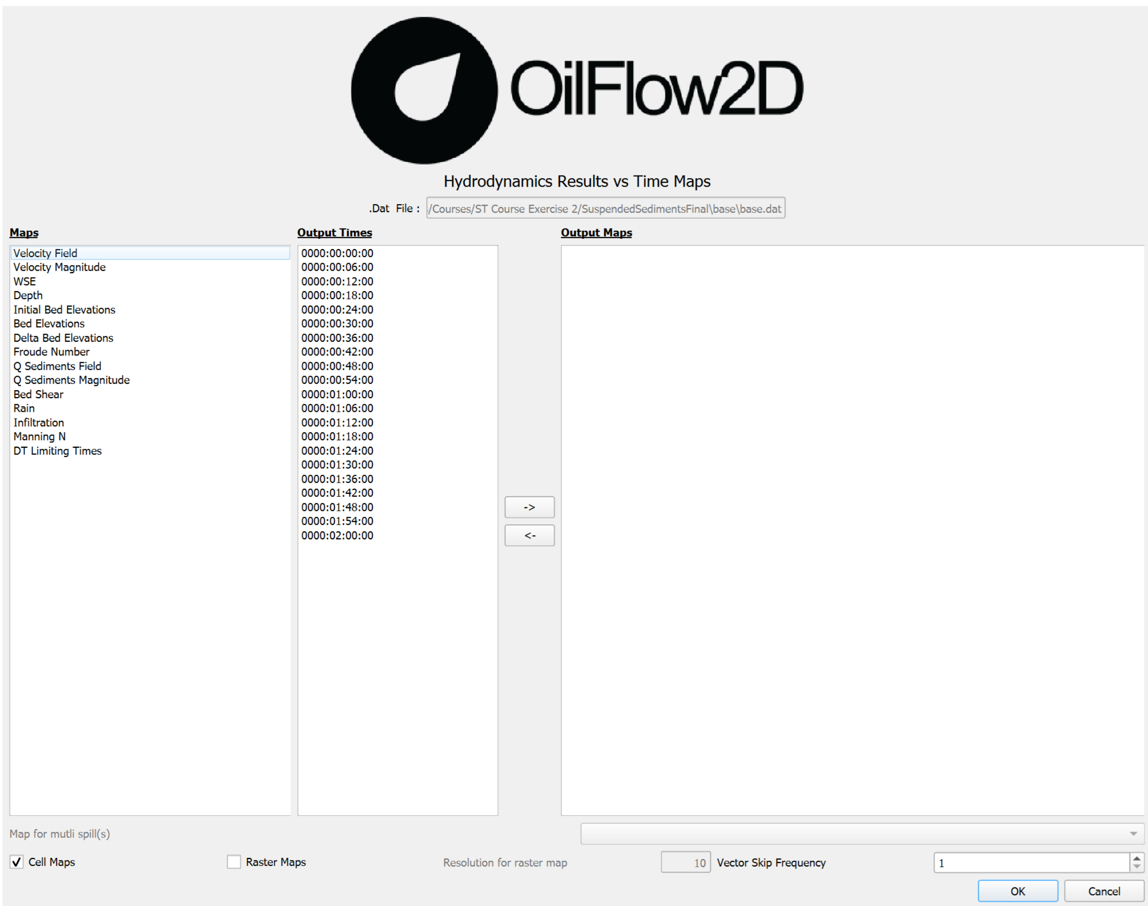


Figure 4.3 – Results vs Time Maps Dialog for OilFlow2D

4.1.2 Dialog Controls

Output File List	*text field*	Displays the path to the simulation results file (read-only).
Maps	*list view*	Displays available map types that can be selected for visualization, such as Velocity Field, Velocity Magnitude, WSE, Depth, etc.
Output Times	*list view*	Displays available simulation output times that can be selected for visualization.
Arrow Buttons (→, ←)	*buttons*	Transfer selected maps and times to/from the Output Maps list.
Output Maps	*list view*	Shows the maps that will be created when OK is clicked.
Cell Maps	*checkbox*	Enables cell-based (vector polygon) map output.
Raster Maps	*checkbox*	Enables raster map output.
Resolution for raster map	*text field*	Sets the resolution for raster interpolation, in projected units. Only enabled when “Raster Maps” is checked.
Vector Skip Frequency	*spinner*	Sets the number of vectors to skip between each vector in vector-type maps, controlling display density.
OK	*button*	Creates the selected maps and adds them to the QGIS project.
Cancel	*button*	Closes the dialog without making changes.

4.1.3 Results vs Time Map Types

The Results vs Time Maps tool can generate the following maps:

Velocity	m/s or ft/s	Vector field showing flow velocity direction and magnitude with arrows.
Velocity Magnitude	m/s or ft/s	Scalar field showing the magnitude of flow velocity.
WSE	m or ft	Water surface elevation at each computational cell.
Depth	m or ft	Water depth at each computational cell.
Initial Bed Elevations	m or ft	Initial bed elevation at each computational cell (available when sediment transport module is active).
Bed Elevations	m or ft	Current bed elevation at each computational cell (available when sediment transport module is active).
Delta Bed Elevations	m or ft	Change in bed elevation from initial conditions (available when sediment transport module is active).

Velocity	m/s or ft/s	Vector field showing flow velocity direction and magnitude with arrows.
Froude Number	dimensionless	Local Froude number at each computational cell.
Q Sediments Field	m^3/s or ft^3/s per unit width	Creates vector field from sediment transport components in columns 9-11, with arrows showing direction and relative magnitude. Available only with sediment transport module.
Q Sediments Magnitude	m^3/s or ft^3/s per unit width	Extracts sediment transport magnitude from column 11, applies Equal Interval classification with 9 classes. Available only with sediment transport module.
Bed Shear	Pa or lbf/ft^2	Bed shear stress at each computational cell (available when sediment transport module is active).
Rain	mm/hr or in/hr	Rainfall intensity at each computational cell (available when rainfall module is active).
Infiltration	mm/hr or in/hr	Infiltration rate at each computational cell (available when infiltration module is active).
Manning N	$s{\cdot}m^{-1/3}$ or $s{\cdot}ft^{-1/3}$	Manning's roughness coefficient at each computational cell.
DT Limiting Times	dimensionless	Time step limiting factors at each computational cell.

4.1.4 Workflow

To generate maps using the Results vs Time Maps tool, follow these steps:

1. Ensure you have completed a simulation and have output files available.

2. From the Results vs Time Maps menu, click on "*Results vs Time Maps*"
.

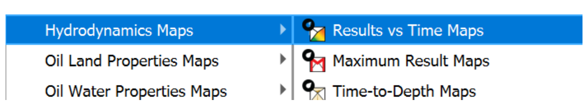


Figure 4.4 – Results vs Time Maps Menu for OilFlow2D

3. In the dialog that appears, the current scenario directory should be automatically selected. If not, browse to the appropriate directory containing your simulation results.

4. Select one or more map types from the list by checking the corresponding map names:

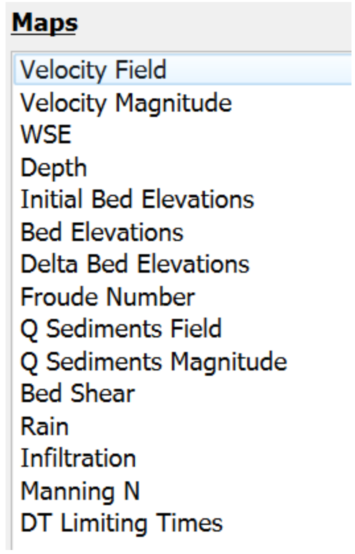


Figure 4.5 – Results vs Time Map List

5. Select the output times you want to visualize. Hold down the Control key to select multiple times.

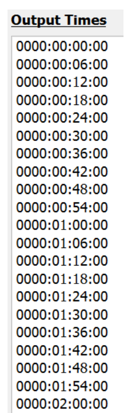
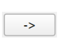
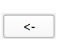


Figure 4.6 – Results vs Time Map Times

6. Transfer the selected maps and times to the *Output Maps* list by clicking the right arrow button 

.

7. If you wish to remove a map or time from the *Output Maps* list, select it and click the left arrow button 

.

8. Choose the output format: Check *Cell Maps* to create polygon-based maps or check *Raster Maps* to create grid-based maps with interpolation (or both!).

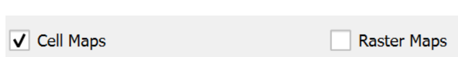


Figure 4.7 – Results vs Time Map Output Format Options

9. If *Raster Maps* is selected, specify the raster resolution in the projected units (ft or m).

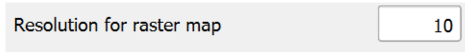


Figure 4.8 – Results vs Time Map Raster Resolution

10. If a vector map such as *Velocity Field* is selected, you can specify the *Vector Skip Frequency* to control the density of the vectors.

Figure 4.9 – Results vs Time Map Vector Skip Frequency

11. Click “OK” to generate the maps.
12. The generated maps will be added to the QGIS layer panel under the “OUTPUT_RESULTS” group.
13. You can now manipulate these layers like any other QGIS layer, adjusting styling, transparency, etc.

4.1.5 Requirements

Active Project	An active project must be loaded in QGIS.
Simulation Results	A completed simulation with output files in the project’s scenario directory.
Required Data Files	The following files must be present in the scenario directory:

4.1.6 Technical Information

The following table provides technical details on how each map type is processed and visualized. Legend values are displayed in metric units (m, m/s, m^2/s, etc.) when the project is in metric mode, and English units (ft, ft/s, ft^2/s, etc.) when in English mode. The unit system is automatically detected based on the QGIS project settings.

Velocity Field	Creates vector field visualization from velocity components (v_x, v_y) in columns 0-1, with arrow size scaled by $v_{mag} = \sqrt{v_x^2 + v_y^2}$. Applies vector skip frequency for density control and uses graduated color ramp from blue to red based on magnitude.
Velocity Magnitude	Extracts scalar velocity magnitude (m/s or ft/s) from column 2, applies Quantile classification with 9 classes and blue-to-red color ramp. Excludes values below 0.01 m/s (or ft/s).
WSE (Water Surface Elevation)	Reads WSE values (m or ft) from column 3, applies Equal Interval classification with 9 classes. Handles no-data values (-9999.0) by using minimum WSE as lower bound.

Velocity Field	Creates vector field visualization from velocity components (v_x, v_y) in columns 0-1, with arrow size scaled by $v_{mag} = \sqrt{v_x^2 + v_y^2}$. Applies vector skip frequency for density control and uses graduated color ramp from blue to red based on magnitude.
Depth	Processes depth values (m or ft) from column 4 using Equal Interval classification with 9 classes and blue color ramp. Excludes depths below 0.01 m (or ft).
Initial Bed Elevations	Reads values (m or ft) from column 5, applies Equal Interval classification with 8 classes and specialized topographic color ramp. Handles no-data values (-9999.0) by excluding them.
Bed Elevations	Processes current bed elevation (m or ft) from column 6 using Equal Interval classification with 8 classes and topographic color ramp. Available only with sediment transport module.
Delta Bed Elevations	Calculates bed elevation changes (m or ft) from column 7, uses Equal Interval classification with 8 classes and diverging color ramp centered on zero. Available only with sediment transport module.
Froude Number	Processes Froude numbers from column 8 using Equal Interval classification with 9 classes. Color ramp highlights subcritical $(Fr < 1)$ and supercritical $(Fr > 1)$ flow regions.
Q Sediments Field	Creates vector field from sediment transport components in columns 9-11, with arrows showing direction and relative magnitude. Vector components (vx, vy) are in sediment transport units (m^3/s or ft^3/s per unit width), and the magnitude (vm) is in column 11. Available only with sediment transport module.
Q Sediments Magnitude	Extracts sediment transport magnitude (m^3/s or ft^3/s per unit width) from column 11, applies Equal Interval classification with 9 classes. Available only with sediment transport module.
Bed Shear	Processes bed shear stress (N/m^2 or lb/ft^2) from column 12 using Equal Interval classification with 9 classes. Available only with sediment transport module.
Rain	Reads rainfall intensity (mm/hr or in/hr) from column 13, applies Equal Interval classification with 9 classes. Available only with rainfall module.
Infiltration	Processes infiltration rates (mm/hr or in/hr) from column 14 using Equal Interval classification with 9 classes. Available only with infiltration module.
Manning N	Displays Manning's roughness coefficient from column 15 using Equal Interval classification with 9 classes to show spatial distribution of roughness values.
DT Limiting Times	Shows time step limiting factors from column 16 using Equal Interval classification with 5 classes. Vector output only, excludes zero values.

4.2 Maximum Result Maps

The Maximum Result Maps tool generates maps showing the maximum values reached during a simulation for various hydraulic parameters.

4.2.1 Dialog Window

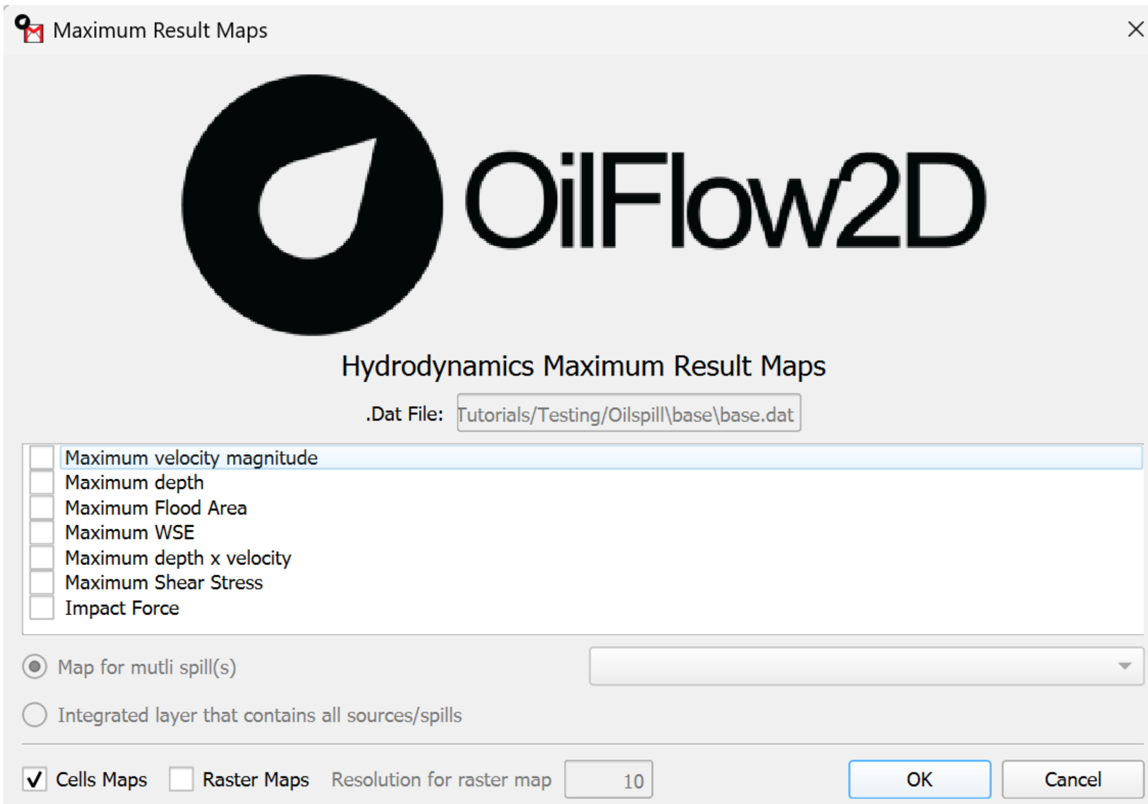


Figure 4.10 – Hydrodynamic Maximum Result Maps Dialog for OilFlow2D

You can reach this dialog by clicking on the *Maximum Result Maps* button under the *Hydrodynamic Maps* menu in the *Results vs Time Maps* tool:

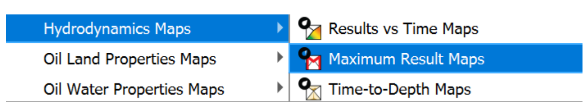


Figure 4.11 – Maximum Result Maps Menu for OilFlow2D

4.2.2 Dialog Controls

Maximum Value File	*text field*	Displays the path to the simulation results file (read-only).	
Value Selector	*list widget*	List of available oil property maximum value map types that can be selected for visualization.	
Map for multi spill(s)	*radio button	dropdown*	Option to create maps for a specific spill selected from the dropdown when multiple spills are present in the simulation.
Integrated layer that contains all sources/spills	*radio button*	Option to create a single map that integrates data from all spills in the simulation.	
Cells Maps	*checkbox*	Enables cell-based (vector polygon) map output. Checked by default.	
Raster Maps	*checkbox*	Enables raster map output.	

Maximum Value File	*text field*	Displays the path to the simulation results file (read-only).
Resolution for raster map	*text field*	Sets the resolution for raster interpolation, in projected units. Only enabled when "Raster Maps" is checked. Default value is 10.
OK	*button*	Creates the selected maximum value maps and adds them to the QGIS project.
Cancel	*button*	Closes the dialog without making changes.

4.2.3 Maximum Value Map Types

The Maximum Result Maps tool for OilFlow2D can generate maximum value maps for the following parameters:

Maximum Oil Temperature	°C or °F	Maximum oil temperature reached at each location during the simulation.
Maximum Oil Density	kg/m^3 or lb/ft^3	Maximum oil density reached at each location during the simulation.
Maximum Oil Viscosity	Pa·s or $lb{\cdot}s/ft^2$	Maximum oil viscosity reached at each location during the simulation.
Maximum Oil Yield Stress	Pa or lb/ft^2	Maximum oil yield stress reached at each location during the simulation.

Additionally, when accessing the Hydrodynamic Maximum Result Maps, these map types are available:

Maximum Velocity Magnitude	m/s or ft/s	Maximum flow velocity magnitude reached at each location during the simulation.
Maximum Depth	m or ft	Maximum water depth reached at each location during the simulation.
Maximum Flood Area	n/a	Polygon outlining the maximum extent of flooding throughout the simulation.
Maximum WSE	m or ft	Maximum water surface elevation reached at each location during the simulation.
Maximum Depth x Velocity	m^2/s or ft^2/s	Maximum product of depth and velocity (DV) reached at each location, used for hazard assessment.
Maximum Shear Stress	N/m^2 or lbf/ft^2	Maximum bed shear stress reached at each location during the simulation.
Impact Force	N/m^2 or lbf/ft^2	Maximum impact force reached at each location, calculated from flow momentum and density.

4.2.4 Workflow

To generate maps using the Maximum Result Maps tool, follow these steps:

1. Ensure you have completed a model simulation with the *Output maximum files* option enabled and have output files available, the *Output maximum files* option is available in the *Control Data* tab of the Hydronia Data Input Program.

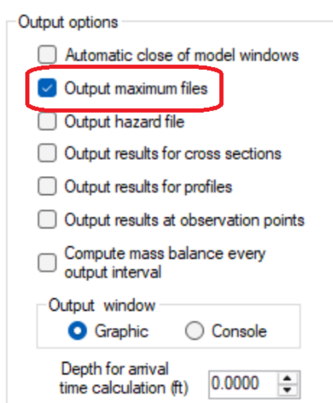


Figure 4.12 – Enabling Maximum file output in the Hydronia Data Input Program

2. From the *Results vs Time Maps* menu, click on *Hydrodynamic Maps* then *Maximum Result Maps*. .

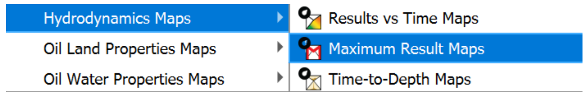


Figure 4.13 – Maximum Result Maps Menu for OilFlow2D

3. In the dialog that appears, the current scenario directory should be automatically selected. If not, browse to the appropriate directory containing your simulation results.

4. Select one or more map types from the list by checking the corresponding checkboxes:

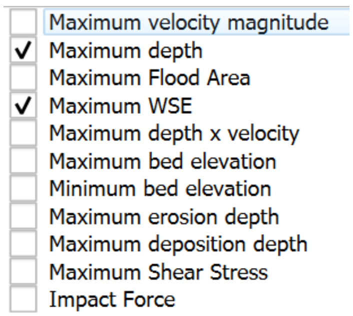


Figure 4.14 – Maximum Result Maps Checkboxes

5. Choose the output format:

   - Check *Cell Maps* to create polygon-based maps (triangular mesh).
   - Check *Raster Maps* to create grid-based maps with interpolation.
   - If *Raster Maps* is selected, specify the raster resolution in project units (ft or m).

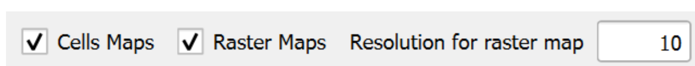


Figure 4.15 – Maximum Result Maps Output Options

6. Click “OK” to generate the map(s).

7. The generated map(s) will be added to the QGIS layer panel under the “OUTPUT_RESULTS” group.

The map(s) will show maximum values reached during the entire simulation period, regardless of when those maximum values occurred.

4.2.5 Requirements

Active Project	An active project must be loaded in QGIS.
Simulation Results	A completed model simulation with the *Output maximum files* option enabled and have output files available, the *Output maximum files* option is available in the *Control Data* tab of the Hydronia Data Input Program.
Required Data Files	The following files must be present in the scenario directory:

4.2.6 Technical Information

The following table provides technical details on how each map type is processed and visualized:

Velocity Magnitude	The tool scans all time steps in the cell_*_max.textout files and identifies the maximum velocity magnitude value for each computational cell. Graduated symbology is applied using the Quantile classification method with a color ramp from blue (low) to red (high). Values below 0.01 m/s are excluded.
Depth	The tool analyzes all time steps and extracts the maximum water depth value for each computational cell. Equal Interval classification is used for graduated symbology, providing a consistent color scale from light blue (shallow) to dark blue (deep). Values below 0.01 m are excluded.
Maximum Flood Area	Creates a polygon by dissolving all cells where maximum depth exceeds a threshold value (typically 0.01 m). The total flooded area and volume are calculated and stored as polygon attributes.
WSE (Water Surface Elevation)	Maximum water surface elevation is determined for each cell. Equal Interval classification is used with a specialized color ramp. No-data values (-9999.0) are handled by using the minimum valid WSE as the lower bound.
Depth x Velocity	Maximum product of depth and velocity (DV) is calculated for each cell. Equal Interval classification is used with a color ramp highlighting hazard levels based on standard thresholds.
Maximum Shear Stress	Maximum bed shear stress is extracted from sediment transport calculations. Equal Interval classification is used with a color ramp from blue (low) to red (high). Only available with sediment transport module.
Impact Force	Maximum impact force is calculated from flow momentum and density. Equal Interval classification is used with a color ramp highlighting areas of significant impact potential.
DT Limiting Times	Shows time step limiting factors using 5-class Equal Interval classification. Vector output only, with zero values excluded. Used to identify areas controlling simulation stability.

Velocity Magnitude	The tool scans all time steps in the cell_*_max.textout files and identifies the maximum velocity magnitude value for each computational cell. Graduated symbology is applied using the Quantile classification method with a color ramp from blue (low) to red (high). Values below 0.01 m/s are excluded.
Maximum Bed Elevation	Maximum bed elevation is tracked during sediment transport simulations. Equal Interval classification with topographic color ramp. Only available with sediment transport module.
Minimum Bed Elevation	Minimum bed elevation is tracked during sediment transport simulations. Equal Interval classification with topographic color ramp. Only available with sediment transport module.
Maximum Erosion Depth	Maximum erosion depth is calculated from bed elevation changes. Equal Interval classification with specialized color ramp. Only available with sediment transport module.
Maximum Deposition Depth	Maximum deposition depth is calculated from bed elevation changes. Equal Interval classification with specialized color ramp. Only available with sediment transport module.

4.3 Time-to-Depth Maps

The Time-to-Depth Maps tool creates maps that analyze the temporal aspects of flooding by correlating water depths with specific times during a simulation. This tool is particularly useful for understanding flood dynamics, such as inundation timing, duration of flooding, and wave propagation. Users can generate maps showing when specific water depths are reached, how long areas remain underwater, and when flood waves first arrive at different locations.

4.3.1 Dialog Window

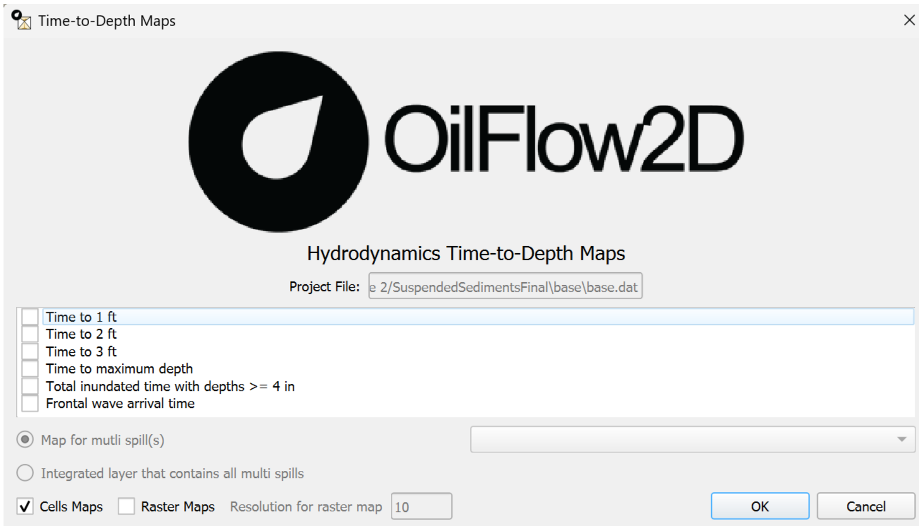


Figure 4.16 – Time-Depth Dialog for OilFlow2D

4.3.2 Dialog Controls

Project File	*text field*	Displays the path to the simulation results file (read-only).
Map Type List	*checkbox list*	List of available time-to-depth map types that can be selected.
Cells Maps	*checkbox*	Enables vector (cell-based) map output.
Raster Maps	*checkbox*	Enables raster map output.
Resolution for raster map	*textbox*	Sets the resolution for raster interpolation, in meters. Only enabled when Raster Maps is checked.
OK	*button*	Creates the selected time-to-depth maps and adds them to the QGIS project.
Cancel	*button*	Closes the dialog without making changes.

4.3.3 Time-Depth Analysis Maps

The Time-to-Depth Maps tool supports the following map types:

Time to 1 ft (0.30 m)	hours	Shows the time when water depth first reaches 1 ft (0.30 m) at each location.
Time to 2 ft (0.50 m)	hours	Shows the time when water depth first reaches 2 ft (0.50 m) at each location.

Time to 1 ft (0.30 m)	hours	Shows the time when water depth first reaches 1 ft (0.30 m) at each location.
Time to 3 ft (1 m)	hours	Shows the time when water depth first reaches 3 ft (1 m) at each location.
Time to maximum depth	hours	Shows the time when water depth reaches its maximum value at each location.
Total inundated time with depths $>=$ 4 in (0.1 m)	hours	Shows the total duration for which water depth exceeds 4 in (0.1 m) at each location.
Frontal wave arrival time	hours	Shows the time when water first arrives at each location, typically using a very small depth threshold.

4.3.4 Workflow

To generate time-to-depth maps, follow these steps:

1. Ensure you have completed a RiverFlow2D simulation and have output files available.
2. From the *Results vs Time Maps* menu, navigate to the *Hydrodynamic Maps* submenu, then click on *Time-to-Depth Maps* .

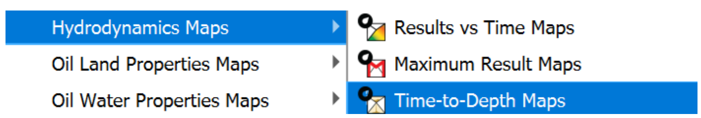


Figure 4.17 – Time-to-Depth Maps Menu for OilFlow2D

3. In the dialog that appears, the current scneario directory and project file information should be automatically populated.
4. Select one or more map types from the list by checking the corresponding checkboxes:
   - Time to 1 ft (0.30 m) - Shows when water depth first reaches 1 ft/0.30 m
   - Time to 2 ft (0.50 m) - Shows when water depth first reaches 2 ft/0.50 m
   - Time to 3 ft (1 m) - Shows when water depth first reaches 3 ft/1 m
   - Time to maximum depth - Shows when maximum water depth occurs
   - Total inundated time with depths $>=$ 4 in (0.1 m) - Shows duration above threshold
   - Frontal wave arrival time - Shows when water first arrives
5. Choose the output format:
   - Check “Cells Maps” to create polygon-based maps (cell-centered).
   - Check “Raster Maps” to create grid-based maps with interpolation.

- If "Raster Maps" is selected, specify the raster resolution in meters.

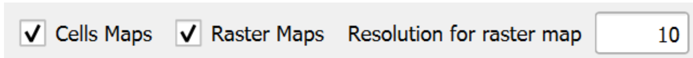


Figure 4.18 – Time-to-Depth Maps Output Options

6. Click "OK" to generate the selected time-to-depth maps.
7. The generated maps will be added to the QGIS layer panel under the "OUTPUT_RESULTS" group.
8. The maps will display time values (in hours) based on the selected map types.

4.3.5 Requirements

Active Project	An active project must be loaded in QGIS.
Simulation Results	A completed simulation with output files in the project's scenario directory.
Required Data Files	The following files must be present in the scenario directory:

4.3.6 Technical Information

The Time-to-Depth Maps tool processes time-series data from model simulations and generates maps based on the following calculations:

Time to 1 ft (0.30 m)	For each cell, the algorithm analyzes the time series of water depths and identifies the first time step where $depth \geq 0.30$ m (or 1 ft). The formula is: $t_{0.30m} = min(t)$ where $depth(t) \geq 0.30$ m. If the threshold is never reached, the cell is excluded from the map.
Time to 2 ft (0.50 m)	For each cell, the algorithm analyzes the time series of water depths and identifies the first time step where $depth \geq 0.50$ m (or 2 ft). The formula is: $t_{0.50m} = min(t)$ where $depth(t) \geq 0.50$ m. If the threshold is never reached, the cell is excluded from the map.
Time to 3 ft (1 m)	For each cell, the algorithm analyzes the time series of water depths and identifies the first time step where $depth \geq 1.0$ m (or 3 ft). The formula is: $t_{1.0m} = min(t)$ where $depth(t) \geq 1.0$ m. If the threshold is never reached, the cell is excluded from the map.
Time to maximum depth	For each cell, the algorithm finds the time step at which the maximum water depth occurs during the entire simulation. The formula is: $t_{max} = t$ where $depth(t) = max(depth)$ for all time steps.

Time to 1 ft (0.30 m)	For each cell, the algorithm analyzes the time series of water depths and identifies the first time step where $depth \geq 0.30$ m (or 1 ft). The formula is: $t_{0.30m} = min(t)$ where $depth(t) \geq 0.30$ m. If the threshold is never reached, the cell is excluded from the map.
Total inundated time with depths $>= 4$ in (0.1 m)	For each cell, the algorithm calculates the cumulative time for which the water depth exceeds 0.1 m (or 4 in). The formula is: $t_{total} = \sum \Delta t$ for all time steps where $depth(t) \geq 0.1$ m.
Frontal wave arrival time	For each cell, the algorithm identifies the earliest time step at which water is present (typically using a very small depth threshold). The formula is: $t_{arrival} = min(t)$ where $depth(t) > 0$ (or some small threshold).

4.4 Hazard Intensity Maps

The Hazard Intensity Maps tool allows users to create maps that classify flood hazards based on various international standards for assessing flood risk to people, vehicles, and structures.

4.4.1 Dialog Window

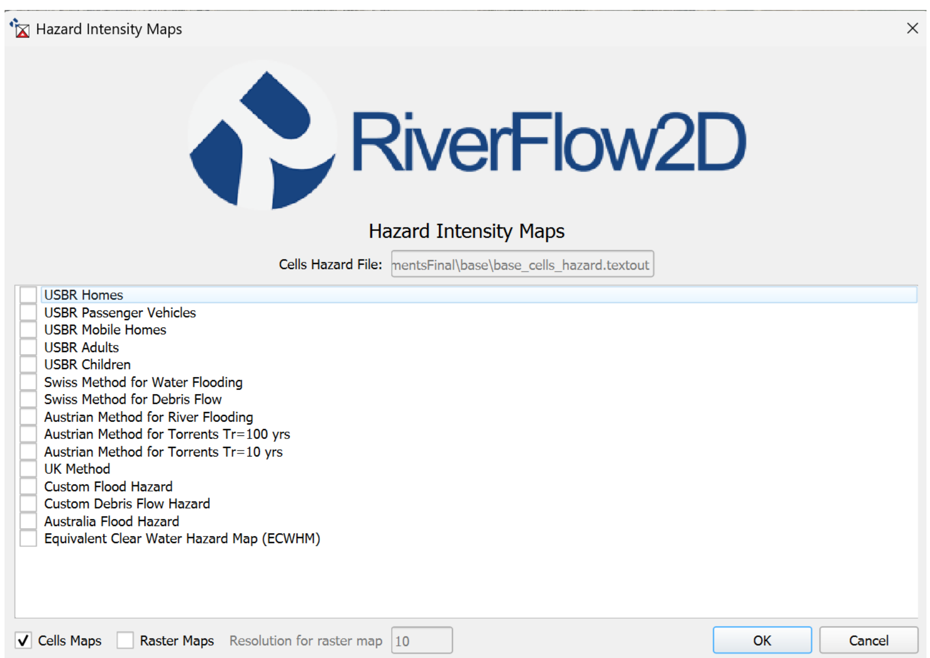


Figure 4.19 – Hazard Intensity Dialog

4.4.2 Dialog Controls

Cells Hazard File	*file browser*	Displays the directory containing simulation results.
Map Type List	*checkbox list*	Selects the hazard classification method to use (DEFRA, Australian, FEMA, Swiss, or Russo).
Cell Maps	*checkbox*	Enables vector (cell-based) map output.
Raster Maps	*checkbox*	Enables raster map output.
Resolution (m)	*textbox*	Sets the resolution for raster interpolation, in meters.
OK	*button*	Creates the hazard intensity map and adds it to the QGIS project.
Cancel	*button*	Closes the dialog without making changes.

4.4.3 Hazard Classification Methods

The Hazard Intensity Maps tool supports several standard hazard classification methods:

USBR Homes	US Bureau of Reclamation classification for residential homes, focused on structural damage risk.
USBR Passenger Vehicles	US Bureau of Reclamation classification for vehicle safety during flooding events.
USBR Mobile Homes	US Bureau of Reclamation classification specific to mobile/manufactured homes.
USBR Adults	US Bureau of Reclamation classification for adult human safety during flooding.
USBR Children	US Bureau of Reclamation classification for child safety during flooding, with more conservative thresholds.
Swiss Method for Water Flooding	Swiss federal classification system for clear water flood hazards, focusing on structural impacts.
Swiss Method for Debris Flow	Modified Swiss classification adjusted for the greater destructive potential of debris flows.
Austrian Method for River Flooding	Austrian classification method for riverine flooding scenarios.
Austrian Method for Torrents Tr=100 yrs	Austrian classification for torrent flooding with 100-year return period.
Austrian Method for Torrents Tr=10 yrs	Austrian classification for torrent flooding with 10-year return period.
UK Method	UK Department for Environment, Food and Rural Affairs (DEFRA) method based on depth and velocity combinations.
Custom Flood Hazard	User-defined classification method for specialized flood hazard assessment.
Custom Debris Flow Hazard	User-defined classification method for specialized debris flow hazard assessment.

USBR Homes	US Bureau of Reclamation classification for residential homes, focused on structural damage risk.
Australia Flood Hazard	Australian hazard classification method, emphasizing velocity as a significant factor.
Equivalent Clear Water Hazard Map (ECWHM)	Specialized map for mud/debris flows that converts the higher hazard of mud into an equivalent clear water hazard.

Each hazard category is visually represented with a distinct color coding scheme appropriate to the selected method, typically ranging from low hazard (green/blue) to extreme hazard (red).

4.4.4 Workflow

To generate hazard intensity maps, follow these steps:

1. Ensure you have completed a model simulation with the *Output hazard files* option enabled and have output files available, the *Output hazard files* option is available in the *Control Data* tab of the Hydronia Data Input Program.

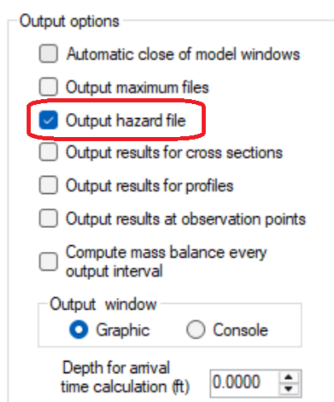


Figure 4.20 – Enabling Hazard file output in the Hydronia Data Input Program

2. From the *Results vs Time Maps* menu, click on *Hazard Intensity Maps* .

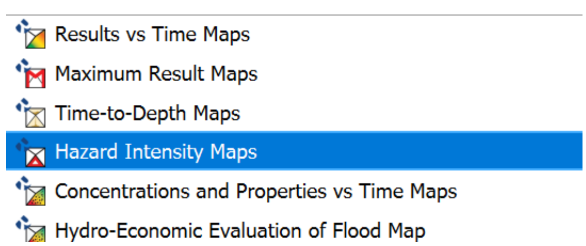


Figure 4.21 – Hazard Intensity Maps Menu for RiverFlow2D

3. In the dialog that appears, the current scenario directory should be automatically selected. If not, browse to the appropriate directory containing your simulation results.

4. Select one or more hazard classification methods from the list by checking the corresponding checkboxes:

    - USBR Homes - Classification for residential structural damage risk
    - USBR Passenger Vehicles - Classification for vehicle safety
    - USBR Mobile Homes - Classification for mobile/manufactured homes
    - USBR Adults - Classification for adult human safety
    - USBR Children - Classification for child safety with conservative thresholds
    - Swiss Method for Water Flooding - Swiss classification for clear water hazards
    - Swiss Method for Debris Flow - Swiss classification adapted for debris flows
    - Austrian Method for River Flooding - Austrian method for riverine flooding
    - Austrian Method for Torrents Tr=100 yrs - For torrent flooding with 100-year return period
    - Austrian Method for Torrents Tr=10 yrs - For torrent flooding with 10-year return period
    - UK Method - DEFRA method based on depth and velocity combinations
    - Custom Flood Hazard - User-defined method for specialized flood assessments
    - Custom Debris Flow Hazard - User-defined method for debris flow assessments
    - Australia Flood Hazard - Australian method emphasizing velocity factors
    - Equivalent Clear Water Hazard Map (ECWHM) - For mud/debris flow conversions

5. Choose the output format:

    - Check *Cell Maps* to create polygon-based maps (triangular mesh).
    - Check *Raster Maps* to create grid-based maps with interpolation.
    - If *Raster Maps* is selected, specify the raster resolution in meters (ft or m).

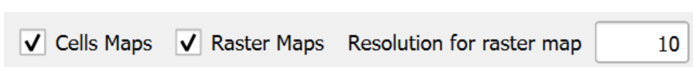


Figure 4.22 – Hazard Intensity Maps Output Options

6. Click “OK” to generate the hazard intensity map.

7. The generated map will be added to the QGIS layer panel under the “OUTPUT_RESULTS” group.

8. The map will display hazard categories based on the combination of maximum water depth and maximum velocity reached during the simulation period.

4.4.5 Requirements

Active Project	An active project must be loaded in QGIS.
Simulation Results	A completed model simulation with the *Output hazard files* option enabled and have output files available, the *Output hazard files* option is available in the *Control Data* tab of the Hydronia Data Input Program.
Required Data Files	The following files must be present in the scenario directory:

4.4.6 Technical Information

The tool processes data from *_cells_hazard.textout files, which contain precalculated hazard indicators for each computational cell in the model domain. The following table provides technical details about how each hazard classification method is calculated and visualized:

USBR Homes	US Bureau of Reclamation classification for residential homes. The algorithm applies the following criteria:
USBR Passenger Vehicles	US Bureau of Reclamation classification for vehicle safety during flooding:
USBR Mobile Homes	US Bureau of Reclamation classification for mobile/manufactured homes:
USBR Adults	US Bureau of Reclamation classification for adult human safety:
USBR Children	US Bureau of Reclamation classification for child safety with more conservative thresholds:
Swiss Method for Water Flooding	The Swiss method classifies hazards based on depth and velocity with a focus on potential damage to structures:
Swiss Method for Debris Flow	Similar to the water flooding method but adjusted for the greater destructive potential of debris flows:
Austrian Method for River Flooding	The Austrian method uses a simpler three-category classification focused on depth thresholds:
Austrian Method for Torrents Tr=100 yrs	Austrian classification for torrent flooding with 100-year return period:
Austrian Method for Torrents Tr=10 yrs	Austrian classification for torrent flooding with 10-year return period:
UK Method	UK Department for Environment, Food and Rural Affairs (DEFRA) method categorizes hazards based on depth, velocity, and their product:
Custom Flood Hazard	User-defined classification method for specialized flood hazard assessment. Parameters are specified by the user in a custom configuration file with format:
Custom Debris Flow Hazard	User-defined classification method for specialized debris flow hazard assessment, with similar format to Custom Flood Hazard but with additional parameters:
Australia Flood Hazard	The Australian method emphasizes velocity in hazard determination with a six-category system:

USBR Homes	US Bureau of Reclamation classification for residential homes. The algorithm applies the following criteria:
Equivalent Clear Water Hazard Map (ECWHM)	Specialized map for mud/debris flows that converts higher hazard of mud into equivalent clear water hazard:

4.5 Concentrations and Properties vs Time Maps

The Concentrations and Properties vs Time Maps tool visualizes the transport and dispersion of tracers or pollutants in the simulation.

4.5.1 Dialog Window

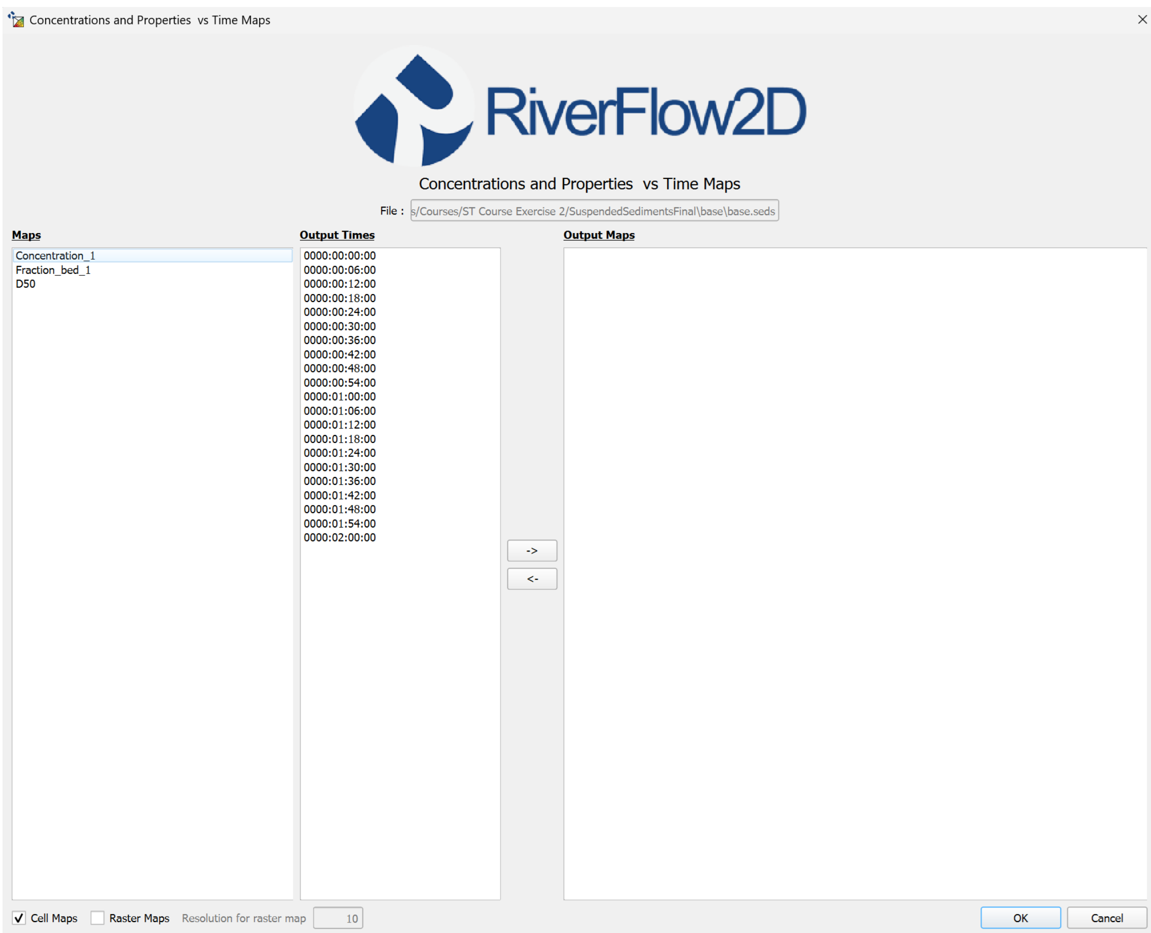


Figure 4.23 – Concentrations and Properties vs Time Maps Dialog for RiverFlow2D

You can reach this dialog by clicking on the *Concentrations and Properties vs Time Maps* button in the *Results vs Time Maps* tool:

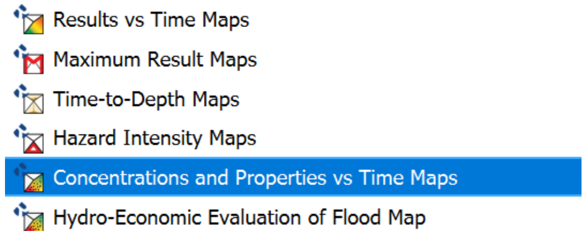


Figure 4.24 – Concentrations and Properties vs Time Maps Menu for RiverFlow2D

4.5.2 Dialog Controls

Scenario Directory	*file browser*	Selects the directory containing simulation results.
Map Type	*list view*	Selects the concentration maps to create, including: Concentration, Maximum Concentration, and Time of Maximum Concentration.
Output Times	*list view*	Displays available simulation output times that can be selected for visualization.
Arrow Buttons (→, ←)	*buttons*	Transfer selected maps and times to/from the Output Maps list.
Output Maps	*list view*	Shows the maps that will be created when OK is clicked.
Cell Maps	*checkbox*	Enables vector (cell-based) map output.
Raster Maps	*checkbox*	Enables raster map output.
Resolution for raster map	*textbox*	Sets the resolution for raster interpolation, in projected units.
OK	*button*	Creates the selected concentration maps and adds them to the QGIS project.
Cancel	*button*	Closes the dialog without making changes.

4.5.3 Concentrations Map Types

The Concentrations and Properties vs Time Maps tool can generate different map types depending on the transport module being used:

Pollutant Transport (.solutes)	dimensionless (0-1)
Sediment Transport (.seds)	
Mud/Debris Flow (.mud)	

For each map type, users can generate time-dependent maps showing values at specific time steps, or create maximum value maps showing the peak values reached during the simulation. Legend units are displayed in metric or English units based on the QGIS project settings.

4.5.4 Workflow

To generate maps using the Concentrations and Properties vs Time Maps tool, follow these steps:

1. Ensure you have completed a simulation with sediment/pollutant transport and have output files available.

2. From the Results vs Time Maps menu, click on "Concentrations and Properties vs Time Maps".

3. In the dialog that appears, the current scenario directory should be automatically selected. If not, browse to the appropriate directory containing your simulation results.

4. Select one or more map types to generate from the available options. The specific maps will depend on which transport module is active in your simulation:

   - For **Pollutant Transport** (.solutes):
     - Conc_1, Conc_2, etc. - Concentration values for each chemical species (mg/L or ppm)
   - For **Sediment Transport** (.seds):
     - Concentration_1, Concentration_2, etc. - Suspended sediment concentration (g/L or ppm)
     - Fraction_bed_1, Fraction_bed_2, etc. - Bed composition (dimensionless fraction, 0-1)
     - D50 - Median grain size distribution (mm or in)
   - For **Mud/Debris Flow** (.mud):
     - Conc_1, Conc_2, etc. - Individual fraction concentrations (g/L or ppm)
     - Conc_Total - Total concentration (g/L or ppm)
     - Density - Mixture density (kg/m^3 or lb/ft^3)
     - Viscosity - Mixture viscosity (Pa·s or $lb{\cdot}s/ft^2$)
     - YieldStress - Mixture yield stress (Pa or lb/ft^2)
     - Hdep - Deposition depth (m or ft)
     - Fraction_bed_1, Fraction_bed_2, etc. - Composition of deposited material (dimensionless fraction, 0-1)
     - D50 - Median grain size of deposited material (mm or in)

5. Select the output times you want to visualize. Hold down the Control key to select multiple times.

6. Transfer the selected maps and times to the *Output Maps* list by clicking the right arrow button [->].

7. If you wish to remove a map or time from the *Output Maps* list, select it and click the left arrow button .

8. Choose the output format:

   - Check *Cell Maps* to create polygon-based maps (triangular mesh).
   - Check *Raster Maps* to create grid-based maps with interpolation.
   - If *Raster Maps* is selected, specify the raster resolution in meters (ft or m).

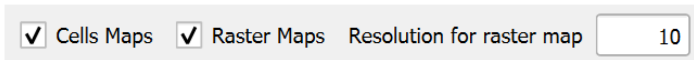


Figure 4.25 – Output Options for Concentrations and Properties vs Time Maps

9. Click “OK” to generate the concentration maps.

10. The generated maps will be added to the QGIS layer panel under the “OUTPUT_RESULTS” group.

4.5.5 Requirements

Active Project	An active project must be loaded in QGIS.
Simulation Type	A completed simulation with one of the following modules activated:
Required Data Files	The following files must be present in the scenario directory:
Concentration Data	The simulation must have output data for at least one pollutant/sediment/mud concentrations or properties.

4.5.6 Technical Information

The following table provides technical details on how different file types are processed and how maps are generated:

File Type Detection	The tool examines the project’s .dat file to determine which transport modules are active:
Pollutant Transport Maps	For .solutes files, the tool processes:
Sediment Transport Maps	For .seds files, the tool processes:
Mud/Debris Flow Maps	For .mud files, the tool processes:
Concentration Maps	For time-dependent concentration maps:
Maximum Concentration Maps	For maximum concentration maps:
Time of Maximum Concentration Maps	For time of maximum concentration maps:
Vector Cell Map Creation	For cell-based (vector) maps:

File Type Detection	The tool examines the project's .dat file to determine which transport modules are active:
Raster Map Creation	For raster maps, the tool uses TIN (Triangulated Irregular Network) interpolation:
Time Series Filtering	The . outfiles file is used to:
Multi-Element Selection	For models with multiple transported elements (pollutants or sediment fractions):

The technical implementation ensures that concentration and property maps accurately represent the numerical results from the transport simulation while providing visually informative representations. For simulations with multiple elements or fractions, the tool maintains the distinction between different constituents, enabling detailed analysis of transport and mixing processes. The option to generate both vector-based and raster-based outputs provides flexibility for different visualization and analysis needs.

4.6 Hydro-Economic Evaluation of Flood Map

The Hydro-Economic Evaluation of Flood Map tool integrates flood hazard information with exposure data to assess risk.

4.6.1 Dialog Window

HEEF Maps Dialog

4.6.2 Dialog Controls

Scenario Directory	*file browser*	Selects the directory containing simulation results.
Exposure Layer	*layer selector*	Selects the vector layer containing exposure information (buildings, infrastructure, etc.).
Hazard Method	*dropdown*	Selects the hazard classification method to use.
HEEF Method	*dropdown*	Selects the HEEF risk assessment method to use.
OK	*button*	Creates the HEEF risk map and adds it to the QGIS project.
Cancel	*button*	Closes the dialog without making changes.

4.6.3 HEEF Map Types

The Hydro-Economic Evaluation of Flood Map tool can generate the following map types:

Damage Map	Shows estimated damage in monetary units (e.g., dollars) for each cell in the domain based on the selected damage assessment method.
Normalized Damage Map	Shows damage as a percentage of the total value of assets in each cell.
Maximum Depth Map	Displays the maximum water depth reached during the simulation. This is used in the damage calculations.
Land Use Map	Shows the land use categories assigned to each cell. Different damage functions are applied based on land use.

4.6.4 Workflow

To generate maps using the Hydro-Economic Evaluation of Flood Map tool, follow these steps:

1. Ensure you have completed a RiverFlow2D simulation and have output files available.
2. From the Results vs Time Maps menu, click on “Hydro-Economic Evaluation of Flood Map”.
3. In the dialog that appears:
    - Select the directory containing the simulation results.
    - Select a land use raster file from the dropdown menu (this should be a GeoTIFF file with land use categories).
    - Choose a damage assessment method from the dropdown menu (e.g., USACE, FEMA, HAZUS, custom).
    - Optionally, select a depth-damage function file if using a custom damage assessment method.
    - Select a currency for damage calculations (e.g., USD, EUR, GBP).
    - Enter property values for each land use category, or use the default values provided.
4. Choose the output format:
    - Check *Cell Maps* to create polygon-based maps (triangular mesh).
    - Check *Raster Maps* to create grid-based maps with interpolation.
    - If *Raster Maps* is selected, specify the resolution in meters (ft or m).
5. Click “Calculate” to perform the damage assessment calculation.
6. A progress bar will display the calculation status.
7. Once complete, select which map types to generate from the “Select Maps” list.
8. Click “OK” to generate the selected maps.
9. The generated maps will be added to the QGIS layer panel under the “OUTPUT_RESULTS” group.
10. A summary report with damage statistics will also be generated and can be accessed through the “View Report” button.

4.6.5 Requirements

Active Project	An active project must be loaded in QGIS.
Simulation Results	A completed simulation with output files in the project's scenario directory.
Required Data Files	The following files must be present in the scenario directory:
Land Use Data	A land use raster file (GeoTIFF format) must be available and properly aligned with the model domain. Each pixel value should correspond to a land use category code.
Depth-Damage Functions	Built-in depth-damage functions are available for standard assessment methods (USACE, FEMA, HAZUS). For custom methods, a CSV file with depth-damage relationships must be provided.
Property Values	Monetary values for each land use category must be specified. Default values are provided but may need adjustment based on local economic conditions.

4.6.6 Technical Information

The following table provides technical details on how damage assessment is performed and how maps are generated:

Data Integration Process	The tool integrates three key data types:
Building Exposure Layer	The tool requires a vector layer (typically called "HEEF_Builds") containing:
Vulnerability Functions	Vulnerability functions are stored in separate text files (with .txt extension) named after their function ID. Two types are supported:
Sampling Flood Parameters	The tool samples flood parameters (depth and velocity) for each building using:
1D Vulnerability Function Interpolation	For depth-only vulnerability functions (Type 1):
2D Vulnerability Function Interpolation	For depth-velocity vulnerability functions (Type 2):
Damage Calculation	Economic damage is calculated for each building using:
Output Map Generation	The tool creates a polygon shapefile ("HEEF_Damage.shp") with the following attributes:
Visualization	The damage map is styled using:
Data Management	The tool automatically:

The Hydro-Economic Evaluation of Flood Map tool provides a quantitative approach to flood risk assessment by translating hydraulic model results into economic impacts. The implementation uses specialized interpolation techniques to accurately represent the relationship between flood parameters and damage, accounting for the uncertainty inherent in vulnerability assessments. The combination of accurate hydraulic modeling with economic damage functions creates a powerful decision support tool for flood risk management and planning.

4.7 Oil on Land Maps

The Oil on Land Maps tool allows users to visualize time-dependent maps showing the physical properties of oil that has been spilled on land.

4.7.1 Dialog Window

Oil on Land Properties vs Time Maps Dialog
This tool provides visualization of how oil properties change over time as they interact with the terrain and environment.

4.7.2 Dialog Controls

.Dat File	*text field*	Displays the path to the simulation results file (read-only).
Maps	*list view*	Displays available oil property map types that can be selected for visualization.
Output Times	*list view*	Displays available simulation output times that can be selected for visualization.
Arrow Buttons (→, ←)	*buttons*	Transfer selected maps and times to/from the Output Maps list.
Output Maps	*list view*	Shows the maps that will be created when OK is clicked.
Map for multi spill(s)	*dropdown*	For simulations with multiple spills, allows selection of which spill(s) to map.
Cell Maps	*checkbox*	Enables cell-based (vector polygon) map output.
Raster Maps	*checkbox*	Enables raster map output.
Resolution for raster map	*text field*	Sets the resolution for raster interpolation, in projected units. Only enabled when “Raster Maps” is checked.
Vector Skip Frequency	*spinner*	Sets the number of vectors to skip between each vector in vector-type maps, controlling display density.
OK	*button*	Creates the selected maps and adds them to the QGIS project.
Cancel	*button*	Closes the dialog without making changes.

4.7.3 Oil on Land Map Types

The Oil on Land Maps tool can generate the following maps showing oil properties:

Oil Temperature	Temperature of the oil at each computational cell, which affects physical properties of the oil.
Oil Density	Density of the oil at each computational cell, which changes as the oil weathers.
Oil Viscosity	Viscosity of the oil at each computational cell, which affects flow behavior and penetration into soil.
Oil Yield Stress	Yield stress of the oil at each computational cell, indicative of its resistance to flow.

4.7.4 Workflow

To generate oil on land property maps, follow these steps:

1. Ensure you have completed an OilFlow2D simulation with the Heat Transfer option enabled.
2. From the Results vs Time Maps menu, click on "*Oil on Land Properties vs Time Maps*" to open the dialog.
3. Select one or more map types from the Maps list (Oil Temperature, Oil Density, Oil Viscosity, Oil Yield Stress).
4. Select one or more output times from the Output Times list.
5. Click the → button to add the selected map types and times to the Output Maps list.
6. If your simulation includes multiple oil spills, select the appropriate spill from the "Map for multi spill(s)" dropdown.
7. Select the desired output format:
   - Check "Cell Maps" to create polygon shapefiles.
   - Check "Raster Maps" to create interpolated raster grids (specify the resolution).
8. Click OK to generate the selected maps.

4.7.5 Requirements

To use the Oil on Land Properties vs Time Maps tool, you need:

- A completed OilFlow2D simulation with Heat Transfer option enabled
- The following files in your project directory:
  - *.dat - Simulation control file
  - *.oilp - Oil properties file with Heat Transfer option enabled (line 12 = 1)

  - *.OUTFILES - List of output times
  - cell_ol_*.textout - Cell oil land data output files for each time step

- Domain Outline shapefile for raster map creation

4.8 Oil on Water Maps

The Oil on Water Maps tool allows users to create spatial maps showing the distribution and properties of oil spilled on water surfaces.

4.8.1 Dialog Window

Oil on Water Spatial Maps Dialog
This tool provides a comprehensive view of oil spill characteristics on water bodies, allowing users to assess the extent and potential impacts of the spill.

4.8.2 Dialog Controls

.DAT File	*text field*	Displays the path to the simulation results file (read-only).
Map Type List	*list widget*	Displays available oil property map types that can be selected for visualization.
Cell Maps	*checkbox*	Enables cell-based (vector polygon) map output.
Raster Maps	*checkbox*	Enables raster map output.
Resolution for raster map	*text field*	Sets the resolution for raster interpolation, in projected units. Only enabled when “Raster Maps” is checked.
OK	*button*	Creates the selected maps and adds them to the QGIS project.
Cancel	*button*	Closes the dialog without making changes.

4.8.3 Oil on Water Map Types

The Oil on Water Maps tool can generate the following maps:

Oil State	Classification of oil location (oil in water, on shore, or on bed) represented with different colors.
Oil Concentration	Concentration of oil at each computational cell.
Oil Mass	Mass of oil at each computational cell.
Oil Thickness	Thickness of the oil slick at each computational cell.

Oil State	Classification of oil location (oil in water, on shore, or on bed) represented with different colors.
Oil Temperature	Temperature of the oil at each computational cell, which affects physical properties.
Oil Density	Density of the oil at each computational cell, which changes as the oil weathers.
Oil Viscosity	Viscosity of the oil at each computational cell, which affects spread rate and interaction with water.
Total Evaporated Mass	Amount of oil that has evaporated from each computational cell.
Maximum Area	The maximum extent of oil coverage over the entire simulation.

4.8.4 Workflow

To generate oil on water maps, follow these steps:

1. Ensure you have completed an OilFlow2D simulation.
2. From the Results vs Time Maps menu, click on "*Oil on Water Spatial Maps*" to open the dialog.
3. Select one or more map types from the Map Type List.
4. Select the desired output format:
    - Check "Cell Maps" to create polygon shapefiles.
    - Check "Raster Maps" to create interpolated raster grids (specify the resolution).
5. Click OK to generate the selected maps.

4.8.5 Requirements

To use the Oil on Water Spatial Maps tool, you need:

- A completed OilFlow2D simulation
- The following files in your project directory:
    - *.dat - Simulation control file
    - *.oilw - Oil on water properties file
    - oil_concentration.txt - Oil concentration data
    - oil_mass.txt - Oil mass data
    - oil_state.txt - Oil state classification data
    - oil_thickness.txt - Oil thickness data
    - oil_temperature.txt - Oil temperature data (if heat transfer is enabled)
- For simulations with multiple spills, corresponding numbered output files
- Domain Outline shapefile for raster map creation

4.9 Common Features and Functionality

All the mapping tools in the plugin share common features and functionality:

4.9.1 Output Options

- **Cell Maps:** Creates polygon shapefiles representing simulation mesh cells, with attributes for the selected parameters.
- **Raster Maps:** Creates interpolated raster grids for smoother visualization, with user-defined resolution.

4.9.2 Raster Map Creation

For raster map creation, the Results vs Time Maps tool uses TIN (Triangulated Irregular Network) interpolation to convert cell-centered values to a continuous surface at user-specified resolution. The resulting raster is clipped to the domain outline and no-data values (-9999.0) are properly handled and excluded. The tool ensures efficient processing of large datasets while maintaining numerical accuracy. The implementation supports both cell-based (vector) and interpolated (raster) visualizations, providing flexibility for different analysis needs.

4.9.3 Layer Management

- All generated maps are added to an "OUTPUT_RESULTS" group in the QGIS layer panel.
- Appropriate styling is automatically applied to each map type, including color ramps and classification methods.
- Vector layers with directional data (velocity, sediment transport) are displayed with arrow symbols showing direction and magnitude.

4.9.4 Export Capabilities

- Maps can be exported as shapefiles or raster files for further analysis or sharing.
- The Animation Tool (described in Chapter 4) can be used to create time-series animations of most results vs time based map types.
- Map outputs can be incorporated into QGIS print layouts for highly customizable report generation.

5 Animation Tool

The Animation Tool is a specialized QGIS plugin component designed to create dynamic visualizations of RiverFlow2D model results over time. This tool allows users to generate animations showing how hydraulic parameters (such as water depth, velocity, and water surface elevation) change throughout a simulation, and export these animations as video files for presentations and analysis.

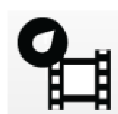


Figure 5.1 – Animation Tool Icon for OilFlow2D

5.1 Main Interface

The Animation Tool provides a dockable widget interface that allows users to control the animation of model results and access various export options.

5.1.1 Dialog Window

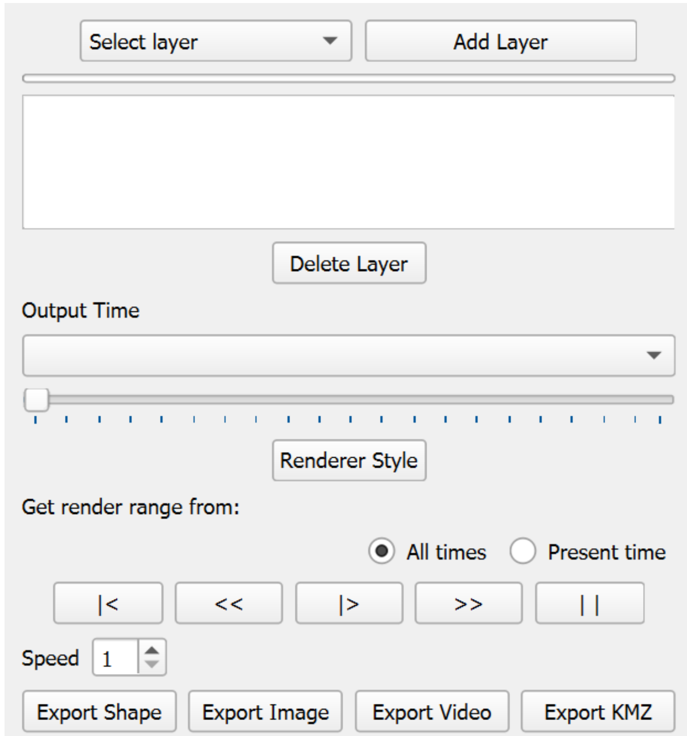


Figure 5.2 – Animation Tool Interface

5.1.2 Dialog Controls

Control	**Type**	**Description**
Select Layer	*Dropdown*	Selects the layer to be animated.
Add Layer	*Button*	Constructs and adds the selected layer to the layer list.
Progress Bar	*Progress Bar*	Shows the progress of the layer construction process.
Layer List	*List Box*	Displays all loaded layers that can be animated. Each layer in the list can be selected to control which one is currently being animated. Appears after clicking the Add Layer button and successfully loading a layer.
Delete Layer	*Button*	Removes the selected layer from the Layer List. Only active when a layer is selected in the Layer List.
Output Time	*Dropdown*	Selects which time step to display from the simulation results.
Time Slider	*Horizontal Slider*	Allows scrolling through available time steps.

Control	Type	Description
Renderer Style	*Button*	Opens the Renderer Style dialog to customize the visualization settings for the current layer, including color ramp, classification method, and value ranges.
Render Range	*Radio Buttons*	Options to set the render range from “All times” (uses min/max values across all time steps) or “Present time” (uses only the current time step’s min/max values).
Animation Controls	*Button Group*	Includes Rewind ($\| <$), Back ($<<$), Play ($\| >$), Step ($>>$), and Pause ($\|\|$) buttons for controlling animation playback.
Animation Speed	*Spin Box*	Controls the speed of the animation playback.
Export Shape	*Button*	Opens a dialog to export the current animation frame as a shapefile.
Export Image	*Button*	Opens the Export Map dialog to create map compositions for each time step.
Export Video	*Button*	Opens the Export Video dialog to create video files from the animation.
Export KMZ	*Button*	Opens the Export KMZ dialog to create Google Earth compatible animations.

5.1.3 Workflow

The main workflow to create an animation is as follows:

1. Select the desired layer in the *Select Layer* dropdown.
2. Click *Add Layer* to load it into the list.
3. Select the layer in the layer list.
4. Click *Export Video* to open the export dialog.
5. Configure the export parameters and generate the video.

5.1.4 Requirements

To use the Animation Tool effectively:

- Model output files with time series data
- Sufficient system memory to handle large datasets
- FFmpeg (bundled with the plugin) for video export functionality
- Google Earth (optional, for viewing KMZ animations)

5.1.5 Technical Details

5.1.5.1 Animation Engine

The tool generates frames by updating layer symbology for each time step and rendering a QGIS layout composition.

- Time steps are computed from *simTime* and *interTime* (optionally *tsta*/*tend*) stored on the layer.
- Each frame is exported from a *QgsPrintLayout* with the map and optional elements (title, time, legend).

5.1.5.2 Video Export

- Temporary images are converted to an AVI video using FFmpeg.
- Quality controls codec and bitrate (best/high/low).

5.1.5.3 KMZ Export

- Each frame is georeferenced and saved as a PNG with its associated KML.
- Files are compressed into a KMZ for Google Earth viewing.

5.1.6 Tips and Best Practices

- **Performance optimization**: For large datasets, consider reducing the resolution or extent of the animation to improve performance.
- **Speed selection**: For simulations with many time steps, a higher frame rate may be appropriate. For simulations with fewer time steps, a lower frame rate may provide better results.
- **Template design**: When creating print layout templates for animations, ensure that all map elements are properly positioned and sized for the intended output resolution.
- **Color ramps**: Choose appropriate color ramps for your data to effectively communicate the changes over time. Consider using diverging color ramps for parameters that have meaningful positive and negative values.
- **File size management**: High-quality videos can become very large. Consider using lower quality settings for draft versions and higher quality only for final outputs.
- **Time step selection**: For long simulations, you may not need to include every time step in the animation. Select a representative subset of time steps to reduce processing time and file size.

5.2 Export Image Dialog - Image Tab

The Export Image Dialog allows users to create a static image of a specific time step from the animation. The Image tab allows configuration of image dimensions, the time step to export, and the output file location.

5.2.1 Dialog Window

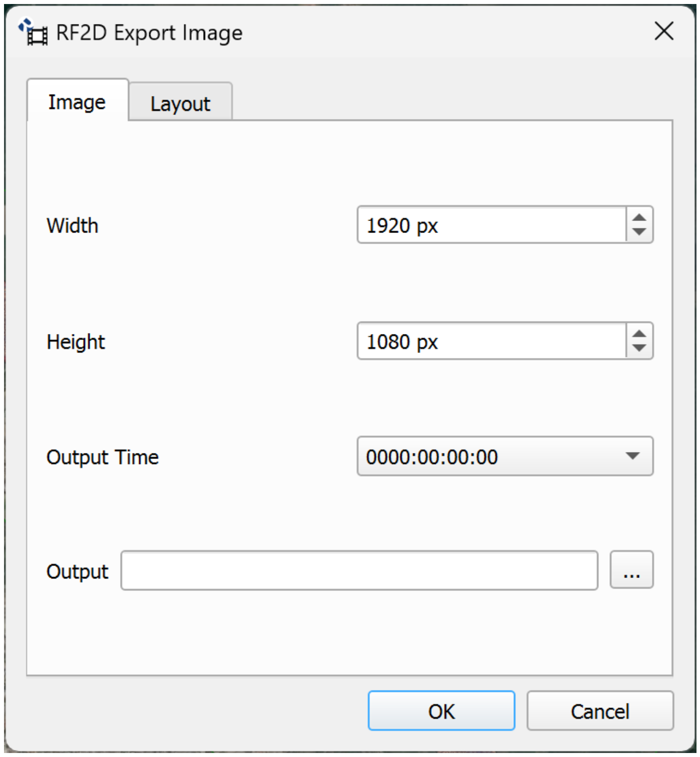


Figure 5.3 – Export Image Dialog - Image Tab

5.2.2 Dialog Controls

Control	Type	Description
Width	*Spin Box*	Sets the width of the image in pixels (default: 1920 px).
Height	*Spin Box*	Sets the height of the image in pixels (default: 1080 px).
Output Time	*Dropdown*	Selects the specific time step to export as an image.
Output	*Text Field*	Specifies the path and filename for the output image file.
Browse	*Button*	Opens a file dialog to select the output file location.
OK	*Button*	Starts the image export process.
Cancel	*Button*	Closes the dialog without exporting.

5.2.3 Workflow

To export an image of a specific time step:

1. Select the layer you want to export in the main Animation Tool interface and click *Add Layer*.
2. With the layer loaded in the list, click the *Export Image* button.
3. In the *Image* tab, configure the desired width and height for the image.
4. Select the time step you want to export from the *Output Time* dropdown.
5. Click the *Browse* button (...) next to *Output* to select the location and filename for the output file (JPG format).
6. Optionally, switch to the *Layout* tab to configure additional elements such as title, time, and legend.
7. Click *OK* to generate the image.

5.2.4 Requirements

- An animatable layer loaded with time series data (fields F1, F2, F3, etc.).
- Output path set for the .jpg file.
- Valid image dimensions (width and height in pixels).

5.2.5 Technical Details

- Image export uses the picture () function which renders a single frame based on the selected time step.
- The attribute field corresponding to the time step is calculated as “F” + (time + 1), where time is the index selected in the *Output Time* dropdown.
- The layer renderer is cloned and updated to display the field corresponding to the selected time step.
- The image is exported using QgsLayoutExporter.exportToImage() with DPI setting of 96 and the specified image size.
- The output format is JPG, determined by the file browser dialog.

5.3 Export Image Dialog - Layout Tab

The Export Image Dialog - Layout Tab allows users to configure the layout of the exported image. You can use a default layout with configurable elements (title, time, legend) or a custom layout created in QGIS Print Layout.

5.3.1 Dialog Window

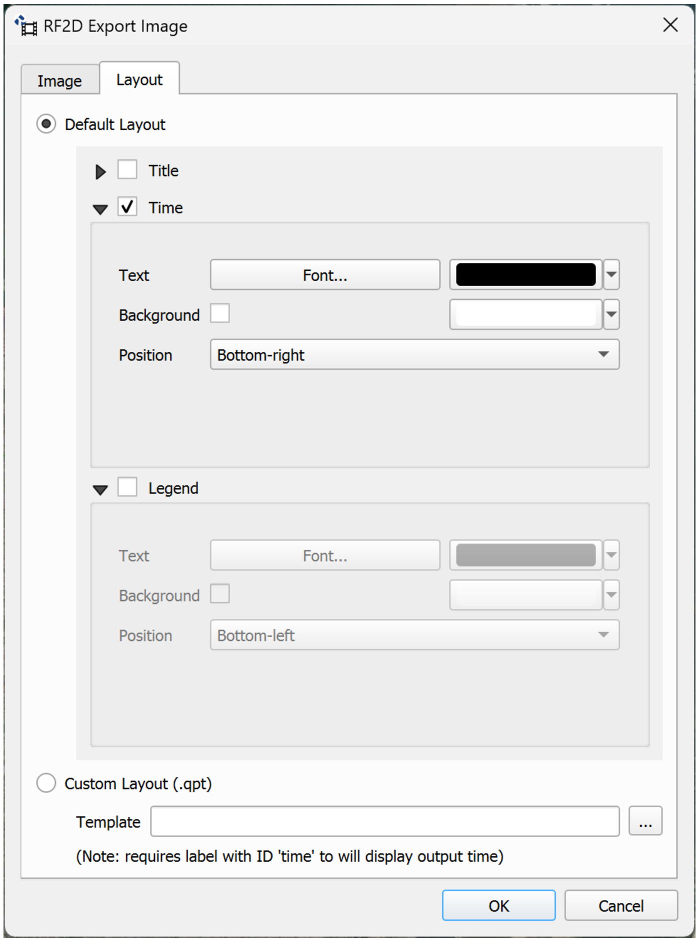


Figure 5.4 – Export Image Dialog - Layout Tab

5.3.2 Dialog Controls

Control	Type	Description
Default Layout	*Radio Button*	Selects the default layout option. When selected, allows configuration of Title, Time, and Legend elements.
Title	*Checkbox Group*	Enables and configures the display of a title on the image. Includes options for label, text, font, position and background.
Time	*Checkbox Group*	Enables and configures the display of the time on the image. Includes options for label text, font, position, and background.

Control	Type	Description
Legend	*Checkbox Group*	Enables and configures the display of the legend on the image. Includes options for label text, font, position and background.
Custom Layout (.qpt)	*Radio Button*	Selects the custom layout option using a QGIS Print Layout template file (.qpt).
Template	*Text Field with Button*	Specifies the path to the .qpt template file. A browse button allows navigating to the file. Includes a note indicating that a label with ID 'time' is required to display the output time.
OK	*Button*	Starts the image export process with the specified layout.
Cancel	*Button*	Closes the Export Image dialog.

5.3.3 Workflow

To configure the image layout:

1. Open the Export Image dialog by clicking the *Export Image* button on the main Animation Tool interface.
2. Select the *Layout* tab.
3. **To use the default layout:**
   - Select the *Default Layout* radio button.
   - Check the *Title* checkbox if you want to display a title. Configure the text, font, color, and position.
   - Check the *Time* checkbox to display the time of the selected step. Configure the font, color, and position.
   - Check the *Legend* checkbox to display the layer legend. Configure the font, color, and position.
4. **To use a custom layout:**
   - Select the *Custom Layout (.qpt)* radio button.
   - Click the *Browse* button (…) next to the *Template* field.
   - Navigate to the .qpt template file previously created in QGIS Print Layout.
   - Select the file and click *Open*.
   - Ensure the template includes a label element with id="time" if you want to display the time.
5. Switch to the *Image* tab to configure dimensions and time step.
6. Click *OK* to generate the image with the configured layout.

5.3.4 Requirements

- For default layout, no additional files are required.
- For custom layout, a valid .qpt template file created in QGIS Print Layout is required.
- Custom templates must include a map item that will be used to render the image.
- If the template includes a label item with id="time", it will be automatically updated with the time of the selected step.
- The image aspect ratio will automatically adjust to match the template's page size when using a custom layout.

5.3.5 Technical Details

- Default layout elements are created using the same classes as video export: QgsLayoutItemLabel (for title and time) and QgsLayoutItemLegend (for legend).
- Element positions use the same constants: TOP_CENTER, TOP_LEFT, TOP_RIGHT, BOTTOM_LEFT, BOTTOM_RIGHT.
- Custom templates are loaded using QDomDocument and c.readLayoutXml() to interpret the .qpt file.
- When using a custom template, the image width is automatically adjusted to maintain the template's aspect ratio, similar to video export.
- Time elements are updated via the composition_set_time() function, which searches for items with id="time" and sets their text to DDDD:HH:MM:SS format.
- The picture () function in composer.py handles composition creation and image export, reusing much of the video export code.

5.4 Export Video Dialog - Layout Tab

The Export Video Dialog - Layout Tab allows users to configure the layout of the animation. You can use a default layout or a custom layout created in the *QGIS Print Layout* program.

5.4.1 Dialog Window

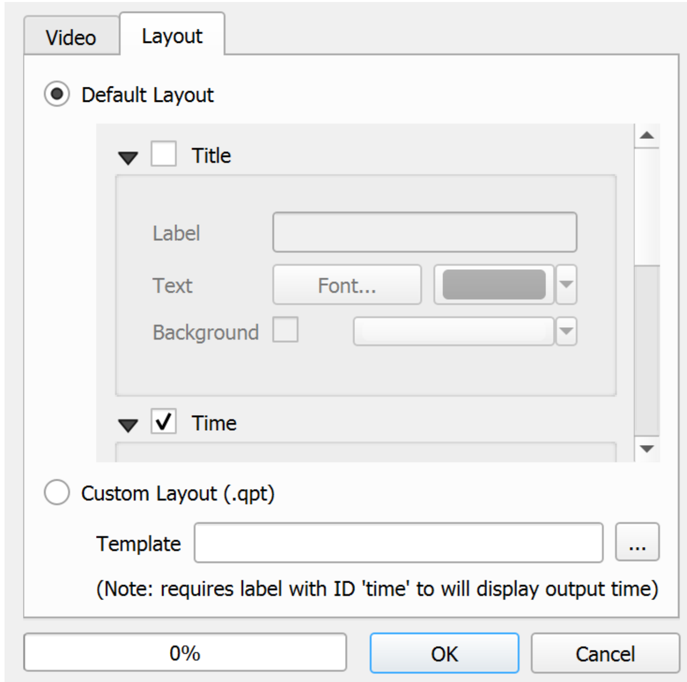


Figure 5.5 – Export Video Dialog - Layout Tab

5.4.2 Dialog Controls

Control	Type	Description
Default Layout	*Radio Button*	Selects the default layout option. When selected, allows configuration of Title and Time elements.
Title	*Checkbox Group*	Enables and configures the display of a title on the video frames. Includes options for label, text, font, position and background.
Time	*Checkbox Group*	Enables and configures the display of the current animation time on the video frames. Includes options for label text, font, position, and background.
Legend	*Checkbox Group*	Enables and configures the display of the animation legend on the video frames. Includes options for label text, font, position and background.
Custom Layout (.qpt)	*Radio Button*	Selects the custom layout option using a QGIS Print Layout template file (.qpt).

Control	Type	Description
Template	*Text Field with Button*	Specifies the path to the .qpt template file. A browse button allows navigating to the file.
Progress Bar	*Progress Bar*	Shows the progress of the video export process.
Export	*Button*	Starts the video export process with the specified layout and video settings.
Close	*Button*	Closes the Export Video dialog.

5.4.3 Workflow

To configure the animation video layout:

1. Open the Export Video dialog by clicking the *Export Video* button on the main Animation Tool interface.
2. Select the *Layout* tab.
3. **To use the default layout:**
   - Select the *Default Layout* radio button.
   - Check the *Title* checkbox if you want to display a title. Configure the text, font, color, and position.
   - Check the *Time* checkbox to display the current simulation time. Configure the font, color, and position.
   - Check the *Legend* checkbox to display the layer legend. Configure the font, color, and position.
4. **To use a custom layout:**
   - Select the *Custom Layout (.qpt)* radio button.
   - Click the *Browse* button next to the *Template* field.
   - Navigate to the .qpt template file previously created in QGIS Print Layout.
   - Select the file and click *Open*.
5. Switch to the *Video* tab to configure video parameters.
6. Click *Export* to generate the video with the configured layout.

5.4.4 Requirements

- For default layout, no additional files are required.
- For custom layout, a valid .qpt template file created in QGIS Print Layout is required.
- Custom templates must include a map item that will be used to render the animation frames.

- If the template includes a label item with id="time", it will be automatically updated with the time for each frame.
- The video aspect ratio will automatically adjust to match the template's page size when using a custom layout.

5.4.5 Technical Details

- Default layout elements are created dynamically using QgsLayoutItemLabel classes (for title and time) and QgsLayoutItemLegend (for legend).
- Element positions are defined by constants: TOP_CENTER, TOP_LEFT, TOP_RIGHT, BOTTOM_LEFT, BOTTOM_RIGHT.
- Custom templates are loaded using QDomDocument and c.readLayoutXml() to interpret the .qpt file.
- When using a custom template, the video width is automatically adjusted to maintain the template's aspect ratio: aspect = page_width / page_height; w = aspect * h.
- Time elements in templates are updated via the composition_set_time() function, which searches for items with id="time" and sets their text to DDDD:HH:MM:SS format.
- Each frame is exported using QgsLayoutExporter.exportToImage() with DPI setting of 96 and the specified image size.

5.5 Export Video Dialog - Video Tab

The Export Video Dialog provides advanced options for creating animations with custom layouts and templates.

5.5.1 Dialog Window

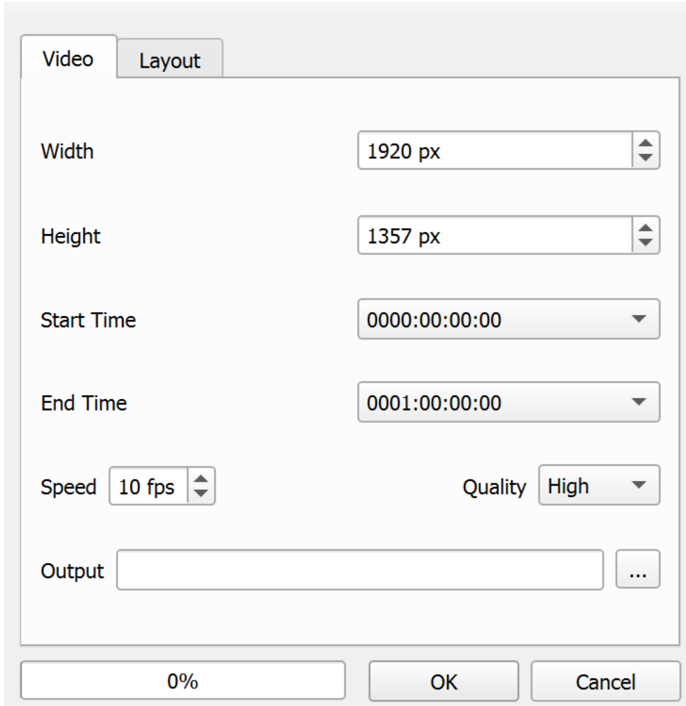


Figure 5.6 – Export Video Dialog - Video Tab

5.5.2 Dialog Controls

Control	Type	Description
Width	*Spin Box*	Sets the width of the animation frames in pixels (default: 1920 px).
Height	*Spin Box*	Sets the height of the animation frames in pixels (default: 1357 px).
Start Time	*Dropdown*	Selects the starting time step for the animation.
End Time	*Dropdown*	Selects the ending time step for the animation.
Speed	*Spin Box*	Sets the frames per second (FPS) for the animation (default: 10 FPS).
Quality	*Dropdown*	Sets the video quality (Low, Medium, High).
Template	*Text Field*	Specifies a QGIS print layout template to use for the animation frames.
Browse Template	*Button*	Opens a file dialog to select a template file.
Output File	*Text Field*	Specifies the path and filename for the output video file.
Browse Output	*Button*	Opens a file dialog to select the output file location.

Control	Type	Description
Export	*Button*	Starts the animation export process.
Close	*Button*	Closes the dialog without exporting.

5.5.3 Workflow

To export a video from the animation:

1. Select the layer you want to animate from the *Select Layer* dropdown and click *Add Layer* to load it into the list.
2. Select the layer in the list and click *Export Video*.
3. Set the time range, speed, quality, and output file.
4. If using a custom layout, select the .qpt file.
5. Click *Export* to start the video creation.

5.5.4 Requirements

- At least one animatable layer loaded via *Add Layer*.
- Valid time range (Start Time $\leq$ End Time).
- Output path set for the .avi file.
- If using a custom layout, the .qpt file must exist.
- Bundled FFmpeg available.

5.5.5 Technical Details

- Export generates temporary images for each time step and then compiles them into an AVI using FFmpeg.
- Time range is derived from layer custom properties (*simTime*, *interTime*, *tsta*, *tend*).
- Layout can be default (title/time/legend) or .qpt-based, adjusting aspect ratio accordingly.

5.6 Export KMZ Dialog

The Export KMZ dialog allows users to create Google Earth compatible animations from the model results.

5.6.1 Dialog Window

Export KMZ Dialog

5.6.2 Dialog Controls

Control	Type	Description
Start Time	*Dropdown*	Selects the starting time step for the KMZ animation.
End Time	*Dropdown*	Selects the ending time step for the KMZ animation.
Output File	*Text Field*	Specifies the path and filename for the output KMZ file.
Browse	*Button*	Opens a file dialog to select the output file location.
Export	*Button*	Starts the KMZ export process.
Close	*Button*	Closes the dialog without exporting.

5.6.3 Workflow

To export a KMZ animation for Google Earth:

1. Select the layer you want to animate in the main Animation Tool interface and click *Add Layer*.
2. With the layer loaded in the list, click the *Export KMZ* button.
3. In the Export KMZ dialog, configure the time range using the *Start Time* and *End Time* drop-downs.
4. Adjust the raster image resolution in the *Resolution for raster image* field. Higher values produce more detailed images but larger files.
5. Optionally, edit the place name in the *Place* field (default is “ProjectName-LayerName”).
6. Click the *Browse* button next to *Output File* to select the location and name for the output KMZ file.
7. Click *Export* to start the export process. The process has two phases:
   - **Rasterizing**: Converts each time step of the vector layer into a raster.
   - **Rendering**: Applies symbology and renders each raster with appropriate colors.
8. Once the export is complete, a confirmation message is displayed. The KMZ file can be opened in Google Earth.

5.6.4 Requirements

- An animatable layer loaded with time series data (fields F1, F2, F3, etc.).
- The layer must be scalar type (depth, elevation, etc.). Vector field layers are not supported for KMZ export.

- Valid time range (Start Time $\leq$ End Time).
- Output path set for the .kmz file.
- GDAL installed (included with QGIS) for rasterization.
- Sufficient disk space for temporary files (automatically deleted upon completion).
- Google Earth installed to view the exported KMZ file (optional).

5.6.5 Technical Details

- The KMZ export process consists of three main stages:
  1. **Rasterization**: Each time field (F1, F2, …, Fn) is rasterized using gdal:rasterize with the specified resolution. NoData values are set to -9999.
  2. **Rendering**: The original layer's color symbology is applied to each raster. The color ramp configured in the animated layer is used with equal interval classification and 8 classes.
  3. **KMZ Generation**: A KML file is created with <GroundOverlay> elements for each frame, specifying geographic coordinates and time interval <TimeSpan>.
- Coordinates are automatically transformed from the layer's coordinate system to WGS84 (EPSG:4326) required by Google Earth using QgsCoordinateTransform.
- The time format in the KML follows the ISO 8601 standard: 1900-01-DDTHH:MM:SSZ, where DD represents the day calculated from the start of the simulation.
- Temporary raster files are stored in two temporary directories: one for raw rasterized rasters and another for rendered rasters. Both are automatically deleted after creating the KMZ file.
- The final KMZ file is a compressed ZIP file containing the doc.kml file and all rendered .tif image files.
- The <LookAt> element in the KML is automatically set to the center of the layer extent with a range of 20,000 meters to provide an appropriate initial view in Google Earth.

6

Cross Sections Tool

The Cross Sections Tool is a specialized QGIS plugin designed to generate, visualize, and analyze cross-sectional profiles of terrain and water surfaces from RiverFlow2D model outputs. This tool allows users to create profiles along user-defined lines, visualize how cross-sections change over time, and export the data for further analysis.

Figure 6.1 – Cross Sections Tool Icon for OilFlow2D

6.1 CrossSection Tab

The CrossSection tab is the primary interface of the Cross Sections Tool for creating and visualizing cross-sectional profiles. It displays a graphical representation of the terrain and water surface profile along a user-defined line, with controls for time navigation, animation, and export.

6.1.1 Tab Window

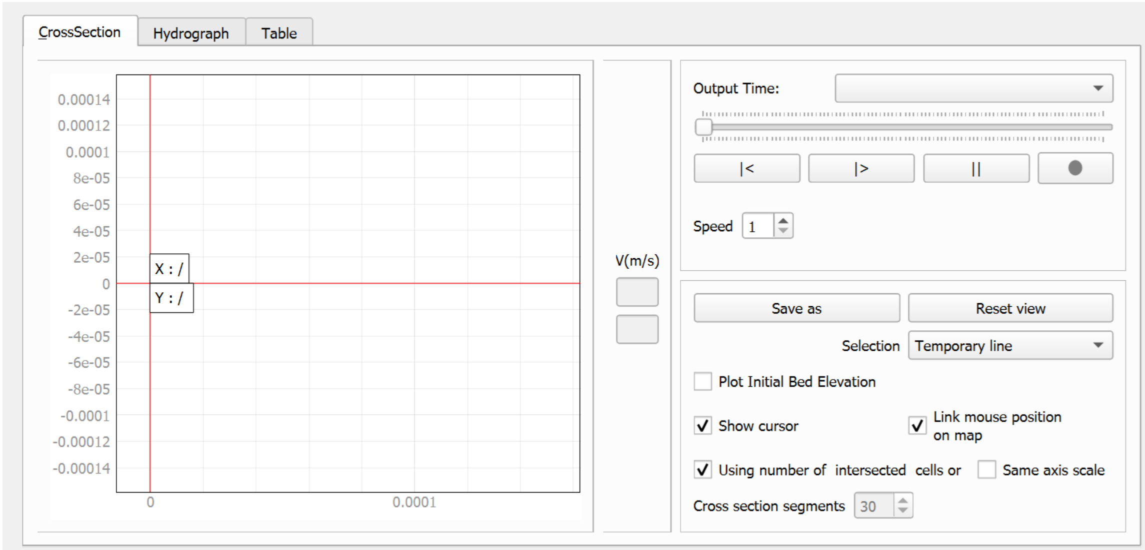


Figure 6.2 – CrossSection Tab of the Cross Sections Tool

6.1.2 Tab Controls

Control	Type	Description
Plot Frame	Frame	Main plotting area where the cross-section profile is displayed.
Velocity Legend	Frame with labels	Displays a color scale with min/max velocity values (m/s).
Output Time	Combo Box	Selects which time step to display from the simulation results.
Time Slider	Horizontal Slider	Allows scrolling through available time steps.
Animation Controls	Button Group	Includes Rewind (| <), Play (| >), and Pause (||) buttons for animating through time steps.
Video Button	Button	Records the animation as a video file.
Animation Speed	Spin Box	Controls the speed of the animation playback.
Save As	Button	Exports the current cross-section to .PNG format.
Reset View	Button	Resets the view to the default display settings.
Selection	Combo Box	Chooses between “Temporary line” and “CrossSection or Profile” modes.
Plot Initial Bed Elevation	Check Box	When checked, displays the initial bed elevation alongside the current profile.
Show Cursor	Check Box	Toggles cursor display on the profile graph.
Link Mouse Position on Map	Check Box	Synchronizes the cursor position between the profile view and the map.

Control	Type	Description
Using number of intersected cells or Cross section segments	Check Box and Spin Box	Controls the sampling density of the cross-section with options for using cell intersections or a fixed number of segments. Uncheck to set number of segments manually.
Same Axis Scale	Check Box	Maintains the same scale for both X and Y axes.

6.1.3 Workflow

To create and analyze a cross-section profile:

1. Open the Cross Sections Tool from the plugin toolbar.
2. Select the profile creation method:
   - **Temporary line (default):** Single-click to add vertices, double-click to complete the line.
   - **Existing feature:** Select "CrossSection or Profile" from the Selection dropdown and click on a line feature on the map.
3. Wait for the tool to generate the cross-section profile.
4. Use the Output Time dropdown or slider to select different time steps.
5. Use the animation controls (Rewind, Play, Pause) to animate through time steps.
6. Click the video button to record an animation of the cross-section changing over time.
7. Use the "Save As" button or right-click on the plot and select "Export…" to export the profile.

6.1.4 Requirements

- Valid RiverFlow2D project file loaded in QGIS.
- Model time-series output files (.OUTFILES index file).
- For existing feature method, a CrossSections or Profile layer with valid line features is required.
- FFmpeg installed for video recording functionality.

6.1.5 Technical Details

- The tool uses PyQtGraph for high-performance graph rendering with interactive zoom and pan capabilities.
- Terrain data sampling is performed using QGIS's raster data provider interface with coordinate transformation between map canvas CRS and layer CRS.

- Hydraulic data is extracted from model output files by processing triangular mesh cells and identifying which cells contain each point along the cross-section.
- Velocity magnitude is calculated using the Pythagorean formula: $velocity = \sqrt{u^2 + v^2}$, where u and v are velocity components.
- Sampling density can be controlled using the cell intersection method (uses natural grid intersections) or fixed segment method (divides into equally spaced segments).
- Video recordings are created using FFmpeg with customizable frame rate settings.

6.2 Hydrograph Tab

The Hydrograph tab displays time-series data at the selected cross-section location, showing how hydraulic parameters (e.g., water level, discharge) change over time. Available parameters vary based on the model's active module (basic hydraulics, sediment transport, solute transport, mud/-tailings flow, or oil spill).

6.2.1 Tab Window

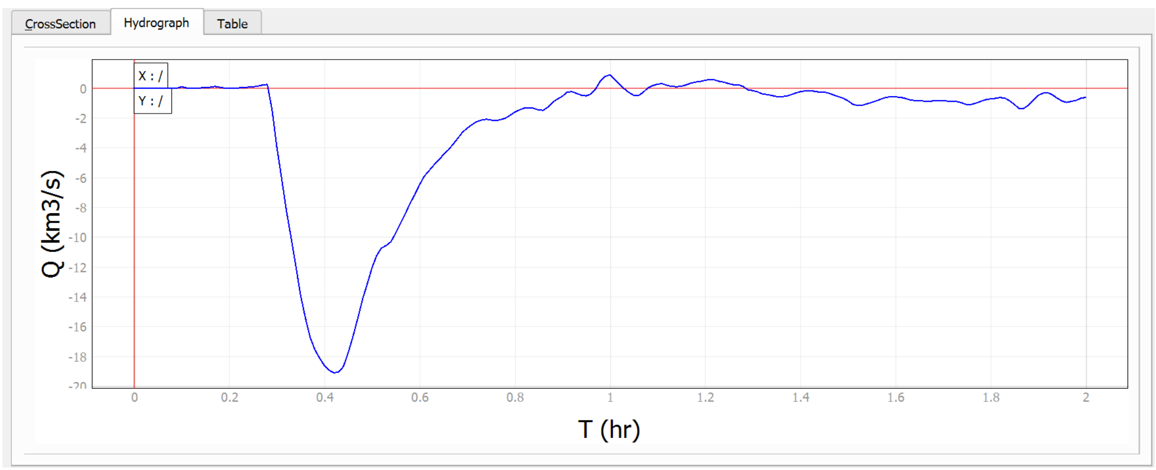


Figure 6.3 – Hydrograph Tab of the Cross Sections Tool

6.2.2 Tab Controls

Control	Type	Description
Plot Frame	Frame	Displays the time-series hydrograph data for the selected cross-section.

Available parameters in the hydrograph vary based on the model's active module:
|p3.5cm|p11cm| **Module** & **Available Parameters**
Basic Hydraulics &

- Output Time (timestamp) as column header
- Output Time (hr) - Cumulative simulation time in hours
- Discharge (m^3/s) or (km^3/s) - Flow rate at the cross-section
- Accumulated Volume (m^3) - Total volume of water that has passed through the cross-section

Sediment Transport & All Basic Hydraulics parameters, plus:

- Sediment Discharge (m^3/s) or (km^3/s) - Rate of sediment transport
- Volumetric concentration of each sediment fraction (Conc_1, Conc_2, etc.)

Solute Transport & All Basic Hydraulics parameters, plus:

- Concentration for each solute fraction(Conc_1, Conc_2, etc.)

Mud/Tailings Flow & All Basic Hydraulics parameters, plus:

- Concentration for each mud fraction (Conc_1, Conc_2, etc.)
- Total Concentration (CvTotal) - Sum of all mud fraction concentrations
- Fluid Density - Density of the mud mixture
- Fluid Dynamic Viscosity - Viscosity of the mud mixture
- Yield Stress - Minimum stress required for the mud to flow

Oil Spill on Land & All Basic Hydraulics parameters, plus:

- Oil Concentration - Concentration of oil in the water
- Oil Volume - Volume of oil at the cross-section
- Oil Mass - Mass of oil at the cross-section
- Evaporated Oil - Amount of oil lost to evaporation
- Infiltrated Oil - Amount of oil that has infiltrated into the soil

6.2.3 Workflow

To visualize and analyze hydrograph data:

1. Create a cross-section profile using the CrossSection tab (temporary line or existing feature method).
2. Switch to the Hydrograph tab.
3. The hydrograph plot will automatically display time-series data for the selected cross-section.
4. Parameters displayed depend on the model's active module (see parameter table above).
5. Use PyQtGraph's zoom and pan capabilities to examine specific time periods.
6. Right-click on the plot and select "Export…" to export the hydrograph in various formats (PNG, SVG, CSV, etc.).

6.2.4 Requirements

- A valid cross-section profile created using the CrossSection tab.
- Model output files with time-series data for all time steps.
- Index file .OUTFILES specifying all available output time steps.
- For module-specific parameters, appropriate configuration files (.seds, .solutes, .mud) must be present.

6.2.5 Technical Details

- Hydrograph values are calculated by averaging parameters across all cells intersected by the cross-section line.
- The tool reads time-specific output files: cell_time_metric_[timestamp].textout or cell_time_eng_[timestamp].tex for basic hydraulics.
- For sediment transport, cell_st_[timestamp].textout files are read.
- For solute concentrations, cell_conc_[timestamp].textout files are read.
- For mud/tailings flow, cell_mt_[timestamp].textout files are read.
- Volumetric calculations use trapezoidal integration: $Volume_t = Volume_{t-1} + \frac{Q_t+Q_{t-1}}{2} \times \Delta t$.
- Time interval Δt is read from the model's .dat configuration file.

6.3 Table Tab

The Table tab presents the cross-section data in tabular format, allowing for more precise examination of values. The table displays hydraulic parameters at each station along the cross-section, including bed elevation, water depth, water surface elevation, velocity, Froude number, and discharge.

6.3.1 Tab Window

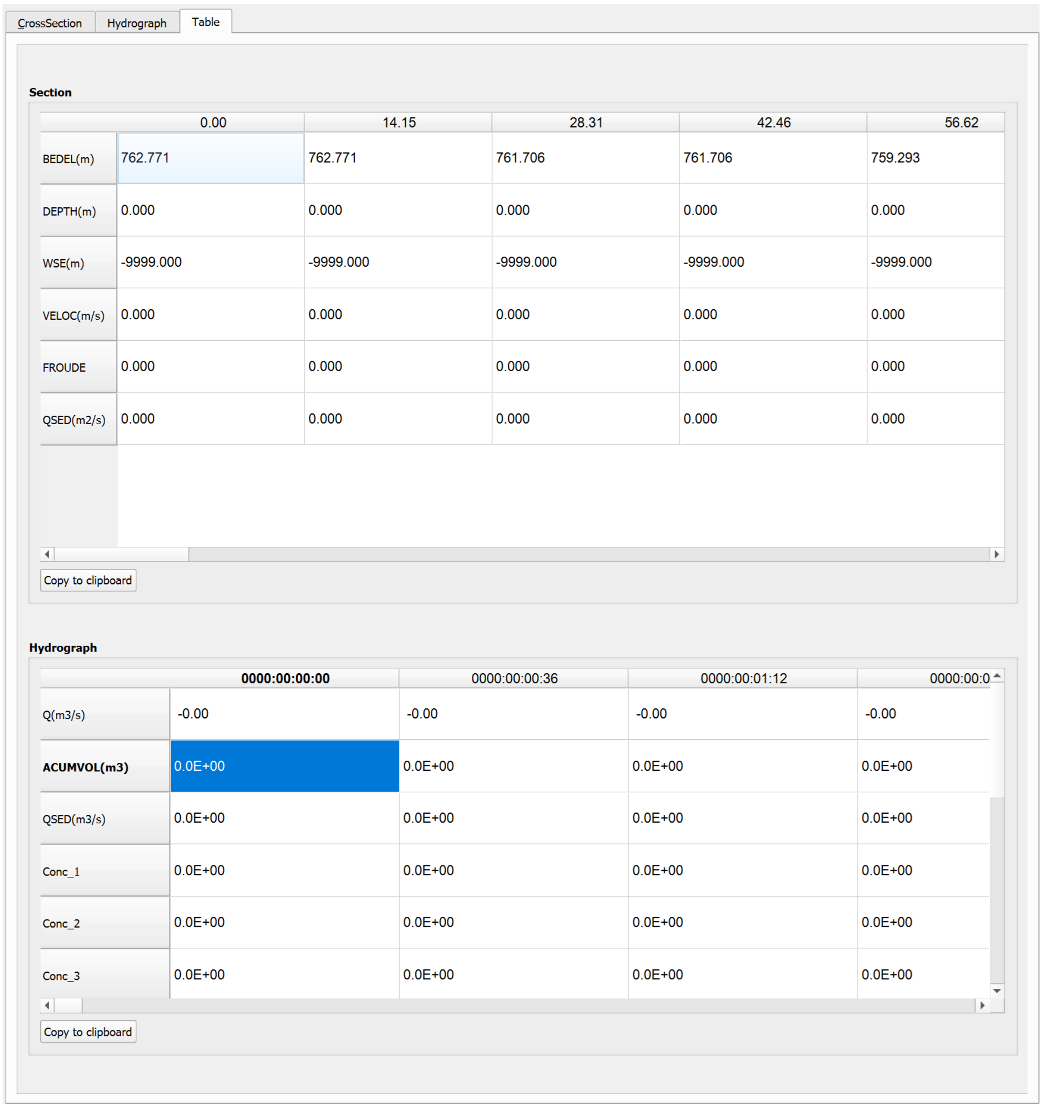


Figure 6.4 – Table Tab of the Cross Sections Tool

6.3.2 Tab Controls

Control	Type	Description
Scroll Area	Scroll Area	Contains a table view displaying all cross-section data points with their corresponding values.
Data Table	Table	Displays hydraulic parameters at each station along the cross-section, including:

Control	Type	Description
Copy to clipboard	Button	Allows users to copy the entire table of values to the system clipboard. This enables easy transfer of data to external applications such as spreadsheet software (e.g., Microsoft Excel) or text editors for further analysis or reporting. Simply click the button and then paste the data into the desired application.
Navigation Controls	Arrow keys	The keyboard arrow keys (ƒ+‘, ƒ+“, ƒ+?, ƒ+’) can also be used to navigate through the data when there are more values than can be displayed at once.

6.3.3 Workflow

To view and export tabular cross-section data:

1. Create a cross-section profile using the CrossSection tab.
2. Switch to the Table tab.
3. The table will automatically display all hydraulic parameters for each station along the cross-section.
4. Use keyboard arrow keys or the scroll bar to navigate through the data.
5. Click the “Copy to clipboard” button to copy all table values.
6. Paste the data into external applications like Excel for further analysis.

6.3.4 Requirements

- A valid cross-section profile created using the CrossSection tab.
- Model output files with hydraulic data for the selected time step.

6.3.5 Technical Details

- The table displays the following parameters for each station:
  - Bed elevation (m) - Terrain elevation
  - Water depth (m) - Water height above bed
  - Water surface elevation (m) - Bed elevation + depth
  - Velocity (m/s) - Velocity magnitude calculated as $\sqrt{u^2 + v^2}$
  - Froude number - Dimensionless number: $Fr = \frac{v}{\sqrt{gd}}$
  - Discharge (m^3/s) - Volumetric discharge

- Data copied to clipboard is in tab-separated table format, compatible with spreadsheets.
- The number of rows in the table corresponds to the number of sample points along the cross-section (determined by the selected sampling method).

6.4 Tips and Best Practices

- **Sampling method**: For most applications, the cell intersection method provides the most accurate representation of the model results. Use the fixed segment method when you need evenly spaced sample points or when comparing multiple profiles.
- **Cross-section orientation**: For best results, draw cross-sections perpendicular to the flow direction. This provides a clearer representation of the hydraulic profile.
- **Vertex density**: When creating temporary lines, use enough vertices to capture changes in topography, but avoid too many points that may slow down performance.
- **Data export**: Use the Table tab and "Copy to clipboard" function to export precise numerical data for further analysis in spreadsheets.
- **Video recording**: Set the appropriate animation speed before recording videos. Slower speeds are better for long simulations, while faster speeds are suitable for brief events.
- **Cursor linking**: Enable "Link Mouse Position on Map" to synchronize cursor position between the profile view and the map, making it easier to identify specific locations.
- **Axis scaling**: Use "Same Axis Scale" when you need to maintain correct geometric proportions in the profile. Disable it to maximize use of the display space.

7

Tools

This chapter provides a comprehensive reference for the tools available in the QGIS plugin. You can access the tools via the toolbar or the Main Menu under *Plugins*:

Figure 7.1 – Tools Menu

Figure 7.2 – Tools Menu

7.1 EPA-SWMM Tools

The plugin includes tools for integrating with EPA's Storm Water Management Model (SWMM), enabling users to create SWMM input files directly from QGIS layers and import existing SWMM files.

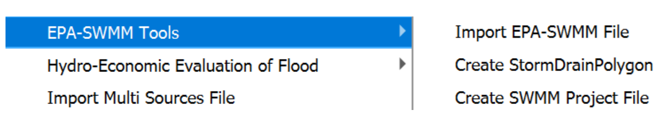


Figure 7.3 – EPA-SWMM Project Submenus

7.1.1 Dialog Window

7.1.1.1 Import EPA-SWMM File

This tool imports an existing EPA-SWMM INP or HYC file into QGIS as vector layers.

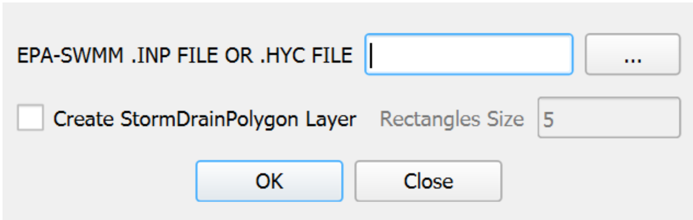


Figure 7.4 – Import EPA-SWMM File Dialog

7.1.1.2 Create StormDrainPolygon

This tool creates polygon representations around storm drain nodes.

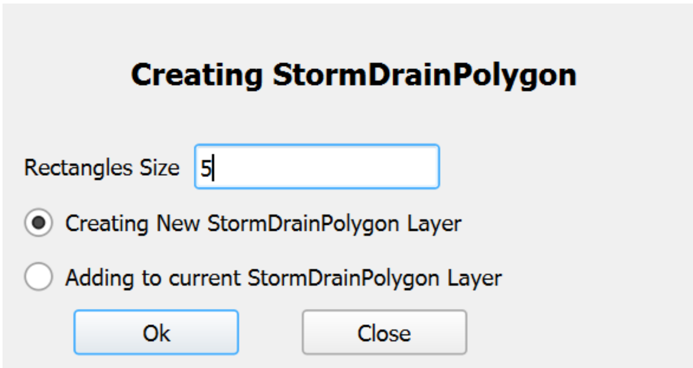


Figure 7.5 – Create StormDrainPolygon Dialog

7.1.1.3 Create SWMM Project File

This tool creates an EPA-SWMM compatible INP file from QGIS layers.

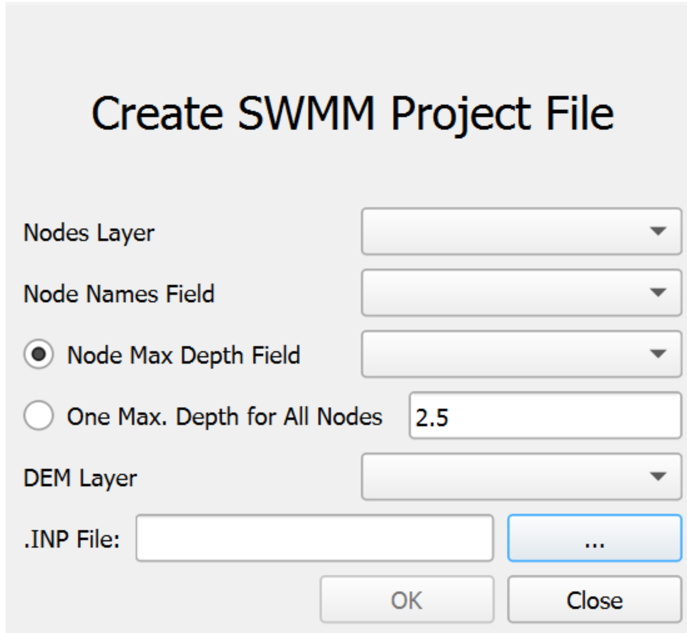


Figure 7.6 – Create SWMM Project Dialog

7.1.2 Dialog Controls

7.1.2.1 Import EPA-SWMM File

EPA-SWMM .INP FILE OR .HYC FILE	*field*	Specify the path to the SWMM input file (INP or HYC).
…	*button*	Opens a file dialog to browse for the SWMM file.
Create StormDrainPolygon Layer	*checkbox*	If checked, creates polygon representations around imported nodes.
Rectangles Size	*field*	Defines the size of the rectangular polygons (default: 5 units). Enabled only if the “Create StormDrainPolygon Layer” checkbox is checked.
Ok	*button*	Executes the import process.
Close	*button*	Closes the dialog.

7.1.2.2 Create StormDrainPolygon

Rectangle Size	*field*	Defines the size of the rectangular polygons (default: 5 units).
Creating New StormDrainPolygon Layer	*radioButton*	Removes any existing StormDrainPolygon layer and creates a new one.
Adding to current StormDrainPolygon Layer	*radioButton*	Adds new polygons to the existing StormDrainPolygon layer, if present.
OK	*button*	Executes the polygon creation.
Close	*button*	Closes the dialog.

7.1.2.3 Create SWMM Project File

Nodes Layer	*dropdown*	Selects the point layer containing SWMM junction nodes.
Node Names Field	*dropdown*	Selects the field containing unique node identifiers.
Node Max Depth Field	*radioButton*	Uses a field from the nodes layer for maximum depth.
Node Max Depth Value	*field*	Input for the maximum depth value (enabled only when the corresponding radio button is selected, with default values based on map units).
One Max. Depth for All Nodes	*radioButton*	Uses a single value for maximum depth for all nodes.
DEM Layer	*dropdown*	Selects the raster layer for elevation data.
INP File	*field*	Specifies the output location for the SWMM INP file.
…	*button*	Opens a file dialog to choose the output file location.
Close	*button*	Closes the dialog.
Ok	*button*	Executes the INP file creation.

7.1.3 Workflow

7.1.3.1 Import EPA-SWMM File

1. Checks for a RiverFlow2D project.
2. Reads the specified EPA-SWMM file (INP or HYC).
3. Extracts node information (coordinates, IDs, attributes).
4. Creates a StormDrain point layer in the project's shape directory.

5. If “Create StormDrainPolygon Layer” is checked, creates polygons around nodes.
6. Adds the layer(s) to the COMPONENTS group in QGIS.
7. Applies styling and labeling to the imported layers.

7.1.3.2 Create StormDrainPolygon

1. Checks for a RiverFlow2D project.
2. Locates the StormDrain point layer.
3. Extracts node coordinates and IDs.
4. Creates rectangular polygons based on the specified size.
5. Creates a new layer or adds to an existing one, based on the selected radio button.
6. Adds the layer to the COMPONENTS group in QGIS.
7. Labels the polygons with node IDs.

7.1.3.3 Create SWMM Project File

1. Select the nodes and DEM layers, along with the required fields in the dialog controls.
2. Define the maximum depth using a field or a single value.
3. Select the output path for the INP file.
4. Click OK to generate the SWMM project file.

7.1.4 Requirements

7.1.4.1 Import EPA-SWMM File

- An active RiverFlow2D project.
- A valid EPA-SWMM INP or HYC file.

7.1.4.2 Create StormDrainPolygon

- An active RiverFlow2D project.
- A StormDrain point layer containing the nodes.

7.1.4.3 Create SWMM Project File

- An active RiverFlow2D project.
- A nodes layer with unique identifiers.
- A DEM layer covering the node area.

7.1.5 Technical Details

7.1.5.1 Import EPA-SWMM File

File Format Support

- **INP Files:** Standard EPA-SWMM input files.
- **HYC Files:** Hydronia-specific format for storm drain networks.

Imported layers are saved in the shape directory of the current project scene.

Coordinate System Considerations The tool checks for coordinate system mismatches between the INP file and the QGIS project, warning the user if units differ.

7.1.5.2 Create StormDrainPolygon

The StormDrainPolygon layer is saved in the shape directory of the current project scene.

7.1.5.3 Create SWMM Project File

Output File Structure The generated INP file includes: TITLE, OPTIONS, EVAPORATION, JUNCTIONS, INFLOWS, REPORT, MAP, and COORDINATES sections.
See the OilFlow2D for QGIS Reference Manual Output Files section for more details.

Processing Method

1. Extracts points from the nodes layer.
2. Samples the DEM at each point.
3. Calculates invert elevations (DEM elevation - maximum depth).
4. Formats data according to SWMM INP specifications.
5. Writes the complete file.

Usage Notes

- Ensure nodes are placed appropriately in the point layer.
- The DEM should cover all junction locations.
- Output units (feet or meters) are determined by QGIS project units.
- The INP file is a basic structure, further customizable in EPA-SWMM.
- Only junctions are created; other elements (conduits, subcatchments) must be added in EPA-SWMM.

7.1.5.4 Integration with UDSWMM Module

The EPA-SWMM tool is designed to work in conjunction with the UDSWMM module for comprehensive urban drainage modeling capabilities.
See the OilFlow2D for QGIS Reference Manual Output Files section for more details.

7.2 Import Multi Sources File Tool

The Import Multi Sources File tool provides functionality for importing point source data from external files into the current project as a QGIS vector layer. This section details the tool's interface, workflow, and technical implementation.

7.2.1 Dialog Window

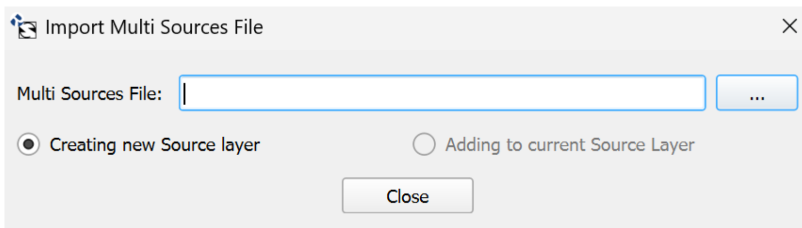


Figure 7.7 – Import Multi Sources File Dialog

7.2.2 Dialog Controls

Multi Sources File	*field*	Path to the multi-source data file to be imported.
…	*button*	Opens a file dialog to browse for the multi-source file.
Creating new Source layer	*radioButton*	When selected, creates a new Sources layer, removing any existing one.
Adding to current Source Layer	*radioButton*	When selected, adds points to the existing Sources layer (enabled only if a Sources layer exists).
Close	*button*	Closes the dialog without making changes.

7.2.3 Workflow

1. The tool first checks for an active RiverFlow2D project.
2. The user selects a multi-source file using the file dialog.
3. Based on the selected radio button option:

- If "Creating new Source layer" is selected, any existing Sources layer is removed from the project and deleted from disk.
- If "Adding to current Source Layer" is selected, the existing Sources layer is used (this option is only enabled if a Sources layer exists).

4. The tool reads the multi-source file and:
   - Extracts the number of sources from the first line.
   - For each source, reads the ID, X/Y coordinates, and associated file.
   - Creates a point feature for each source.
5. The layer is styled with red point symbols and labeled using the SOURCEID field.
6. The layer is added to the COMPONENTS group in the QGIS layer tree.
7. Configuration is applied to the layer's attribute form:
   - SOURCETYPE field is configured with a value map (1: Discharge vs Time, 2: Rating table depth vs Q).
   - FILENAME field is configured with an external resource widget.
   - A custom UI form and Python script are attached to the layer for advanced functionality.
8. A confirmation message is displayed when the import is complete.

7.2.4 Requirements

- An active RiverFlow2D project is required.
- The multi-source file must follow the specified format.
- The tool adds the layer to a COMPONENTS group in the layer tree, creating the group if it doesn't exist.
- The layer uses the project's coordinate reference system.
- The Sources layer is created in the project's shape directory.

7.2.5 Technical Details

7.2.5.1 File Format

The multi-source file should follow this format:

- First line: Number of sources (integer)
- Subsequent lines (one per source): SourceID X-coordinate Y-coordinate Filename

Example:

```
3
Source1 125.5 345.7 discharge1.txt
Source2 220.8 410.2 discharge2.txt
Source3 180.3 275.9 discharge3.txt
```

7.2.5.2 Integration with RiverFlow2D

Each point represents a source or sink where water enters or leaves the model domain. The points are associated with files that define the discharge characteristics:

- **Discharge vs Time (Type 1):** Files containing time series of discharge values.
- **Rating Table (Type 2):** Files defining the relationship between water depth and discharge.

The layer includes a custom form interface that facilitates the assignment and editing of source properties directly within QGIS.

7.3 Import RF2D Layers Tool

The Import RF2D Layers tool provides functionality for importing various RiverFlow2D components and data files into QGIS as vector and raster layers. This comprehensive tool supports multiple file formats specific to hydrodynamic modeling and allows users to integrate simulation components with GIS data.

7.3.1 Dialog Window

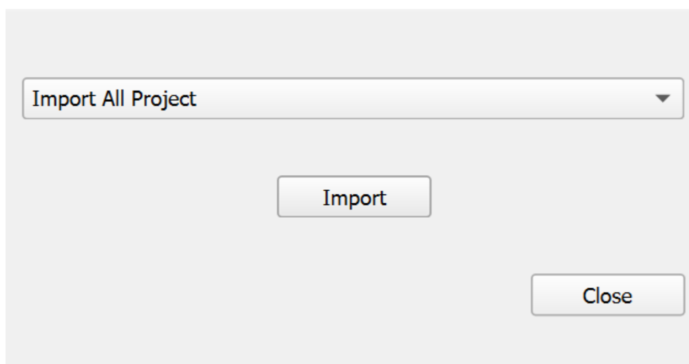


Figure 7.8 – Import RF2D Layers Dialog

7.3.2 Dialog Controls

Component Selector	*dropdown*	Dropdown list to select which RiverFlow2D component to import. Options include TriMesh, Domain Outline, Boundary Conditions, Manning's n values, and many others.
Import	*button*	Executes the import process for the selected component.
Close	*button*	Closes the dialog without making changes.

7.3.3 Workflow

1. The tool first checks for an active RiverFlow2D project and retrieves the project's coordinate reference system.
2. The user selects a component type from the dropdown list.
3. When "Import" is clicked, the tool performs one of the following workflows:
   - If "Import All Project" is selected, the tool attempts to import all available components from the project directory in sequence, checking for the existence of each file type.
   - For individual component selections, the tool opens a file dialog with the appropriate file extension filter, allowing the user to select the specific file to import.
4. During import, the tool:
   - Checks for and optionally removes any existing layer of the same type
   - Reads the selected file format and extracts geometry and attribute data
   - Creates appropriate QGIS vector or raster layers
   - Applies symbology and labeling based on the layer type
   - Organizes the layers into logical groups in the QGIS layer tree

When importing the entire project, a progress dialog displays the status of the import process. For some layer types, the tool creates a custom form interface for editing the attributes, enhancing the user's ability to modify the model parameters directly within QGIS.

7.3.4 Requirements

- An active RiverFlow2D project is required.
- The project must have a defined coordinate reference system.
- For full project import, all component files should be present in the project directory with the expected naming conventions.
- The tool requires write access to the project directory to create shapefiles for the imported layers.
- Sufficient memory is needed when importing large meshes (particularly TriMesh components with many elements).

7.3.5 Technical Details

7.3.5.1 Supported File Types

The tool supports various RiverFlow2D file types, each representing a different component of the hydrodynamic model:

Component	File Extension	Description
All Project	.FED	Imports all available RiverFlow2D components from the project directory in a single operation. Uses the FED file as the starting point to locate other project files.
TriMesh	.FED	Finite element mesh defining the computational domain.
Domain Outline	_domain_outline.exp	Polygon defining the domain boundary.
Boundary Conditions	_BoundaryConditions.exp	Lines defining where boundary conditions are applied.
Manning's n	.MannN2	Roughness zones for flow resistance.
Manning's nz	.MANNN	Directional roughness zones.
Rain/Evaporation	.LRAIN	Precipitation and evaporation zones.
Infiltration	.LINF	Soil infiltration zones.
Wind	.WIND	Wind forcing zones.
Gates	.TGATES	Hydraulic gates and structures.
Bridges	.TBRIDGES	Bridge structures within the flow domain.
Culverts	.CULVERTS	Culvert structures for flow routing.
Weirs	.TWEIRS	Weir structures for flow control.
Dam Breach	.TDAMS	Dam failure modeling parameters.
Sources/Sinks	.SOURCES	Point sources and sinks within the domain.
Cross Sections	.XSECS	River cross-sectional data.
Profiles	.PROFILES	Longitudinal profiles along rivers.
Observation Points	.OBS	Locations for time series output.
DEM	_BedElevations.exp	Bed elevation data for the domain.

7.3.5.2 Best Practices

- Before importing a new version of an existing component, it's recommended to first delete the old layer from QGIS to avoid conflicts.
- The "Import All Project" option provides the most streamlined workflow when setting up a new project or updating an entire model.
- For incremental updates or when working with specific components, use the individual import options to target only the needed data.
- After import, verify that the geometry and attributes of the imported layers match the expected values from the source files.
- For large projects, consider importing components individually rather than using "Import All Project" to better manage memory usage.

7.4 Recover Layers Tool

The Recover Layers tool provides functionality for recovering RiverFlow2D component layers that exist in the project directory but have been removed from the QGIS canvas. This tool allows users to quickly restore project layers without needing to re-import them from their source files.

7.4.1 Dialog Window

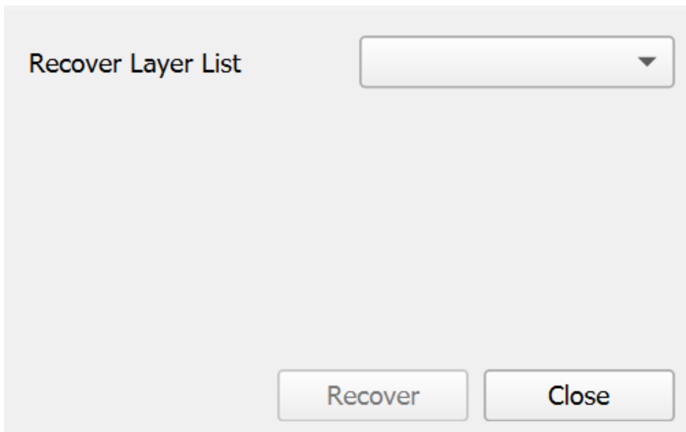


Figure 7.9 – Recover Layer Dialog

7.4.2 Dialog Controls

Recover Layer List	*dropdown*	Displays a list of project layers that exist in the project directory but are not currently loaded in QGIS.
Recover	*button*	Loads the selected layer from the project directory and adds it to the appropriate group in the QGIS layer tree.
Close	*button*	Closes the dialog without recovering any layers.

7.4.3 Workflow

1. When opened, the tool automatically scans the active RiverFlow2D project's directory structure.

2. The tool compares the layers currently loaded in QGIS against the shapefiles present in the project's shape directory.

3. Any layers that exist as files but are not currently loaded in QGIS are added to the "Recover Layer List" dropdown.

4. The user selects a layer to recover from the dropdown list.

5. When the "Recover" button is clicked, the tool:

   - Loads the shapefile from the project directory
   - Applies appropriate styling and labeling based on the layer type
   - For specific layer types like Boundary Conditions, configures custom attribute forms and widgets
   - Adds the layer to the appropriate group in the QGIS layer tree (MESH_SPATIAL_DATA, COMPONENTS, or OUTPUT_CONTROL)

6. If no layers are available for recovery, the "Recover" button is disabled.

7.4.4 Requirements

- An active RiverFlow2D project is required.
- The project must have a defined directory structure with a shape subdirectory containing component shapefiles.
- Shapefiles must follow the standard naming conventions for the tool to recognize them.
- The user must have read access to the project directory to load the shapefiles.

7.4.5 Technical Details

7.4.5.1 Supported Layer Types

The tool can recover a wide range of RiverFlow2D component layers, including:

- Domain Outline
- Mesh components (MeshBreakLine, MeshDensityLine, MeshDensityPolygon)
- Boundary Conditions
- Manning's roughness coefficients (Manning N, Manning Nz)
- Hydraulic structures (Bridges, Gates, Culverts, Weirs, DamBreach)
- Environmental components (Infiltration, RainEvap, Wind)
- Sources and Sinks
- Analysis components (CrossSections, Profiles, ObservationPoints)
- Other specialized components (MultipleDemBoundaries, InitialConcentrations, InternalRatingTable, etc.)

Each layer type is recovered with its proper styling and configuration based on predefined settings in the tool.

7.4.5.2 Best Practices

- Use this tool when you accidentally remove layers that are part of an existing project rather than reimporting them.
- After recovering layers, verify their styling and configuration to ensure they match your expectations.
- If you need to recover multiple layers, recover them one by one, selecting each from the dropdown and clicking “Recover”.
- The tool only recovers layers that still have their corresponding shapefile in the project directory; if files have been deleted, use the Import RF2D Layers tool instead.

7.5 Compare Output Raster Maps Tool

The Compare Output Raster Maps tool provides functionality for comparing raster outputs from different simulation scenarios. This tool enables quantitative assessment of differences between model runs, allowing users to evaluate the impact of parameter changes, analyze alternative designs, or validate model results against different scenarios.

7.5.1 Dialog Window

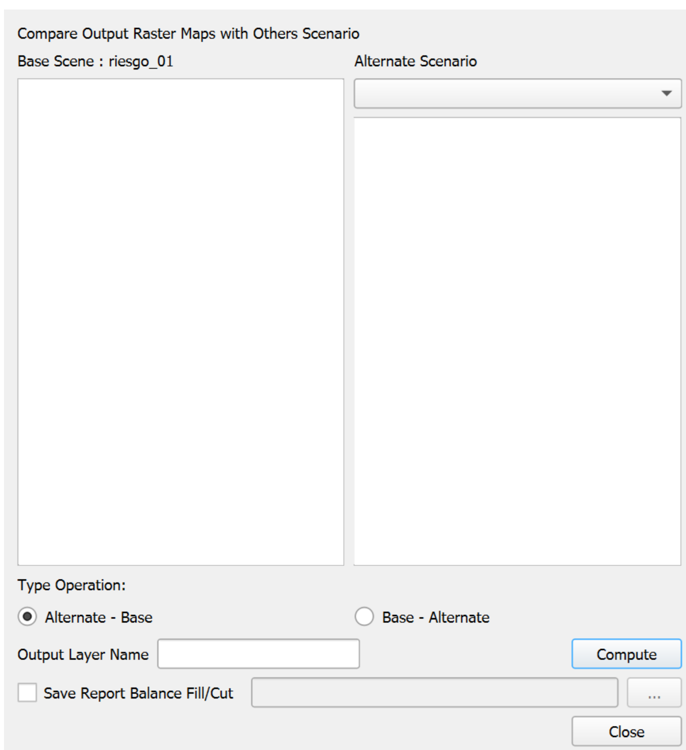


Figure 7.10 – Compare Output Raster Maps Dialog

7.5.2 Dialog Controls

Base Scenario List	*list view*	Displays all available raster layers from the current project for selection as the base scenario.
Alternate Scenario Dropdown	*combo box*	Selects the alternate scenario to compare against the base scenario.
Alternate Scenario List	*list view*	Shows available raster layers from the selected alternate scenario.
Type Operation	*radio buttons*	Specifies the comparison operation to perform: "Alternate - Base" (default) or "Base - Alternate".
Output Layer Name	*text field*	Sets the name for the resulting difference layer.
Compute	*button*	Executes the comparison operation and creates the difference layer.
Save Report Balance Fill/Cut	*checkbox*	When checked, generates a quantitative report of the differences between the maps, including volumes of fill and cut.
Report File Path	*text field*	Specifies the location to save the report file (enabled when the checkbox is checked).
Browse	*button*	Opens a file dialog to select the report file location.
Close	*button*	Closes the dialog.

7.5.3 Workflow

1. The tool first loads all available raster layers from the current active RiverFlow2D project as the base scenario.
2. Select a raster layer from the base scenario list that you want to compare.
3. From the alternate scenario dropdown, select an alternate project:
   - The dropdown is populated with available project scenarios.
   - When selected, the tool loads all raster layers from that scenario.
4. Select a corresponding raster layer from the alternate scenario list to compare against the base layer.
5. Choose the type of operation:
   - *"Alternate - Base"* shows where the *alternate* scenario has higher values than the *base* (positive differences) and where it has lower values (negative differences).

   - *"Base - Alternate"* shows where the *base* scenario has higher values than the *alternate* (positive differences) and where it has lower values (negative differences). This effectively inverts the results of the first option.

6. Enter a name for the output difference layer.

7. If you want to generate a volumetric report of the differences:

   - Check the *"Save Report Balance Fill/Cut"* option.
   - Specify a location to save the report using the text field or browse button.

8. Click "Compute" to execute the comparison operation.

9. The tool performs the following steps:

   - Loads both selected raster layers into memory.
   - Ensures they have matching extents and resolutions.
   - Performs the specified arithmetic operation (subtraction) between the layers.
   - Creates a new raster layer with the calculated differences.
   - Applies appropriate symbology to highlight positive and negative differences.
   - If requested, calculates the volumetric differences and generates a report.

10. The resulting difference layer is added to the QGIS project with a color ramp symbology.

7.5.4 Requirements

- An active RiverFlow2D project is required for the base scenario.
- Both base and alternate raster layers must:
  - Have the same coordinate reference system.
  - Have compatible extents and resolutions.
  - Represent the same type of data (e.g., water depth, velocity magnitude) for meaningful comparison.
- Write access to the project directory is needed for creating the difference layer and report file.

7.5.5 Technical Details

7.5.5.1 Difference Layer Interpretation

The resulting difference layer helps visualize and quantify changes between scenarios:

- **Positive Values (typically shown in blue):** Indicate areas where the alternate scenario has higher values than the base scenario. For water depth comparisons, these represent areas with deeper water in the alternate scenario.

- **Negative Values (typically shown in red):** Indicate areas where the alternate scenario has lower values than the base scenario. For water depth comparisons, these represent areas with shallower water in the alternate scenario.
- **Values Near Zero (typically shown in white or transparent):** Indicate areas with minimal differences between scenarios.

7.5.5.2 Volumetric Balance Report

When the “Save Report Balance Fill/Cut” option is selected, the tool generates a detailed report containing:

- **Header Information:** Date, time, project names, and layers compared.
- **Raster Statistics:** Min, max, mean, and standard deviation of difference values.
- **Fill Volume:** Total volume of positive differences (where alternate $>$ base).
- **Cut Volume:** Total volume of negative differences (where alternate $<$ base).
- **Net Volume:** The algebraic sum of fill and cut volumes.
- **Cell Details:** Number of cells with positive, negative, and zero differences.
- **Area Summary:** Total area affected by changes, broken down by positive and negative differences.

This report is useful for quantitative assessment of design alternatives, impact analysis, or verification of model changes.

7.5.5.3 Best Practices

- Compare rasters of the same type (e.g., water depth with water depth) to ensure meaningful results.
- Use descriptive output layer names that indicate which scenarios are being compared.
- For elevation or bathymetry comparisons, consider the “Alternate - Base” operation to show areas of deposition as positive and erosion as negative.
- Review both the visual difference layer and the volumetric report for comprehensive analysis.
- When comparing multiple pairs of layers, organize output layers in a group for easier management.
- Consider using transparency in the difference layer symbology to see underlying reference layers.
- For time-dependent outputs, ensure you are comparing the same time step from each scenario.

7.6 Setting TriMesh Elevation Tool

The Setting TriMesh Elevation tool provides functionality for assigning elevation values from a FED file to a TriMesh layer. This tool allows users to update the elevation attributes of mesh elements without reimporting the entire mesh, making it useful for updating terrain information in an existing model or applying elevation adjustments.

7.6.1 Dialog Window

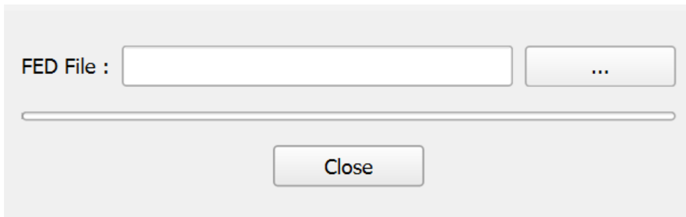


Figure 7.11 – Setting TriMesh Elevation Dialog

7.6.2 Dialog Controls

FED File	*field*	Path to the FED file containing elevation data for the mesh elements.
…	*button*	Opens a file dialog to browse for and select a FED file.
Progress Bar	*progress bar*	Shows the progress of the elevation data import process.
Close	*button*	Closes the dialog.

7.6.3 Workflow

1. The tool first checks for an active RiverFlow2D project.
2. The dialog presents a field for selecting a FED file, with a browse button to facilitate file selection.
3. When a FED file is selected using the browse button:
   - The tool verifies that a TriMesh layer exists in the current project.
   - It reads the FED file to extract elevation data for each mesh element.
   - The tool compares the number of elements in the FED file with the number of features in the TriMesh layer to ensure they match.
   - If the element counts match, the tool updates the elevation attribute (field index 5) for each mesh element with the corresponding value from the FED file.

- A progress bar visualizes the import process.
- Upon completion, a confirmation message is displayed.

4. If an error occurs during the process (e.g., no TriMesh layer found, or element count mismatch), an appropriate error message is displayed.

7.6.4 Requirements

- An active RiverFlow2D project is required.
- A TriMesh layer must exist in the QGIS project.
- The FED file must contain elevation data for all mesh elements.
- The number of elements in the FED file must match the number of features in the TriMesh layer.
- The TriMesh layer must have an attribute field at index 5 for storing elevation values.

7.6.5 Technical Details

7.6.5.1 File Format Requirements

The FED file used for setting TriMesh elevation must follow the standard RiverFlow2D FED file format:

- First line: Contains the number of elements and number of header lines.
- Header lines: Contains header information (skipped during processing).
- Element data: Each subsequent line contains element data, with the elevation value in the 6th column (index 5).

7.6.5.2 Best Practices

- Before updating elevations, ensure that the TriMesh layer is the one you want to modify, as the changes will affect the mesh geometry.
- Verify that the FED file contains appropriate elevation values before applying them to the mesh.
- After updating elevations, review the mesh to ensure the changes were applied correctly.
- Consider saving your project before using this tool, as the changes to the TriMesh layer are permanent once applied.
- For large meshes with many elements, the import process may take some time. The progress bar provides visual feedback on the operation status.

7.7 Check Internal Angle of TriMesh's Cells Tool

The Check Internal Angle of TriMesh's Cells tool provides functionality for identifying and locating problematic cells in a triangular mesh where internal angles are less than 5 degrees. Meshes with extremely small angles can cause computational instability and inaccuracies in hydrodynamic simulations, making this tool essential for quality control of the computational domain.
This tool also detects cells with null area. In these cases, the tool displays the cell ID, but because these cells have no geometry, the zoom and selection tools cannot be used. Therefore, the cell must be located manually to determine the cause, which is usually two nodes with the same coordinates.

7.7.1 Dialog Window

This tool has a minimal interface as it automatically runs the checking process when launched and displays results in a separate dialog if problematic cells are found.

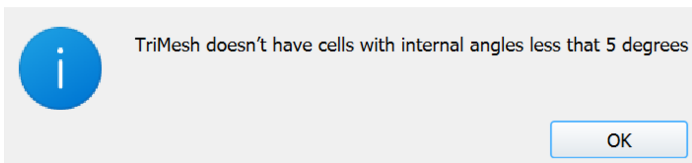


Figure 7.12 – Check Internal Angle of TriMesh's Cells negative result

7.7.2 Dialog Controls

Close	*button*	Closes the dialog after the checking process.

7.7.3 Workflow

1. When launched, the tool automatically checks for an active TriMesh layer in the project.
2. A progress dialog appears, showing the status of the checking process.
3. The tool examines each triangular cell in the TriMesh, calculating all three internal angles.
4. For each cell, the tool:
    - Extracts the coordinates of the three vertices.
    - Computes the squared distances between each pair of vertices.
    - Calculates the internal angles using the law of cosines.
    - Checks if any angle is less than 5 degrees (approximately 0.08726647 radians).
5. If any cells with problematic angles are found:
    - A list dialog opens showing the IDs of all problematic cells.

- The user can select a cell from the list and use the "Zoom to cell" button to inspect it in the map canvas.
- The selected cell is highlighted in the TriMesh layer.

6. If no problematic cells are found, an information message is displayed and the tool closes.
7. If a cell with zero area is detected during the checking process (indicating overlapping vertices), an error message is shown with the cell ID, and the checking process terminates.

7.7.4 Requirements

- An active RiverFlow2D project is required.
- A TriMesh layer must be present in the QGIS project with the name "TriMesh".
- The TriMesh layer must contain valid triangular geometry.
- The tool requires read access to the mesh data to perform angle calculations.

7.7.5 Technical Details

7.7.5.1 List of Cells Dialog

If the tool identifies cells with internal angles less than 5 degrees, it displays a secondary dialog listing these cells:

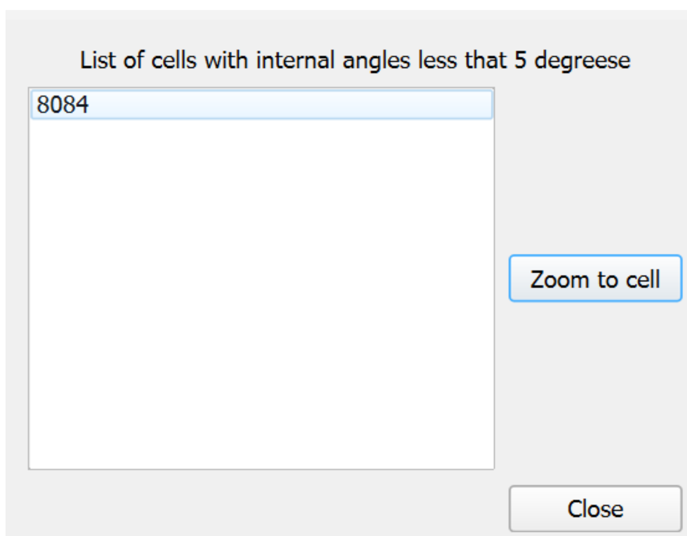


Figure 7.13 – List of cells with small internal angles

List of cells	*list view*	Displays cell IDs of triangular elements with internal angles less than 5 degrees.
Zoom to cell	*button*	Zooms the map canvas to the selected cell for visual inspection.

List of cells	*list view*	Displays cell IDs of triangular elements with internal angles less than 5 degrees.
Close	*button*	Closes the dialog.

7.7.5.2 Numerical Method

The tool uses the following approach to calculate and check internal angles:

1. For each triangular cell with vertices v_0, v_1, and v_2:
2. Calculate squared distances between vertices:
   - $d_{12}^2 = (v_1.x - v_0.x)^2 + (v_1.y - v_0.y)^2$
   - $d_{13}^2 = (v_2.x - v_0.x)^2 + (v_2.y - v_0.y)^2$
   - $d_{23}^2 = (v_2.x - v_1.x)^2 + (v_2.y - v_1.y)^2$
3. Calculate internal angles using the law of cosines:
   - $\cos(\angle v_0) = \frac{d_{12}^2 + d_{13}^2 - d_{23}^2}{2\sqrt{d_{12}^2 \cdot d_{13}^2}}$
   - $\cos(\angle v_1) = \frac{d_{12}^2 + d_{23}^2 - d_{13}^2}{2\sqrt{d_{12}^2 \cdot d_{23}^2}}$
   - $\angle v_2 = \pi - (\angle v_0 + \angle v_1)$
4. Check if any angle is less than 5 degrees (0.08726647 radians).

7.7.5.3 Best Practices

- Run this tool after creating or modifying a TriMesh to ensure the mesh quality meets numerical stability requirements.
- Use this tool before running simulations to identify and fix potential problems in the computational mesh.
- When problematic cells are identified:
  - Consider remeshing the affected areas with different mesh density parameters.
  - Manually edit the mesh to correct geometric issues.
  - If using breaklines to control mesh generation, adjust their placement to avoid creating sliver triangles.
- For optimal numerical simulation, all internal angles should ideally be between 30 and 120 degrees, though the tool only flags the most extreme cases below 5 degrees.
- Consider running this tool alongside other mesh quality checks such as ensuring smooth elevation transitions between adjacent cells.

7.8 Oil Pipeline Break Model Tool

The Oil Pipeline Break Model tool generates spill source points along an Oilpipeline polyline using break spacing, inflow conditions, and oil properties. It computes spill outflow rates for each break, samples elevations along the pipeline, and creates a Sources layer that can be used in OilFlow2D simulations.

7.8.1 Dialog Window

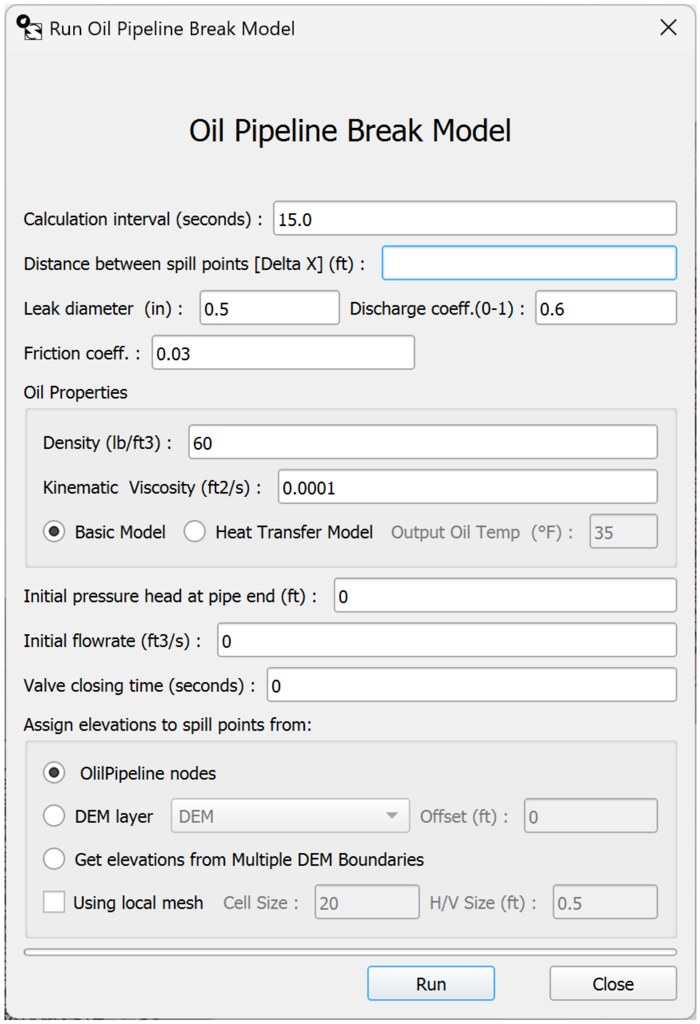


Figure 7.14 – Oil Pipeline Break Model Dialog

7.8.2 Dialog Controls

Calculation interval (seconds)	*field*	Time step used to compute spill sources.
Distance between spill points [Delta X]	*field*	Spacing between break points along the pipeline.
Leak diameter	*field*	Leak diameter used for each break point; must be smaller than the pipeline diameter.
Discharge coeff. (0-1)	*field*	Discharge coefficient for leak flow calculations.
Friction coeff.	*field*	Friction coefficient for the pipeline.
Density	*field*	Oil density.
Kinematic viscosity	*field*	Oil kinematic viscosity.
Basic Model	*radio button*	Runs the basic model without temperature output.
Heat Transfer Model	*radio button*	Runs the heat transfer model and enables output temperature.
Output temp	*field*	Output temperature for the heat transfer model.
Initial pressure head at pipe end	*field*	Initial pressure head at the pipe end.
Initial flowrate	*field*	Initial flowrate in the pipeline.
Valve closing time (seconds)	*field*	Valve closing time for internal boundary conditions.
Oilpipeline nodes	*radio button*	Uses Z values from the Oilpipeline geometry to assign elevations.
DEM layer	*radio button*	Samples elevations from a raster DEM layer.
DEM layer list	*dropdown*	Selects the DEM raster to sample.
Offset	*field*	Adds an elevation offset when sampling from a DEM.
Get elevations from Multiple DEM Boundaries	*radio button*	Samples elevations using the MultipleDemBoundaries layer.
Using local mesh	*checkbox*	Enables local mesh generation options.
Cell Size	*field*	Local mesh cell size around spill points.
H/V Size	*field*	Local mesh proportion relative to spill locations.
Run	*button*	Runs the Oil Pipeline Break Model tool.
Close	*button*	Closes the dialog without running.

7.8.3 Workflow

1. Open the OilFlow2D Tools menu and select Run Oil Pipeline Break Model.
2. Ensure the Oilpipeline layer is loaded and contains valid diameter and roughness attributes.
3. Optional: Load the OilPipelineValves layer if internal valve points are needed.

4. Set the calculation interval, break spacing (Delta X), leak diameter, discharge coefficient, and friction coefficient.

5. Enter oil properties and choose the model type (Basic or Heat Transfer). If using Heat Transfer, set the output temperature.

6. Provide initial pressure head, initial flowrate, and valve closing time.

7. Select how elevations are assigned: Oilpipeline Z values, DEM layer with offset, or Multiple DEM Boundaries.

8. If needed, enable local mesh options and set Cell Size and H/V Size.

9. Click Run. The plugin generates spill source points and adds a Sources layer to the COMPONENTS group.

10. Save the QGIS project to preserve the generated sources.

7.8.4 Requirements

- An active OilFlow2D project and scene directory must be configured.
- The Oilpipeline layer must exist and be active.
- Leak diameter must be smaller than the pipeline diameter attribute.
- If DEM sampling is used, the selected raster must cover the pipeline extent.
- If Multiple DEM Boundaries is selected, the MultipleDemBoundaries layer must exist and contain features.
- The Oil Pipeline module executable and license must be available.

7.8.5 Technical Details

7.8.5.1 Spill Hydrograph Calculation

The tool writes the pipeline geometry and user inputs to *datain.txt*, *output.txt*, and *pointZ.txt*, then runs the Oil Pipeline module to compute one discharge hydrograph for each generated spill point. The module first computes the initial steady energy head along the pipeline from the entered flow rate and pressure head, using Darcy-Weisbach pipe losses and a friction factor from either the user setting or the pipe roughness.

For each break, the pipeline is split into an upstream and downstream contributing segment. The downstream segment is treated as gravity drainage from the pipe. The upstream segment uses the leak orifice equation

$$Q_l = C_d A_l \sqrt{2 g h_p}$$

where Q_l is leak discharge, C_d is the discharge coefficient, A_l is leak area, g is gravitational acceleration, and h_p is the local pressure head. During the pre-closure and valve-closing period, the upstream inflow is held at the initial flow rate until the valve-closing start time, then reduced linearly to zero over the valve-closing time. The available pipe volume is updated at every calculation interval, and the pressure head is reduced in proportion to the remaining available volume.

7.8.5.2 Outputs

- Creates spill source files in the scene directory:
  - *datain.txt*, *output.txt*, *pointZ.txt*
  - *opbmp.txt* (parámetros guardados)
  - *<project>_spillSources.txt*
- Creates a Sources layer at *shape/Sources.shp* and adds it to the COMPONENTS group.
- When using Multiple DEM Boundaries, writes a *.MDPoly* file and converts rasters to *.asc* as needed.

7.8.5.3 Usage Notes

- If the Oilpipeline layer is missing, the tool will not run.
- If a DEM does not cover the pipeline, the tool stops with an error.
- Negative pressure warnings appear in the OF2D log messages panel after processing.
- Units are based on the current project map units (meters or feet).

7.9 HCA Impact Analysis Tool

The HCA Impact Analysis tool computes the impact of oil spills on High Consequence Areas (HCA) using spatial intersections between simulation results and HCA/line datasets (NHD). The result includes intersection tables and, optionally, a point layer with spill metrics per source.

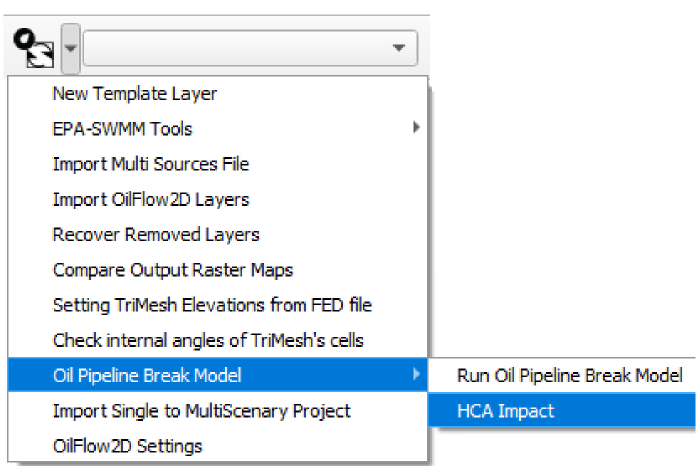


Figure 7.15 – Accessing HCA Impact from the Tools menu

7.9.1 Dialog Window

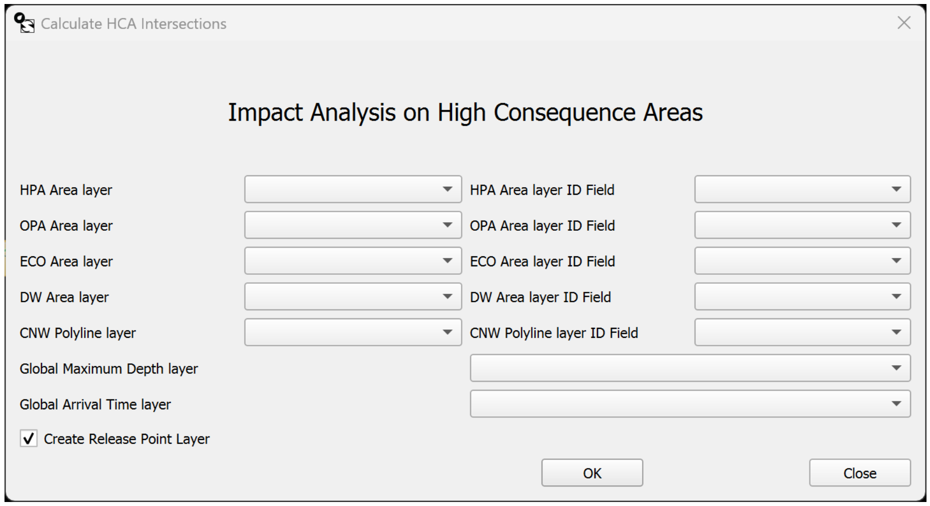


Figure 7.16 – HCA Impact Analysis Dialog

7.9.2 Dialog Controls

HPA Area layer	*dropdown*	Selects the HPA polygon layer (NHDArea).
HPA Area layer ID Field	*dropdown*	Identifier field for the HPA layer (recommended: OBJECTID).
OPA Area layer	*dropdown*	Selects the OPA polygon layer.
OPA Area layer ID Field	*dropdown*	Identifier field for the OPA layer (recommended: OBJECTID).
ECO Area layer	*dropdown*	Selects the ECO polygon layer.
ECO Area layer ID Field	*dropdown*	Identifier field for the ECO layer (recommended: OBJECTID).
DW Area layer	*dropdown*	Selects the DW polygon layer.
DW Area layer ID Field	*dropdown*	Identifier field for the DW layer (recommended: OBJECTID).
CNW Polyline layer	*dropdown*	Selects the CNW polyline layer (NHDLine).
CNW Polyline layer ID Field	*dropdown*	Identifier field for the CNW layer (recommended: OBJECTID).
Global Maximum Depth layer	*dropdown*	Integrated global maximum depth map.
Global Arrival Time layer	*dropdown*	Integrated frontal wave arrival time map.
Create Release Point Layer	*checkbox*	Generates the releasePoint layer based on the Sources layer.
OK	*button*	Runs the HCA impact analysis.
Close	*button*	Closes the dialog without running.

7.9.3 Workflow

1. Ensure simulation results and integrated *Global Maximum Depth* and *Global Arrival Time* maps are available from the hydrodynamic mapping tools.
2. Open OilFlow2D Tools → Oil Pipeline Break Model → HCA Impact.
3. Select the HPA/OPA/ECO/DW layers and the CNW layer, along with their ID fields (OBJECTID recommended).
4. Select at least one depth or arrival time map; the analysis requires at least one of these.
5. Optionally enable *Create Release Point Layer* to generate the releasePoint layer.
6. Click OK to run the analysis. Intersection layers and tables are created in the TABLES group.

7.9.4 Requirements

- An active OilFlow2D project with spill results and a Sources layer.
- Integrated *Global Maximum Depth* and/or *Global Arrival Time* maps.
- NHDArea and NHDLine layers with an OBJECTID field (or equivalent) for identifiers.
- At least one HCA/line layer selected with no duplicate selections in the dialog.
- If NHD layers are sourced from a geodatabase/REST service, reproject them to the project CRS and convert curved geometries to straight segments before running the analysis.

7.9.5 Technical Details

7.9.5.1 Outputs

- Creates tables in the *TABLES* group: *Spills-HPAAreas*, *Spills-OPAAreas*, *Spills-ECOAreas*, *Spills-DWAreas*, and *Spills-CNWLine*.
- Optional: creates the *releasePoint* layer with spill metrics per source.
- Links *releasePoint* to the HCA tables for Identify Features queries.

7.9.5.2 Key Results

Key fields in *releasePoint* include: SPILLID, DRAINUPVOL, DRAINDWVOL, SPILLVOL, SPILLAREA, MAXQ, MEANQ, DRAINTIME, and NHD_Intersected (yes/no). The Spills-NHDArea tables include volume, area, and min/max arrival times per area; the Spills-NHDLine table includes intersection length per segment. Units depend on the project CRS.

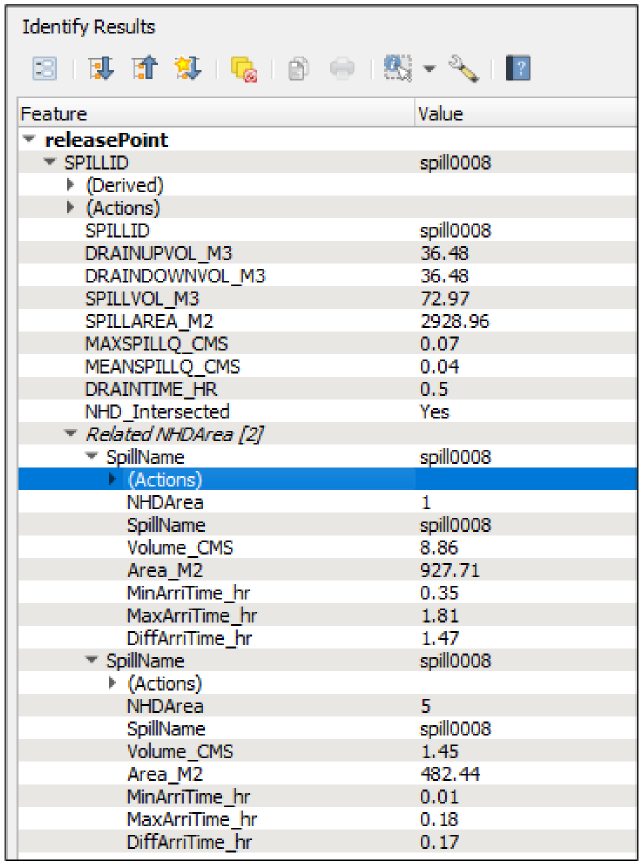


Figure 7.17 – Querying relationships from releasePoint

7.9.5.3 Usage Notes

- If the NHDArea/NHDLine layers are not named as expected, select them manually in the dialog.
- The analysis requires at least one HCA/line layer and at least one depth or arrival time layer.
- Do not select the same layer in more than one dialog field.

7.10 RiverFlow2D Settings Tool

The RiverFlow2D Settings Tool provides a centralized interface for configuring essential parameters required for RiverFlow2D project operations. This tool allows users to specify the RiverFlow2D executable location, set project paths, and configure mesh processing options that affect simulation performance and accuracy.

7.10.1 Dialog Window

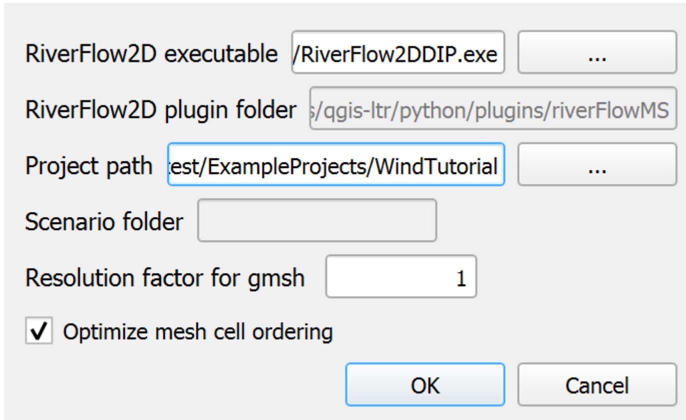


Figure 7.18 – RiverFlow2D Settings Dialog

7.10.2 Dialog Controls

RiverFlow2D executable	*text field*	Path to the RiverFlow2D executable file (.exe) on the system.
Browse (executable)	*button*	Opens a file dialog to select the RiverFlow2D executable file.
Project Path	*text field*	Directory path where project files are stored.
Browse (project path)	*button*	Opens a directory dialog to select the project folder.
Project File	*text field*	Name of the project file without extension.
Browse (project file)	*button*	Opens a file dialog to select a .dat project file.
Resolution factor for gmsh	*text field*	Numeric value that determines mesh element size during gmsh processing. Default is 1.
Optimize mesh cell ordering	*checkbox*	When checked, enables optimization of mesh cell numbering to improve computational efficiency.
OK	*button*	Saves the settings and closes the dialog.
Cancel	*button*	Closes the dialog without saving changes.

7.10.3 Workflow

1. When the tool is launched, it verifies that a RiverFlow2D project has been set up. If no project is configured, a warning message appears, and the dialog will not open.
2. If a project exists, the dialog opens and loads the current configuration from both:
   - QGIS application settings (for the RiverFlow2D executable path)
   - Project-specific settings (for project paths and mesh settings)

3. The user can modify any of the following settings:
   - **RiverFlow2D executable**: Path to the simulation software executable file, typically located in the RiverFlow2D installation directory.
   - **Project Path**: Directory where all project files will be saved.
   - **Project File**: Base name for the project file (without extension).
   - **Resolution factor for gmsh**: Numeric value that controls mesh element sizing during mesh generation with gmsh. Higher values produce finer meshes.
   - **Optimize mesh cell ordering**: Option to renumber mesh cells for improved computational performance.
4. After making the desired changes, clicking "OK" saves all settings:
   - The RiverFlow2D executable path is saved in the QGIS application settings.
   - Project path, project file name, resolution factor, and mesh optimization setting are saved in the current QGIS project.
5. The saved settings are then used by other tools in the plugin, including mesh generation, simulation runs, and results processing.

7.10.4 Requirements

- A valid RiverFlow2D installation is required on the system.
- The user must have appropriate permissions to access and execute the RiverFlow2D executable.
- The project path directory must exist and be writable by the user.
- If specifying a project file, the corresponding .dat file should be valid if it already exists.
- When using the resolution factor for gmsh, numeric values between 0.1 and 10 are recommended for most applications.

7.10.5 Technical Details

7.10.5.1 Configuration Details

RiverFlow2D Executable Path
The executable path is critical as it allows the plugin to launch RiverFlow2D for simulations. The default location is typically:

```
C:/Program Files/Hydronia/RiverFlow2D/RiverFlow2DDIP.exe
```

Resolution Factor for gmsh
The resolution factor controls the element size in the mesh generation process:

- Values less than 1 create coarser meshes with larger elements (faster computation, lower accuracy).

- Values greater than 1 create finer meshes with smaller elements (slower computation, higher accuracy).
- The default value of 1 maintains the original resolution derived from the elevation data.

Mesh Cell Optimization
The "Optimize mesh cell ordering" option improves computational efficiency by:

- Reordering mesh cells to minimize the bandwidth of the system matrix.
- Reducing memory requirements during simulation.
- Potentially decreasing simulation time, especially for large meshes.

7.10.5.2 Best Practices

If instructed to do so or you are sure about the settings, you can change the settings to help solve a problem. If you are unsure about the settings, please contact Hydronia Support.

- Sometimes QGIS will misplace the path of the project file, setting it here will help the plugin find the project file in those cases.
- When working with large domains:
  - Start with a lower resolution factor (0.5 - 0.8) for initial testing and calibration.
  - Gradually increase the resolution factor for final production runs.
- Always enable mesh cell optimization for large meshes (over 500,000 elements) to improve computational efficiency.
- Document the resolution factor used for each simulation to ensure reproducibility of results.

7.11 Multi-hydrograph to Single-hydrograph Files Tool

The Multi-hydrograph to Single-hydrograph Files Tool provides a specialized function for converting time-series hydrograph data from a combined multi-source file format into individual single-hydrograph files. This tool is particularly useful for preparing hydrologic inputs for simulations when working with multiple inflow sources that were initially stored in a unified data file.

7.11.1 Dialog Window

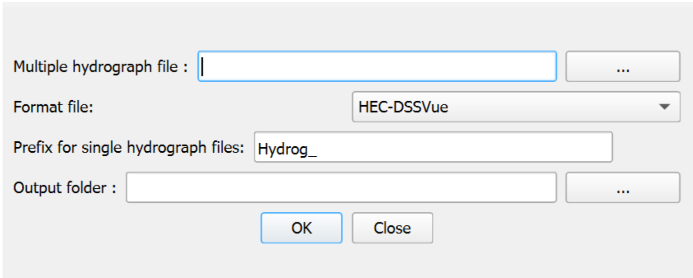


Figure 7.19 – Multi-hydrograph to Single-hydrograph Files Dialog

7.11.2 Dialog Controls

Multiple hydrograph file	*text field*	Path to the input file containing multiple hydrograph data.
Browse	*button*	Opens a file dialog to select the input multiple hydrograph file.
Format file	*dropdown*	Specifies the format of the input file (currently supports HEC-DSSVue format).
Prefix for single hydrograph files	*text field*	Text prefix to be added to all generated output files (default: "Hydrog_").
Output folder	*text field*	Directory where the individual hydrograph files will be saved.
Browse (output folder)	*button*	Opens a directory dialog to select the output folder.
OK	*button*	Processes the input file and generates the individual hydrograph files.
Close	*button*	Cancels the operation and closes the dialog.

7.11.3 Workflow

1. When launched, the tool automatically sets the default output directory to the current project's scene directory.
2. The user selects a multiple hydrograph input file by either:
   - Typing the full path in the "Multiple hydrograph file" field, or
   - Clicking the "Browse" button to navigate to and select the file.
3. The tool automatically populates the "Output folder" field with the directory containing the selected input file, but the user can specify a different output location if desired.
4. The user may customize the prefix for the generated files by modifying the "Prefix for single hydrograph files" field. The default prefix is "Hydrog_".
5. The user selects the appropriate format from the "Format file" dropdown. Currently, the tool supports HEC-DSSVue formatted files.
6. After configuring all settings, the user clicks "OK" to process the file. The tool:
   - Reads the multi-hydrograph input file
   - Identifies the number of separate hydrographs contained within
   - Creates individual files for each hydrograph
   - Names each file using the specified prefix followed by a sequential number
   - Displays a summary message showing how many files were created
7. The tool displays a completion message indicating the number of hydrograph files that were successfully created.

7.11.4 Requirements

- An active RiverFlow2D project is recommended for proper file path management, though not strictly required.
- The input multi-hydrograph file must be properly formatted according to the HEC-DSSVue standard, with time values in the expected columns.
- The user must have write permissions for the selected output directory.
- For HEC-DSSVue format, the file must contain:
  - Time data in column 5 (in hours:minutes format, e.g., “08:30”)
  - Flow values for different hydrographs starting from column 6

7.11.5 Technical Details

7.11.5.1 File Processing Details

The tool processes HEC-DSSVue formatted files using the following approach:

1. The file is scanned to determine the number of data rows and columns.
2. The tool identifies how many separate hydrographs are present by counting the number of data columns after the time column (starting from column 6).
3. For each identified hydrograph, a separate output file is created with the naming pattern: [prefix][number].txt (e.g., “Hydrog_1.txt”, “Hydrog_2.txt”).
4. Each output file is structured with:
   - A header line containing the number of time steps
   - Data rows with two columns: cumulative time (in hours) and flow value
5. The tool converts time data from the HEC-DSSVue format (HH:MM) to decimal hours for the output files, calculating cumulative time as it processes each row.

7.11.5.2 Best Practices

- Use this tool early in the RiverFlow2D project setup process when preparing input data.
- Verify that the generated single-hydrograph files have the correct number of time points and appropriate time intervals before using them in simulations.
- Keep the original multi-hydrograph file as a backup in case you need to regenerate the individual files with different settings.
- When managing multiple inflow sources, use descriptive prefixes to clearly identify the purpose of each hydrograph set (e.g., “Inflow_”, “Tributary_”).
- If creating multiple sets of hydrograph files, consider organizing them in separate directories to avoid confusion.

- After generating the hydrograph files, use the RiverFlow2D Source tool to properly assign them to source points in your model.

7.12 Import Single to Multi-scenario Project Tool

This tool facilitates the migration of standalone RiverFlow2D projects into the multi-scenario framework, allowing users to organize and compare multiple simulation scenarios within a single QGIS project.

7.12.1 Dialog Window

Figure 7.20 – Import Single to Multi-scenario Project Dialog

7.12.2 Dialog Controls

.DAT File Name	*text field*	Read-only field displaying the path to the selected RiverFlow2D project file (.dat).
Browse	*button*	Opens a file dialog to select the RiverFlow2D project file (.dat) to import.
OK	*button*	Initiates the import process after confirming the selected project file.
Cancel	*button*	Closes the dialog without importing any project.

7.12.3 Workflow

1. When launched, the tool checks if a multi-scenario project structure already exists. If not, it will prompt the user to create one first.

2. The user selects a RiverFlow2D project file (.dat) by clicking the browse button, which opens a file selection dialog. Only files with the .dat extension will be shown.

3. After selecting a valid project file, its path appears in the ".DAT File Name" field.

4. When the user clicks "OK", the tool performs the following operations:

- Verifies that the selected file is a valid RiverFlow2D project file.
- Creates a new scenario folder within the multi-scenario project structure.
- Copies all relevant project files from the source location to the new scenario folder.
- Updates file paths in the project to reflect the new location.
- Registers the new scenario in the multi-scenario project configuration.
- Updates layer data sources to point to the new file locations.

5. Upon completion, a success message is displayed, and the imported project becomes available as a scenario in the multi-scenario project.
6. If errors occur during the import process (e.g., invalid file structure, permissions issues), appropriate error messages are shown.

7.12.4 Requirements

- A valid multi-scenario RiverFlow2D project structure must exist or be created before using this tool.
- The source project must be a valid RiverFlow2D project with a proper .dat file and associated data files.
- The user must have read permissions for the source project files and write permissions for the multi-scenario project directory.
- Sufficient disk space must be available to copy all project files to the new location.
- The QGIS project should be saved before performing the import operation to ensure all current settings are properly maintained.

7.12.5 Technical Details

7.12.5.1 File Operations

The tool performs several file system operations during the import process:

1. **Project Structure Validation**: Checks that both the source project and target multi-scenario structure are valid.
2. **Scenario Name Generation**: Creates a unique scenario name based on the source project name (the *.dat file name), avoiding conflicts with existing scenarios.
3. **Directory Creation**: Creates a new scenario directory within the multi-scenario project structure.
4. **File Copying**: Recursively copies all project files from the source location to the new scenario directory, preserving the directory structure.
5. **Path Reconfiguration**: Updates all file paths within the project files to reflect the new location in the multi-scenario structure.

6. **Layer Source Updates**: Modifies QGIS layer data sources to point to the files in the new location.

The tool preserves the entire project structure, including:

- Project definition files (.dat, .qgs, .qgz)
- Mesh files (.2dm, .msh)
- Input data files (hydrographs, boundary conditions)
- Simulation output files (if present)
- Reference data and GIS layers

7.12.5.2 Best Practices

- Before importing a project, ensure it runs correctly as a standalone project to avoid transferring configuration issues.
- When working with large projects containing significant output data, consider removing unnecessary simulation results before importing to reduce the import time and storage requirements.
- After importing, verify that all layers and references are correctly linked in the new scenario.
- Keep a backup of the original standalone project until you have confirmed the imported scenario functions correctly.

8

Hydronia Tools Context Menus

This chapter documents the Hydronia tools that are only available by right-clicking supported layers in the QGIS Layers panel. The context menu appears as *Hydronia tools*. Each tool is activated from the menu, then used by left-clicking a feature in the map canvas. Right-clicking exits the tool.

8.1 Boundary Conditions HydroGraphs

This tool plots boundary condition hydrographs from the current scenario.

8.1.1 Dialog Window

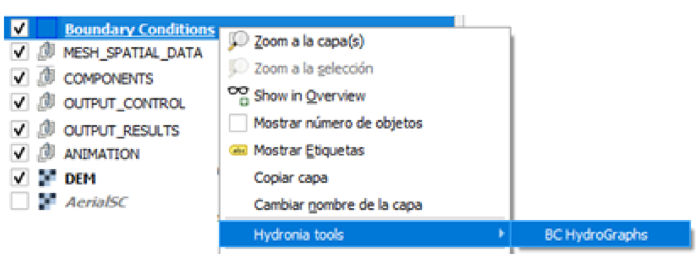


Figure 8.1 – Boundary Conditions Context Menu

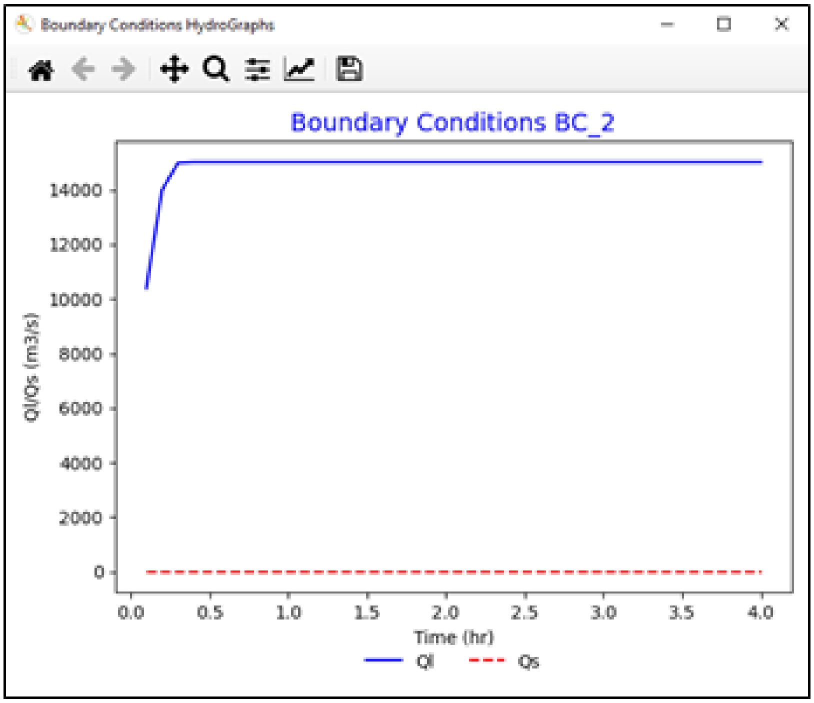


Figure 8.2 – Boundary Conditions HydroGraph

8.1.2 Dialog Controls

All Hydronia context-menu plots use Matplotlib dialogs and share the same navigation toolbar behavior. The toolbar provides the standard Matplotlib controls:

- **Home:** Resets the view to the original extents.
- **Back/Forward:** Steps through prior zoom and pan states.
- **Pan:** Click-drag to move the plot; scroll wheel zooms when supported by the backend.
- **Zoom:** Drag a rectangle to zoom into a region.
- **Save:** Saves the current plot view to an image file (PNG is supported by default).

Some plot dialogs include additional buttons tied to the plot content. For example, ObservationPoints plots allow loading measured data from a two-column text file to overlay on the model results, and a variable selector lets you choose which series the measured data represents. When these controls are present, the plot refreshes in place using the same toolbar for navigation and saving.

8.1.2.1 Figure Options

Many plot windows include a small button that opens the Figure Options dialog. This dialog provides quick access to styling and labeling controls, organized by tabs:

- **Axes Tab:** Edit axis titles, labels, ranges, and tick formatting.
- **Curves Tab:** Adjust line styles, colors, markers, and visibility for plotted series.

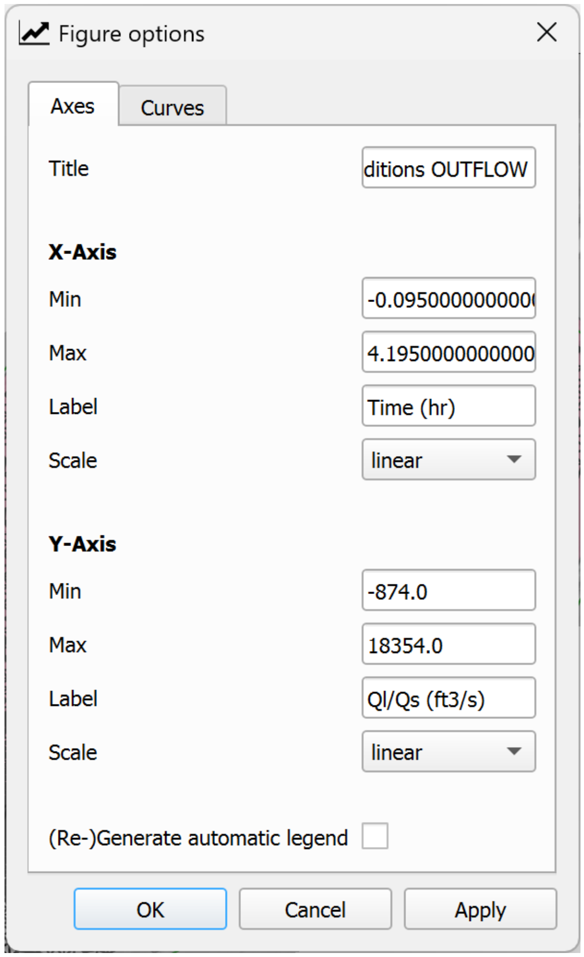


Figure 8.3 – Figure Options: Axes Tab

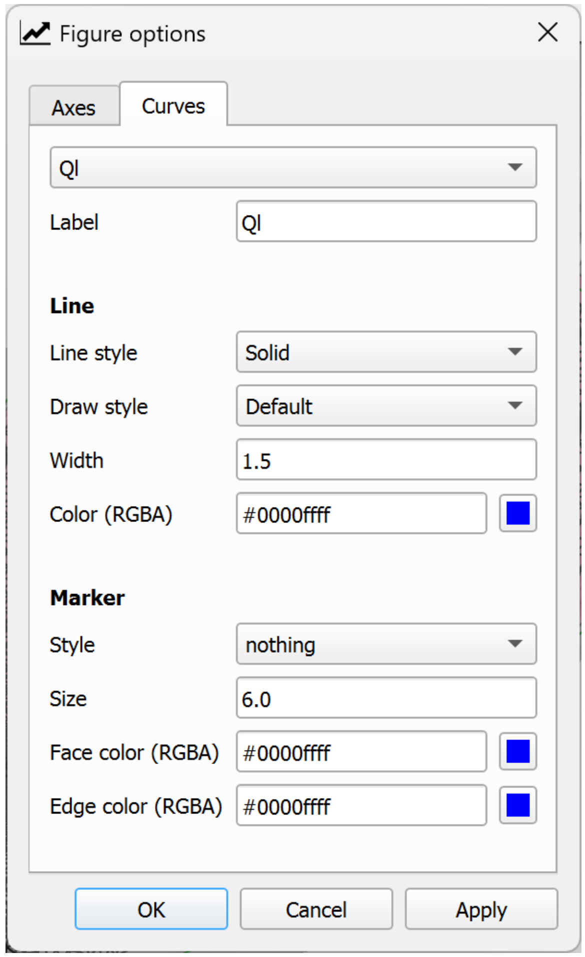


Figure 8.4 – Figure Options: Curves Tab

8.1.2.2 Configure Subplots

Some plots expose a button that opens the Configure Subplots dialog for fine control of margins and spacing. Use it to adjust the plot layout before saving.

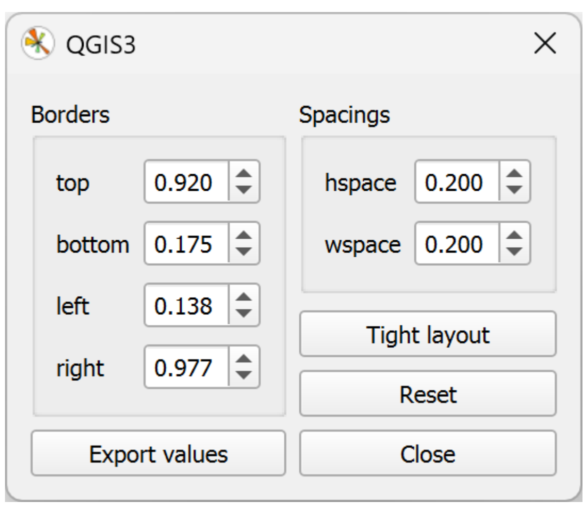


Figure 8.5 – Configure Subplots Dialog

8.1.3 Workflow

1. Right-click the *Boundary Conditions* layer and choose *Hydronia tools BC HydroGraphs*.
2. Left-click a boundary condition polygon in the map canvas.
3. The tool reads the matching data from the scenario .ROUT file and plots inflow and outflow hydrographs.
4. Right-click to exit the tool.

8.1.4 Requirements

- Valid *Boundary Conditions* layer with features.
- Model output file: <scene>.ROUT

8.1.5 Technical Details

- The tool identifies the closest boundary condition to the clicked point using the feature's ID attribute.
- Hydrograph values are read from the .ROUT file, which contains time series of inflow (QI) and outflow (Qs) discharge for each boundary condition.
- Units are automatically determined based on project settings (m^3/s or ft^3/s).

8.2 Bridges HydroGraphs

This tool plots discharge and water surface elevation through a selected bridge.

8.2.1 Dialog Window

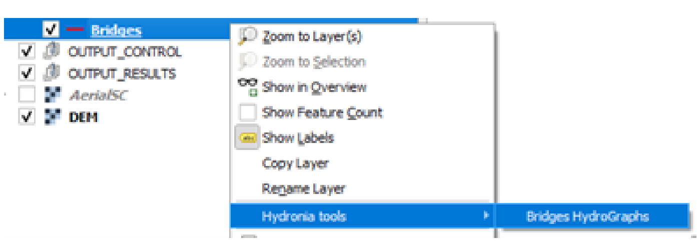


Figure 8.6 – Bridges Context Menu

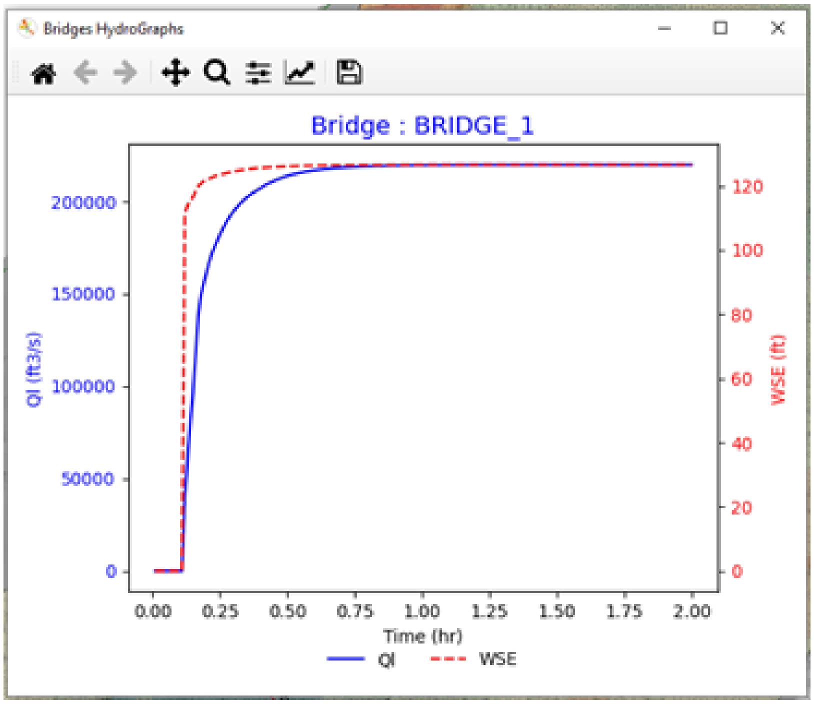


Figure 8.7 – Bridges HydroGraph

8.2.2 Dialog Controls

This tool uses standard Matplotlib plot controls. See Section 7.1.2 for details on navigation controls, Figure Options, and Configure Subplots.

8.2.3 Workflow

1. Right-click the *Bridges* layer and choose *Hydronia tools Bridges HydroGraphs*.
2. Left-click a bridge polyline in the map canvas.
3. The tool reads the corresponding bridge output file and plots discharge and water surface elevation.
4. Right-click to exit the tool.

8.2.4 Requirements

- Valid *Bridges* layer with features.
- Model output file: BRIDGE_<ID>.OUT

8.2.5 Technical Details

- The tool identifies the closest bridge to the clicked point using the feature's ID attribute.
- Each bridge has its own output file containing time series of discharge and water surface elevation.

8.3 Culverts HydroGraphs

This tool plots discharge and water surface elevation at the inlet and outlet of a culvert.

8.3.1 Dialog Window

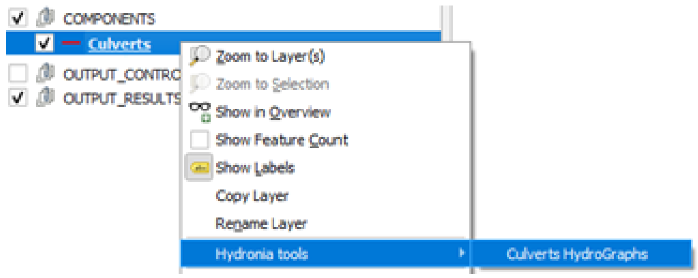


Figure 8.8 – Culverts Context Menu

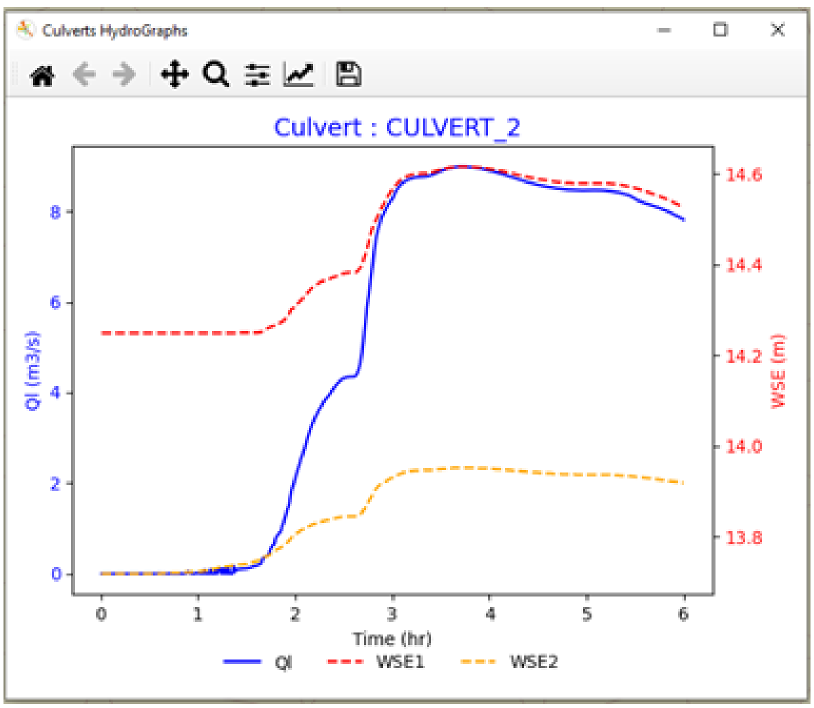


Figure 8.9 – Culverts HydroGraph

8.3.2 Dialog Controls

This tool uses standard Matplotlib plot controls. See Section 7.1.2 for details on navigation controls, Figure Options, and Configure Subplots.

8.3.3 Workflow

1. Right-click the *Culverts* layer and choose *Hydronia tools Culverts HydroGraphs*.

2. Left-click a culvert polyline in the map canvas.

3. The tool reads the corresponding culvert output file and plots discharge, inlet WSE, and outlet WSE.

4. Right-click to exit the tool.

8.3.4 Requirements

- Valid *Culverts* layer with features.
- Model output file: CULVERT_<ID>.OUT
- Project must use metric units (this tool is not available with English units).

8.3.5 Technical Details

- The tool identifies the closest culvert to the clicked point using the feature's ID attribute.
- Each culvert has its own output file containing time series of discharge and water surface elevations at inlet and outlet.
- The plot displays three series: discharge (Q), inlet water surface elevation (inlet WSE), and outlet water surface elevation (outlet WSE).

8.4 Gates HydroGraphs

This tool plots the discharge through a selected gate.

8.4.1 Dialog Window

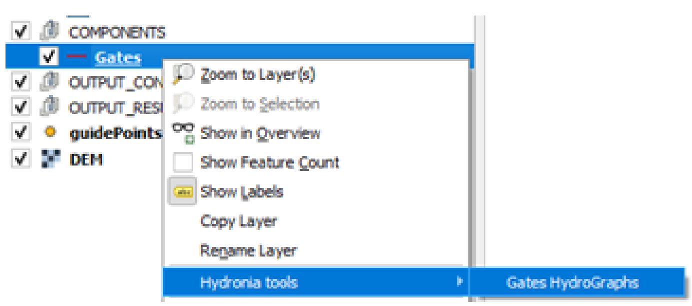


Figure 8.10 – Gates Context Menu

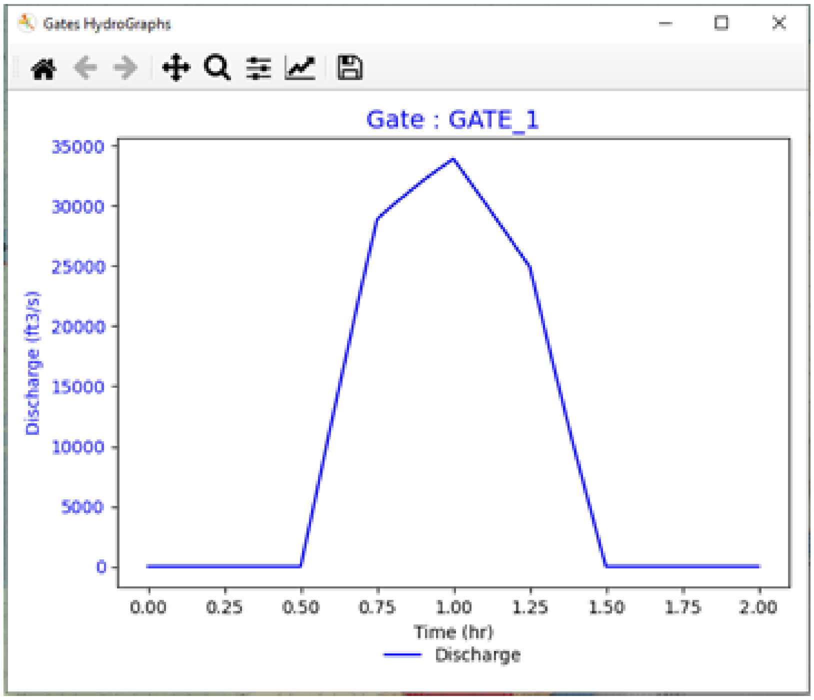


Figure 8.11 – Gates HydroGraph

8.4.2 Dialog Controls

This tool uses standard Matplotlib plot controls. See Section 7.1.2 for details on navigation controls, Figure Options, and Configure Subplots.

8.4.3 Workflow

1. Right-click the *Gates* layer and choose *Hydronia tools Gates HydroGraphs*.
2. Left-click a gate polyline in the map canvas.
3. The tool reads the gate output file and plots the discharge hydrograph for the selected gate.
4. Right-click to exit the tool.

8.4.4 Requirements

- Valid *Gates* layer with features.
- Model output file: <scene>.GATEH

8.4.5 Technical Details

- The tool identifies the closest gate to the clicked point using the feature's ID attribute.
- Data for all gates is stored in a single .GATEH file, and the tool extracts the time series corresponding to the selected gate's ID.

8.5 StormDrain HydroGraphs

This tool plots inflow, flooding, and water depth for a storm drain node.

8.5.1 Dialog Window

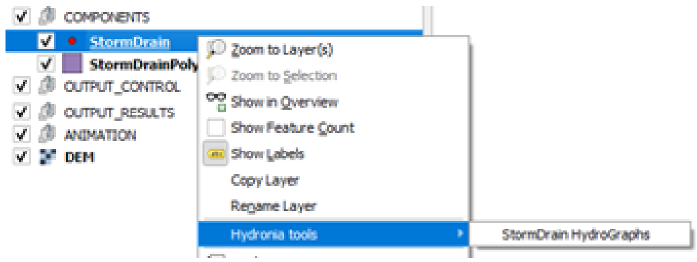


Figure 8.12 – StormDrain Context Menu

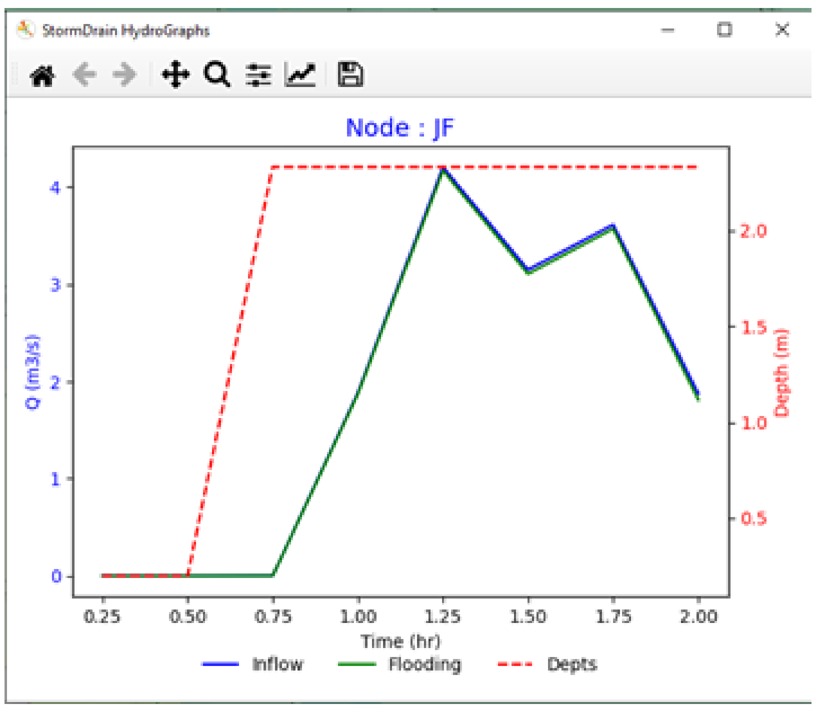


Figure 8.13 – StormDrain HydroGraph

8.5.2 Dialog Controls

This tool uses standard Matplotlib plot controls. See Section 7.1.2 for details on navigation controls, Figure Options, and Configure Subplots.

8.5.3 Workflow

1. Right-click the *StormDrain* layer and choose *Hydronia tools StormDrain HydroGraphs*.

2. Left-click a StormDrain node in the map canvas.
3. The tool reads the corresponding SWMM node output and plots inflow, flooding, and depth over time.
4. Right-click to exit the tool.

8.5.4 Requirements

- Valid *StormDrain* layer with features.
- Model output files:
  - <scene>.inp (node list)
  - _swmm_node<X>.out

8.5.5 Technical Details

- The tool identifies the closest storm drain node to the clicked point.
- Data is read from SWMM output files, which contain time series of inflow, flooding volume, and water depth at each node.
- The plot displays three series: inflow, flooding, and water depth.

8.6 Weirs Tools

The Weirs layer exposes three tools in the *Hydronia tools* context menu.

8.6.1 Dialog Window

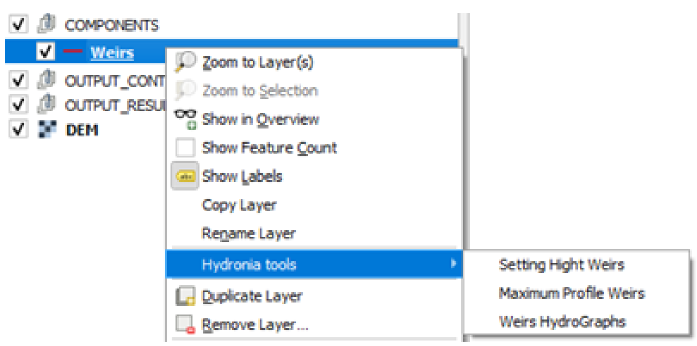


Figure 8.14 – Weirs Context Menu

8.6.2 Dialog Controls

The weir plotting tools use standard Matplotlib controls. See Section 7.1.2 for details on navigation controls, Figure Options, and Configure Subplots.

8.6.2.1 Setting Weirs Height

Sets a constant crest elevation or a constant height for the vertices of each weir.

- **Weirs crest elevation:** assigns a fixed elevation to all vertices of each weir.

- **Weirs height:** assigns a fixed height above the DEM; requires a DEM layer.

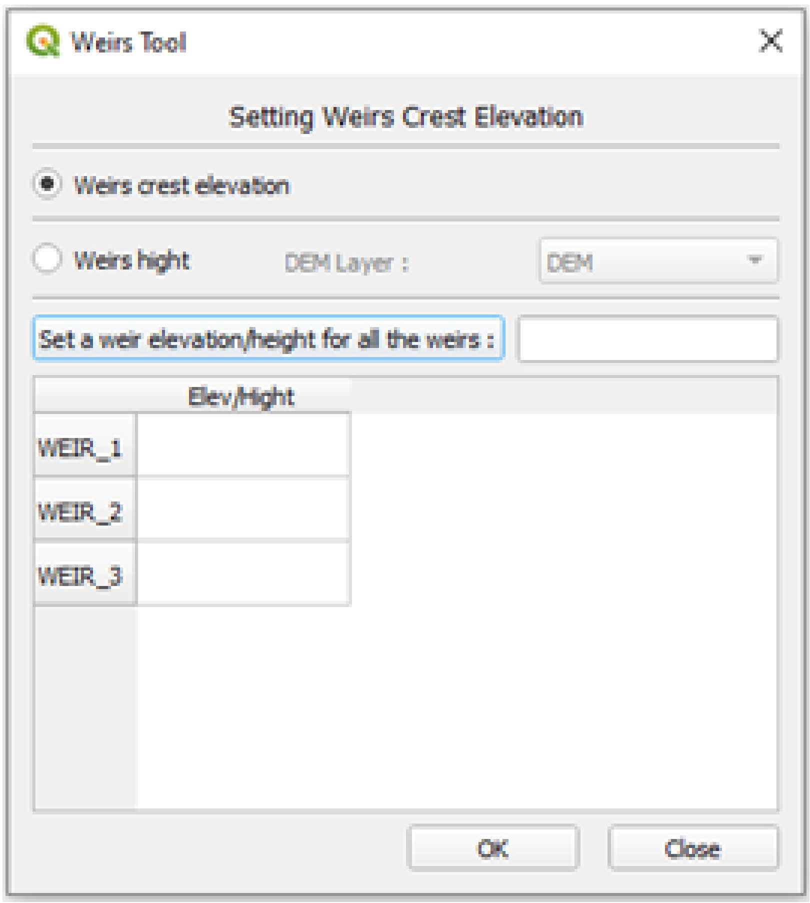


Figure 8.15 – Setting Weirs Height

8.6.2.2 Maximum Profile Weirs

Plots the weir profile and the maximum water surface elevation on both sides of the weir.

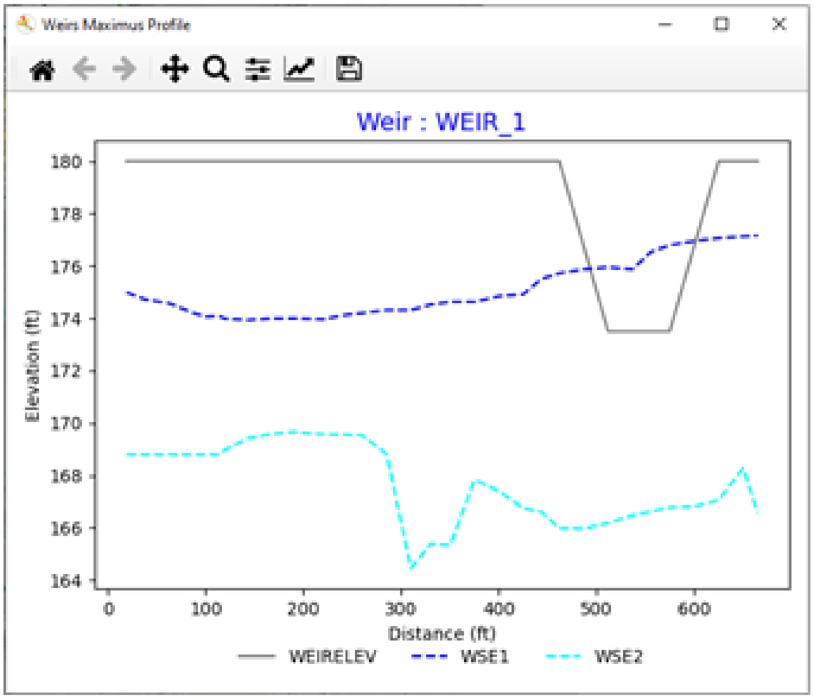


Figure 8.16 – Weirs Maximum Profile

8.6.2.3 Weirs HydroGraphs

Plots the discharge hydrograph over the selected weir.

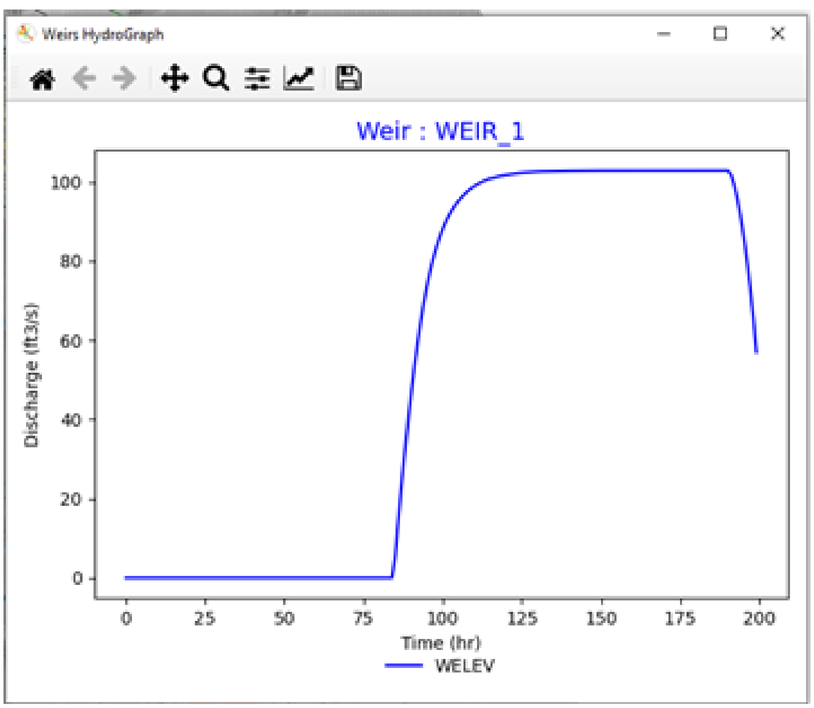


Figure 8.17 – Weirs HydroGraph

8.6.3 Workflow

1. Right-click the *Weirs* layer and choose *Hydronia tools*.
2. Select one of the three Weirs tools.
3. Left-click a weir polyline in the map canvas to run the selected tool.
4. Right-click to exit the tool.

8.6.4 Requirements

- Valid *Weirs* layer with features.
- Model output files:
  - <scene>.TWEIRS
  - <scene>.WEIRI or <scene>.WEIRE
- For the weirs height tool: DEM layer if using the height above terrain option.

8.6.5 Technical Details

- The tool identifies the closest weir to the clicked point using the feature's ID attribute.
- Weir data is read from .TWEIRS (time series) and .WEIRI/.WEIRE (profile data) files.
- The maximum profile displays the weir crest elevation along with maximum water surface elevations on the upstream and downstream sides.

8.7 Sources HydroGraphs

This tool plots the time series defined by a source or sink feature.

8.7.1 Dialog Window

The tool displays a Matplotlib plot with the time series for the selected source.

8.7.2 Dialog Controls

This tool uses standard Matplotlib plot controls. See Section 7.1.2 for details on navigation controls, Figure Options, and Configure Subplots.

8.7.3 Workflow

1. Right-click the *Sources* layer and choose *Hydronia tools Sources HydroGraphs*.
2. Left-click a source point in the map canvas.
3. The tool reads the time-series file referenced by the source feature and plots the series.
4. Right-click to exit the tool.

8.7.4 Requirements

- Valid *Sources* layer with features.
- Time-series file referenced by the SourceFile attribute (name resolved from the current scenario path).

8.7.5 Technical Details

- The tool identifies the closest source to the clicked point.
- The feature's SourceFile attribute contains the time-series filename, which is resolved relative to the current scenario folder.
- The plot displays the source (positive values) or sink (negative values) discharge over time.

8.8 ObservationPoints Tools

The ObservationPoints layer exposes three tools in the *Hydronia tools* context menu.

8.8.1 Dialog Window

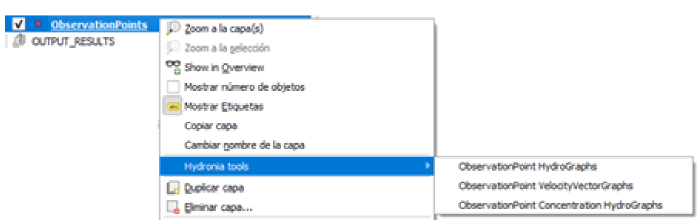


Figure 8.18 – ObservationPoints Context Menu

8.8.2 Dialog Controls

The observation points tools use standard Matplotlib controls. See Section 7.1.2 for details on navigation controls, Figure Options, and Configure Subplots.

8.8.2.1 ObservationPoint HydroGraphs

Plots velocity, depth, and water surface elevation (WSE) versus time for a selected observation point.

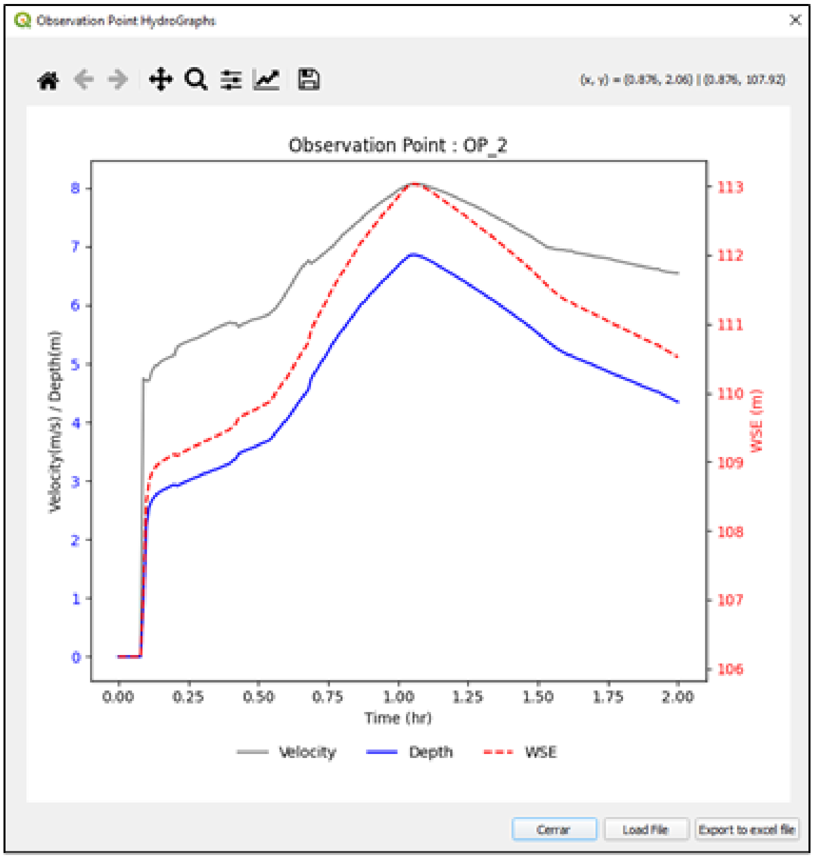


Figure 8.19 – ObservationPoint HydroGraphs

8.8.2.2 ObservationPoint VelocityVectorGraphs

Plots velocity vectors versus time for a selected observation point.

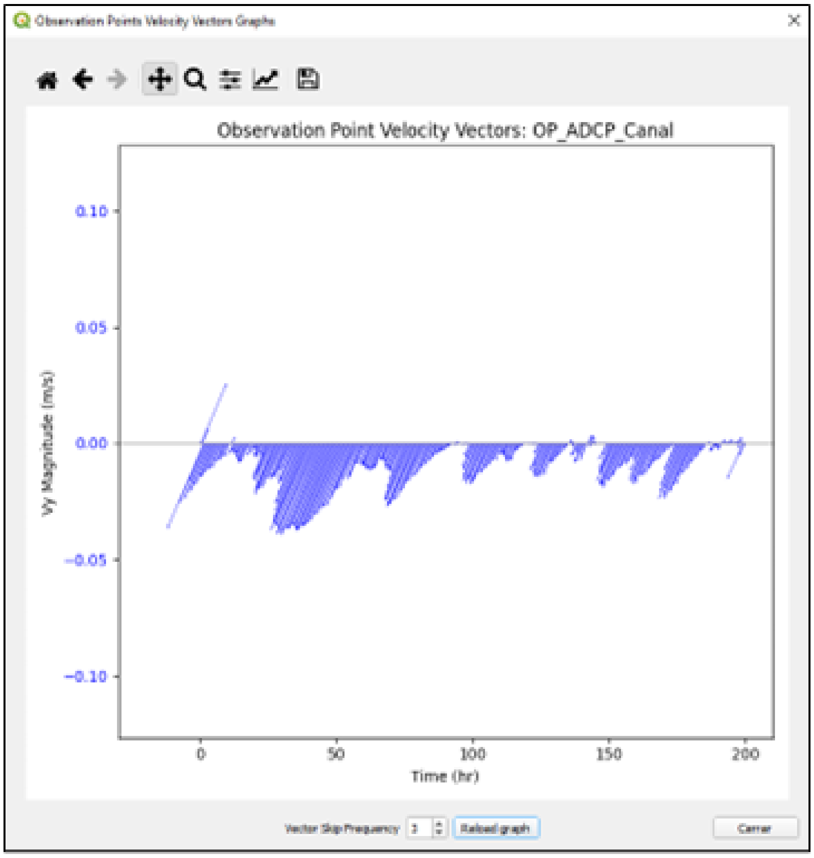


Figure 8.20 – ObservationPoint Velocity Vector Graphs

8.8.2.3 ObservationPoint Concentration HydroGraphs

Plots concentration time series for pollutants or suspended sediment fractions when those modules are enabled.

8.8.2.4 Optional Measured Data Overlay

The HydroGraphs and Concentration tools allow loading measured data from a two-column text file (time in hours, value). After loading, select the variable type (Velocity, Depth, or WSE) to overlay the measured series on the plot.

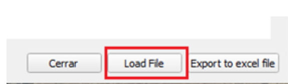


Figure 8.21 – Load File Button

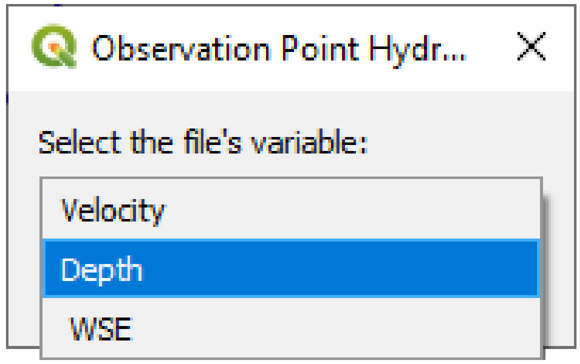


Figure 8.22 – Select Variable Dialog

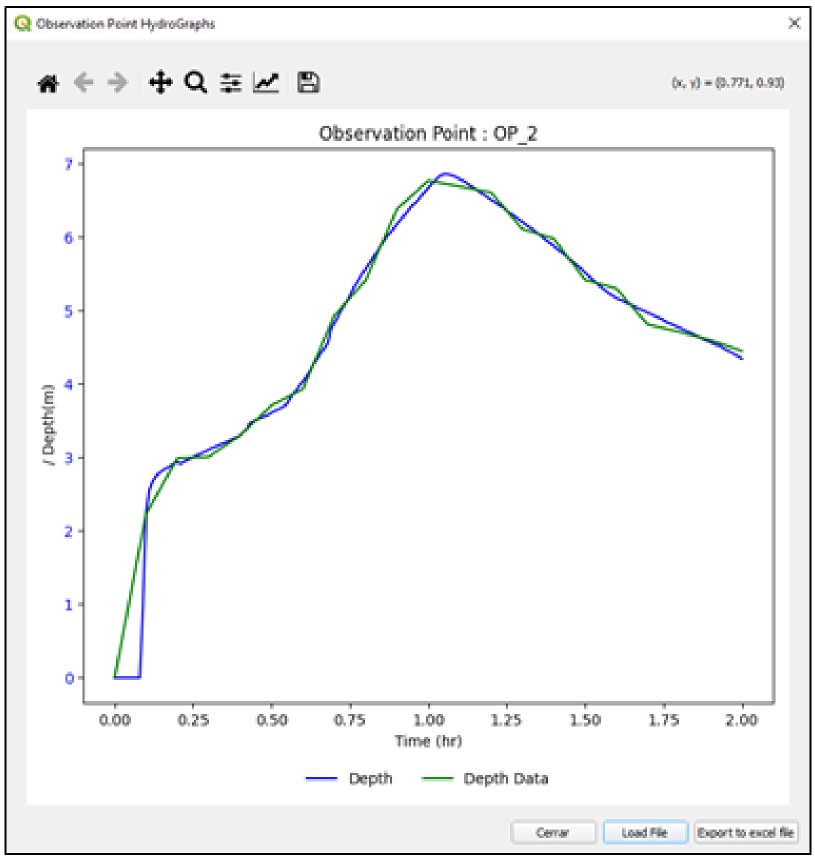


Figure 8.23 – Measured Data Overlay

8.8.3 Workflow

1. Right-click the *ObservationPoints* layer and choose *Hydronia tools*.
2. Select one of the three ObservationPoint tools.
3. Left-click an observation point in the map canvas.
4. The tool reads the corresponding output file and plots the requested series.
5. Right-click to exit the tool.

8.8.4 Requirements

- Valid *ObservationPoints* layer with features.
- Model output file: RESvsT_<ID>.OUTI
- The HydroGraphs and Concentration tools require metric units (not available with English units).

8.8.5 Technical Details

- The tool identifies the closest observation point to the clicked point using the feature's ID attribute.
- Data is read from RESvsT_<ID>.OUTI files, which contain time series of velocity, depth, WSE, and optionally concentrations.
- The velocity vector plot displays the u and v velocity components over time.
- Measured data files must have two columns: time (in hours) and value, with no header.

9 Appendix: QGIS Plugin Layer Attributes Reference

9.1 Layer Attributes Reference

The following tables detail the attribute fields for the default layers created by the New OilFlow2D Project tool and the New Template Layer Tool.

MeshDensityLine

Field Name	**Type**	**Description**		
Field Name	**Type**	**Description** CellSize	Real	Target mesh element size along this line.

MeshDensityPolygon

Field Name	**Type**	**Description**		
Field Name	**Type**	**Description** CellSize	Real	Target mesh element size within this area.

MeshBreakLine

Field Name	Type	Description		
Field Name	**Type**	**Description** CellSize	Real	Target mesh element size along this enforced breakline.

MultipleDemBoundaries

Field Name	Type	Description		
Field Name	**Type**	**Description** dem_layer	String	Identifier or path for the DEM layer associated with this boundary.

Domain_Outline

Field Name	Type	Description		
Field Name	**Type**	**Description** CellSize	Real	Specifies the default cell size for mesh generation within the domain.

BoundaryConditions

Field Name	Type	Description		
Field Name	**Type**	**Description** BCID	String	User-defined identifier for the boundary segment.
BCType	Integer	Type: 1=WSE vs Time (Free), 6=Discharge vs Time, 9=Rating Curve, 12=Normal Depth (Outflow), etc. See plugin UI for full list.		

Field Name	Type	Description
BCFileName	String	Path/Name of the associated data file (e.g., hydrograph, rating curve).

Manning N

Field Name	Type	Description		
Field Name	**Type**	**Description** ManningN	Real	Isotropic Manning's roughness coefficient for this area.

Manning_Nz

Field Name	Type	Description		
Field Name	**Type**	**Description** MANNNFILE	String	Path to the file defining anisotropic Manning's values (n, nx, ny, angle).

Initial_WSE

Field Name	Type	Description		
Field Name	**Type**	**Description** InitialWSE	Real	Initial water surface elevation value for this zone.

MaximumErosionDepth

Field Name	Type	Description		
Field Name	**Type**	**Description** MAXERODEPT	Real	Maximum allowable erosion depth in this zone.

Infiltration

Field Name	**Type**	**Description**		
Field Name	**Type**	**Description** INFILFILE	String	Path to the file defining infiltration parameters for this zone.

RainEvap

Field Name	**Type**	**Description**		
Field Name	**Type**	**Description** RAINEVFILE	String	Path to the file defining rainfal-l/evaporation rates for this zone.

Wind

Field Name	**Type**	**Description**		
Field Name	**Type**	**Description** CD	Real	Wind drag coefficient.
AIRDENSITY	Real	Air density.		
WINDFILE	String	Path to the file containing wind speed and direction time series.		

Bridges

Field Name	**Type**	**Description**		
Field Name	**Type**	**Description** BRIDGEID	String	Unique identifier for the bridge.
LELEM	Integer	Number of elements spanned by the bridge representation.		

Field Name	Type	Description
BRIDGEFILE	String	Path to the file defining bridge geometry/properties.
ZUPPER	Real	Elevation of the bridge deck soffit (underside).
DECK	Real	Thickness of the bridge deck.

Gates

Field Name	Type	Description		
Field Name	**Type**	**Description** GATEID	String	Unique identifier for the gate.
LELEM	Integer	Number of elements spanned by the gate representation.		
GATEFILE	String	Path to the file defining gate operation rules (time vs opening).		
CRESTELEV	Real	Elevation of the gate sill/crest.		
GHEIGHT	Real	Height of the gate opening when fully open.		
GATECD	Real	Discharge coefficient for the gate.		

Culverts

Field Name	Type	Description		
Field Name	**Type**	**Description** CULVERTID	String	Unique identifier for the culvert.

Field Name	Type	Description
CULVERTYPE	Integer	Type defining culvert shape and hydraulic calculation method (e.g., 1=Rectangular, 2=Circular). See plugin UI for full list.
CULVERTFIL	String	Path to rating curve file (if CULVERTYPE=0).
Nb	Integer	Number of identical barrels.
Ke	Real	Entrance loss coefficient.
nc	Real	Manningś n for the culvert barrel.
KP	Real	Coefficient used in FHWA equation.
M	Real	Coefficient used in FHWA equation.
Cp	Real	Coefficient used in FHWA equation.
Y	Real	Coefficient used in FHWA equation.
m1	Real	Coefficient used in FHWA equation (0.7 or -0.5).
Hb	Real	Height of box culvert.
Base	Real	Base width of box culvert.
Dc	Real	Diameter of circular culvert.
INVERT_Z1	Real	Invert elevation at the upstream end.

Field Name	Type	Description
INVERT_Z2	Real	Invert elevation at the downstream end.

Weirs

Field Name	Type	Description		
Field Name	**Type**	**Description** WEIR_ID	String	Unique identifier for the weir.
WEIRCD	Real	Discharge coefficient for the weir.		
WCRESTELEV	String	Path to the file defining weir crest elevation profile or a constant value.		
LELEM	Real	Number of elements spanned by the weir representation.		

DamBreach

Field Name	Type	Description		
Field Name	**Type**	**Description** DAMID	String	Unique identifier for the dam/breach segment.
DAMFILE	String	Path to file defining prescribed breach parameters (if iFailure=1).		
L_CENTER	Real	Location of breach center along the dam line.		
Z_CREST	Real	Initial elevation of the dam crest.		

Field Name	Type	Description
BREACHANG	Real	Angle of the breach side slopes.
BREACHCD	Real	Discharge coefficient for the breach flow.
LELEM	Real	Number of elements spanned by the dam representation.
iFailure	Integer	Failure mode: 1=Prescribed, 2=Overtopping, 3=Piping.
D50	Real	Median grain size of dam material (for erosion modes).
Tau_c	Real	Critical shear stress (for erosion modes).
K_sm	Real	Coefficient (for erosion modes).
T_initial	Real	Time of breach initiation (for erosion modes).
GS	Real	Specific gravity of dam material (for erosion modes).
Porosity	Real	Porosity of dam material (for erosion modes).
kd	Real	Erodibility coefficient (for erosion modes).
Zb0	Real	Initial breach bottom elevation (for erosion modes).
C	Real	Coefficient (for erosion modes).

Field Name	Type	Description
cWidth	Real	Crest width of the dam.
uSlope	Real	Upstream slope of the dam.
dSlope	Real	Downstream slope of the dam.

Sources_Sinks

Field Name	Type	Description		
Field Name	**Type**	**Description** SOURCEID	String	Unique identifier for the source/sink.
SOURCETYPE	Integer	Type: 1=Discharge vs Time, 2=Rating Curve (Depth vs Q).		
FILENAME	String	Path to the associated data file (hydrograph or rating curve).		

InitialConcentrations

Field Name	Type	Description		
Field Name	**Type**	**Description** CONCENTFIL	String	Path to file defining initial concentration values for this zone.

Internal_Rating_Table

Field Name	Type	Description		
Field Name	**Type**	**Description** IRT_ID	String	Unique identifier for the internal rating table location.

Field Name	Type	Description
IRT_BCType	Integer	Boundary condition type (typically 19 for Stage vs Discharge).
IRTFileName	String	Path to the file containing the stage-discharge data.
CellSize	Real	Cell size parameter related to the internal boundary representation.

Piers

Field Name	Type	Description		
Field Name	**Type**	**Description** PIERID	String	Unique identifier for the pier.
ICOMP	Integer	Calculation method: 1=Complex piers, 2=HEC-18, 3=Coarse Bed, 4=Cohesive Materials.		
ALFA	Real	Angle of attack (flow relative to pier alignment).		
ISHAPE	Integer	Pier shape: 1=Square Nose, 2=Round Nose, 3=Cylindrical, 4=Sharp Nose, 5=Group of Cylinders.		
PIER_L	Real	Length of the pier parallel to flow.		
PIER_A	Real	Width of the pier perpendicular to flow.		

Field Name	Type	Description
IBEDCO	Integer	Bed condition: 1=Clear-Water Scour, 2=Plane bed and Antidune flow, 3=Small Dunes, 4=Medium Dunes, 5=Large Dunes.
D50	Real	Median grain size of bed material.
D84	Real	Grain size for which 84
SS	Real	Specific gravity of sediment.
SW	Real	Specific weight of water.
THETA	Real	Critical Shields parameter.
VC	Real	Critical velocity for initiation of motion.
CD	Real	Drag coefficient for the pier.

Abutments

Field Name	Type	Description		
Field Name	**Type**	**Description** ABUTID	String	Unique identifier for the abutment.
IABUTMENT	Integer	Abutment type: 1=Spill-through, 2=Vertical wall with wind walls.		
IXSECTION	Integer	Cross-section type: 1=Bridge Section, 2=Upstream Section.		

Field Name	Type	Description
D50	Real	Median grain size of bed material.
SW	Real	Specific weight of water.

OilSpills

Field Name	Type	Description		
Field Name	**Type**	**Description** IDSPILLSTE	String	Unique identifier for the spill source/site.
WDENSITY	Real	Density of the water.		
COORD_X	Real	X-coordinate of the spill location.		
COORD_Y	Real	Y-coordinate of the spill location.		
COORD_Z	Real	Z-coordinate (elevation) of the spill location.		
NPARCELS	Integer	Number of parcels released per time step for this spill.		
OILDENSITY	Real	Density of the spilled oil.		
OILVISCOS	Real	Viscosity of the spilled oil.		
INITIALST	Real	Initial time (hours) when the spill starts.		
DISP_L	Real	Longitudinal dispersion coefficient.		
DISP_T	Real	Transverse dispersion coefficient.		
DISP_V	Real	Vertical dispersion coefficient.		

Field Name	Type	Description
HYDROGFILE	String	Path to the file defining the spill hydrograph (volume vs time).
s_bottom	Boolean	Flag to activate interaction with the bottom.
s_shore	Boolean	Flag to activate interaction with the shoreline.
s_dispapp	Boolean	Flag related to dispersant application effects.
s_vegtrapp	Boolean	Flag to activate trapping by vegetation.

Shorelines

Field Name	Type	Description		
Field Name	**Type**	**Description** IDTYPE	Integer	Type of shoreline (e.g., 1=Rocky cliffs, 2=Sand beaches, etc. See plugin UI for full list).

InitialBedFractions

Field Name	Type	Description		
Field Name	**Type**	**Description** CONCENTFIL	String	Path to file defining initial bed sediment fraction values for this zone.

SpillBooms

Field Name	Type	Description		
Field Name	**Type**	**Description** BOOMID	String	Unique identifier for the spill boom.
BOOMTYPE	Integer	Type of boom (e.g., 1=Curtain, 2=Fence, 3=Sorbent, 4=Bubble). See plugin UI for full list.		
SKIRTHEIGH	Real	Skirt height of the boom.		
POILLOSS	Real	Percentage of oil loss through the boom.		
VU	Real	Boom velocity component in U direction.		
VV	Real	Boom velocity component in V direction.		

SpillPaths

Field Name	Type	Description		
Field Name	**Type**	**Description** IDSPILLSTE	String	Unique identifier for the spill source/site associated with this path.
WDENSITY	Real	Density of the water.		
NPARCELS	Integer	Number of parcels released per time step along this path.		
OILDENSITY	Real	Density of the spilled oil.		
OILVISCOS	Real	Viscosity of the spilled oil.		

Field Name	Type	Description
INITIALST	Real	Initial time (hours) when the spill starts.
DISP_L	Real	Longitudinal dispersion coefficient.
DISP_T	Real	Transverse dispersion coefficient.
DISP_V	Real	Vertical dispersion coefficient.
HYDROGFILE	String	Path to the file defining the spill hydrograph (volume vs time).

OilPipelines

Field Name	Type	Description		
Field Name	**Type**	**Description** OPLID	String	Unique identifier for the pipeline segment.
OPLFILE	String	Path to file containing detailed pipeline properties or data.		

OilPipeLineBoundCond

Field Name	Type	Description		
Field Name	**Type**	**Description** OPLBCID	String	Unique identifier for the pipeline boundary condition point.
OPLBCTYPE	Integer	Type of boundary condition (e.g., leak rate, pressure).		

Field Name	Type	Description
OPLBCFILE	String	Path to file containing time series data for the boundary condition.

OilRetDepth

Field Name	Type	Description		
Field Name	**Type**	**Description** RETDEPFIL	String	Path to file defining oil retention depth parameters for this zone.

VegTrapp

Field Name	Type	Description		
Field Name	**Type**	**Description** VEGTRAPFIL	String	Path to file defining vegetation trapping parameters for this zone.

CrossSections

Field Name	Type	Description		
Field Name	**Type**	**Description** XSECID	String	Unique identifier for the cross-section line.
ND_CS	Integer	Number of discretization points along the cross-section for output.		

Profiles

Field Name	**Type**	**Description**		
Field Name	**Type**	**Description** PROFILEID	String	Unique identifier for the profile line.
ND_PR	Integer	Number of discretization points along the profile line for output.		

ObservationPoints

Field Name	**Type**	**Description**		
Field Name	**Type**	**Description** ObsID	String	Unique identifier for the observation point.