NORWEGIAN UNIVERSITY OF SCIENCE AND TECHNOLOGY
Faculty of Engineering
Department of Ocean Operations and Civil Engineering

Digital Twin Platform

User Manual

Lam Van Nguyen
Razak Seidu
Russell Burch Primeau
Joseph Kwarko-Kyei

Smart Water Group
NTNU Ålesund

01/2026

1. Introduction

1.1. Overview of the Digital Twin platform

The Digital Twin Platform is a web-based application for automating the simulation and visualization of hydrodynamics and water quality transport in drinking water reservoirs, such as lakes and rivers. The system also enables quick setup and running of different scenarios to investigate possible responses of a surface water body under changing contaminant levels and weather conditions.

Building a model from traditional hydrodynamics and water quality modelling systems typically requires manual preprocessing of input data, running calculations, and finally postprocessing model outputs into charts for interpretation. The main purpose of the platform is to automate all these manual steps in a way that enables direct integration of the model with remote data acquisition from field sensors. This allows changes in essential parameters of water quality in raw water sources to be continuously simulated and visualized under near-real-time conditions.

The simulation and visualization system in this platform relies on the Delft3D Flexible Mesh Suite (Delft3D FM), which can simulate storm surges, hurricanes, tsunamis, detailed flows and water levels, waves, sediment transport and morphology, water quality and ecology, and can handle the interactions between these processes. Delft3D FM itself is composed of several modules; however, to simulate water quality in drinking water reservoirs, the following modules are used in the platform:

Information about the Delft3D FM software can be found at https://oss.deltares.nl/web/delft3dfm/home.

1.2. Access the Digital Twin platform

Open URL: https://digiwater.ddns.net/visualization with a web browser. The user can use any web browser, but Edge is recommended. Provide username and password when asked (**Figure 1.1**).

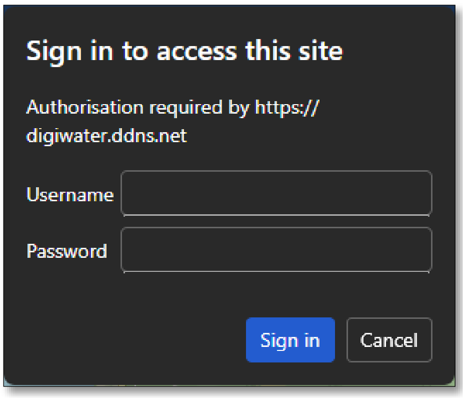


Figure 1.1. Access the platform

The user needs to wait for the platform to set up the database and load components successfully until an overview map is displayed (**Figure 1.2**).

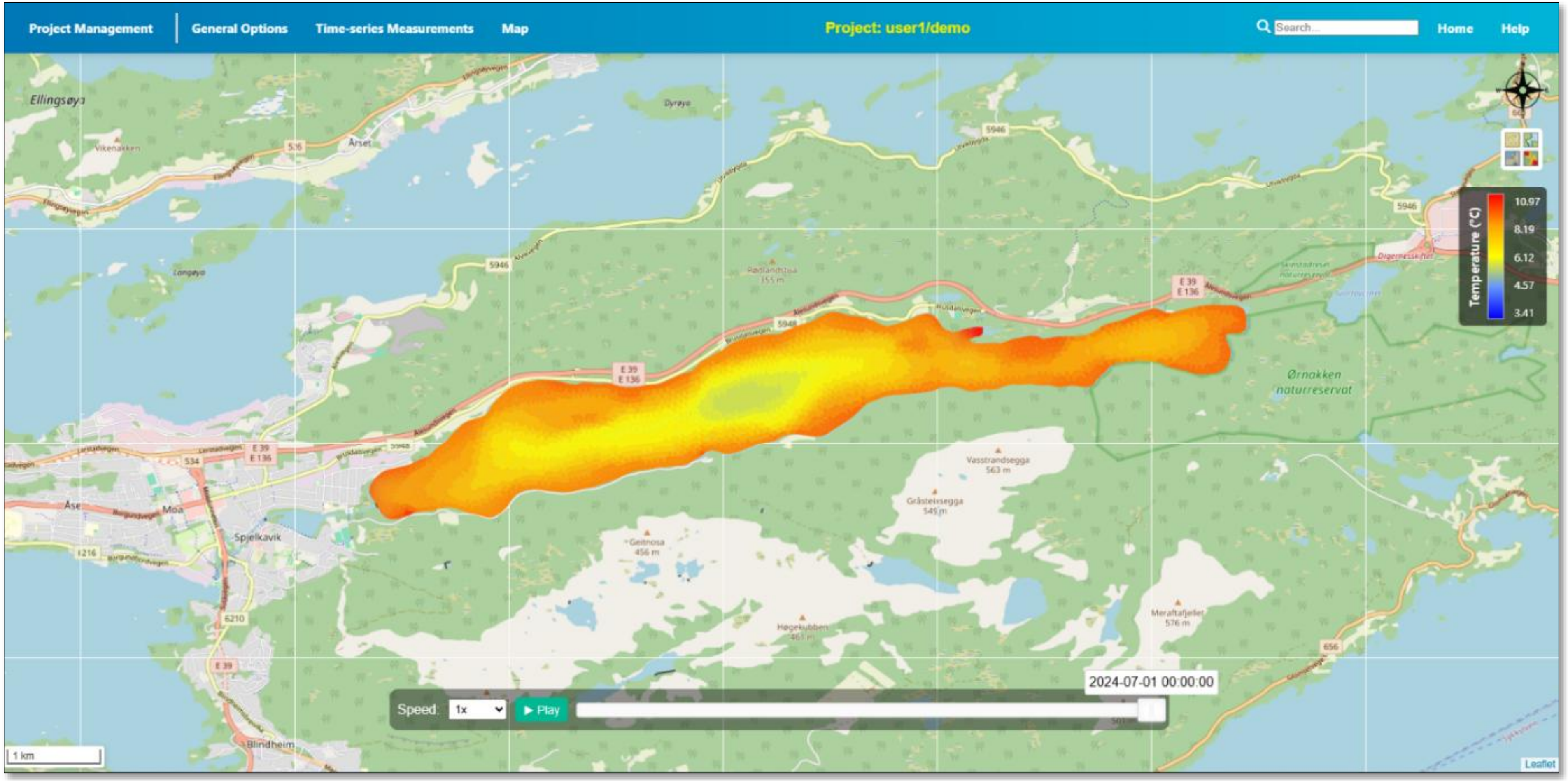


Figure 1.2. Main window

1.3. Main components

1.3.1. Project Management

The "***Project Management***" option allows the user to create and run a hydrodynamic (HYD) simulation, create and run a water quality (WAQ) simulation, and select a scenario to visualize on a map. Components of the main menu are shown in **Figure 1.3**.

Figure 1.3. Project management

1.3.2. Main menu

After the user selects and opens a specific project, the main menu provides functionality to interact with different elements of the project. For example, project information, load objects, contact information, and so on.

The number of main items on the menu will depend on the availability of each project. The main components of the main menu are shown in **Figure 1.4**.

Figure 1.4. Main menu

- *General Options:* allow the user to see overviews of the project, such as a summary of the project, information on POI during the simulation, select and plot profiles, customize and draw thermocline charts (**Figure 1.5**).

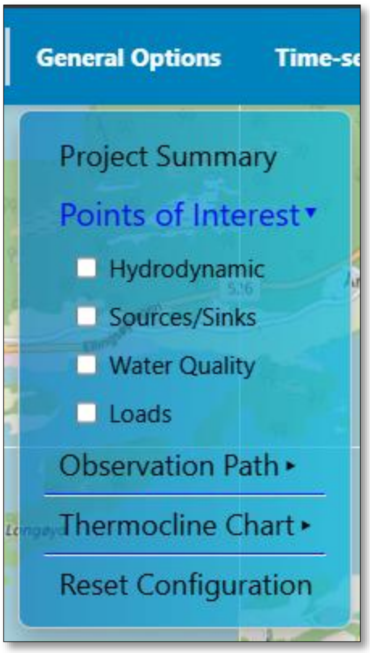


Figure 1.5. General options

- *Time-series Measurements:* allow the user to see time-based measurements for both the HYD simulation (i.e., water level, meteorology, water balance, etc.) and the WAQ simulation. General items of this option are shown in **Figure 1.6**.

Figure 1.6. Time-series measurements option

- *Map:* this option provides the ability to visualize many types of maps, including both static and dynamic maps, single and multiple layers (**Figure 1.7**).

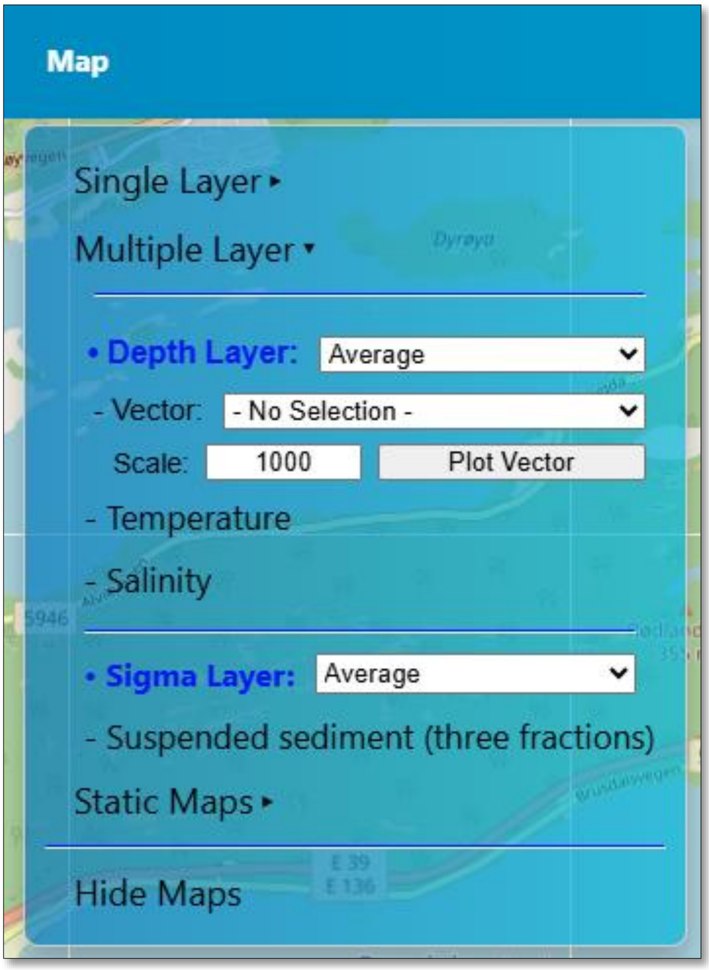


Figure 1.7. Map option

- *Project Title:* this area (in this case is "**Project: user1/demo**") shows detailed information of the used project, including the username in the first part (*user1*) and the opened project in the second part (*demo*).

- *Search:* the user can use this option to quickly find a specific place around the world (**Figure 1.8**).

Figure 1.8. Search for a place

➤ *Home:* the user can go back to the main page using this option.

➤ *Help:* this option shows more detailed information to contact our people (**Figure 1.9**).

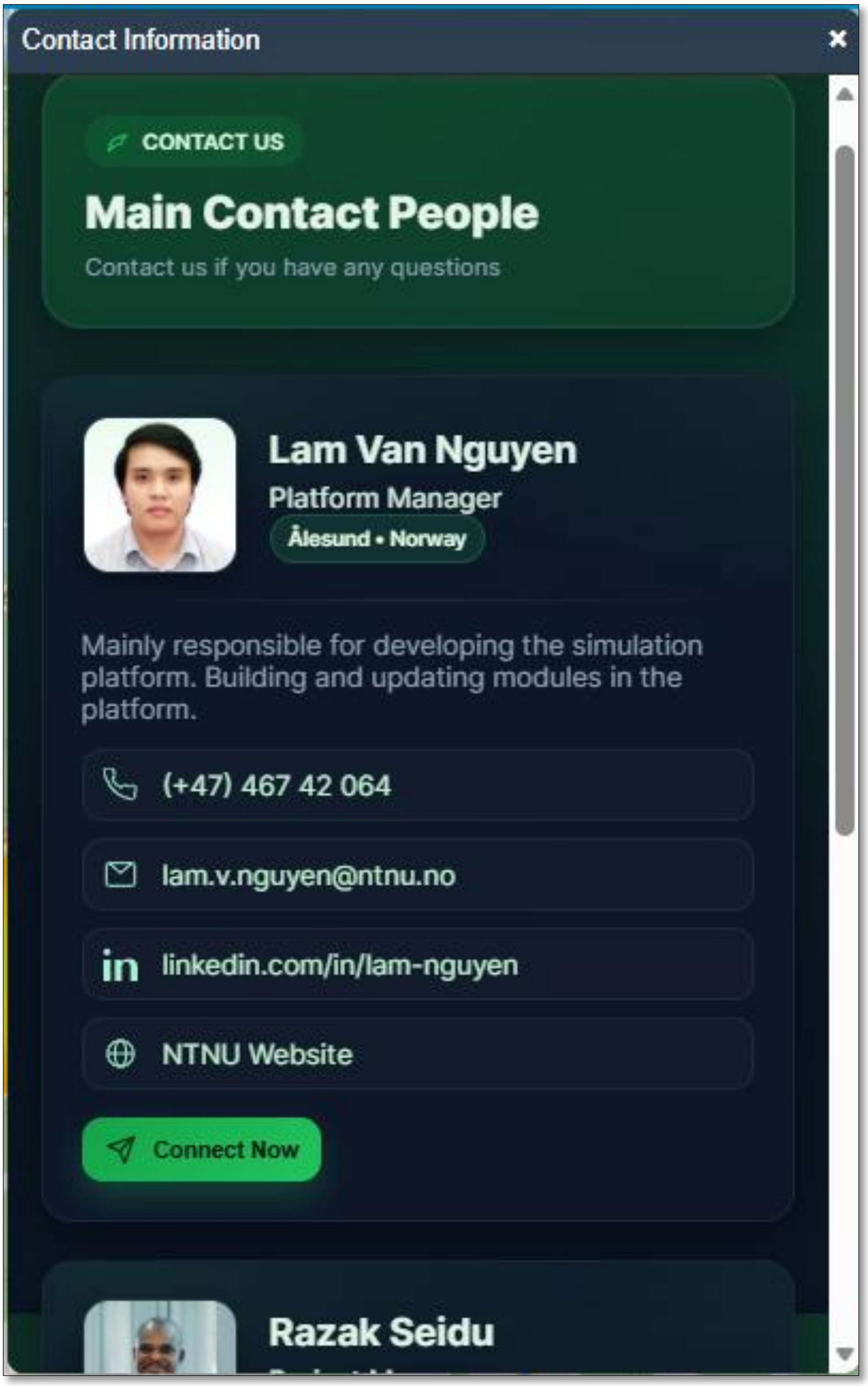


Figure 1.9. Contact information

1.3.3. Workspace

Workspace is a place where all important information and visualizations are displayed. The main interface of the workspace is a map-based interface where spatial and non-spatial information are visualized (**Figure 1.10**).

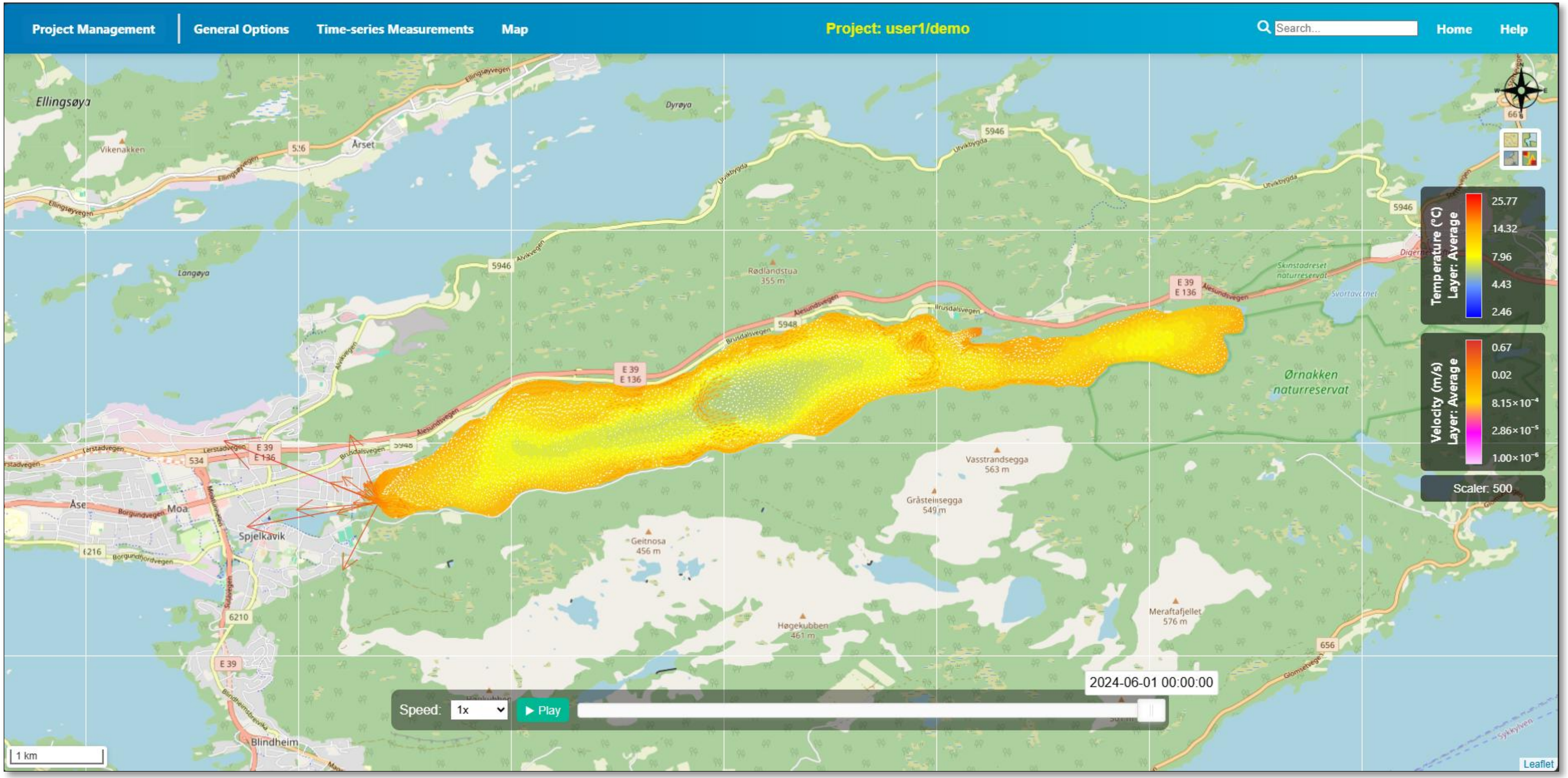


Figure 1.10. Spatial map

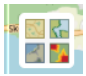
The user can change map backgrounds by hovering the mouse over this symbol  in the top-right corner. Some mapping templates can be shown in **Figure 1.11**.

Figure 1.11. Base map

2. Building and running a hydrodynamic (HYD) simulation

2.1. Creating a flexible mesh grid

The Delft3D FM software provides a tool allowing the user to create a flexible mesh grid. This is a core component that enables the user to construct computational domains with a highly adaptable mesh structure, suitable for simulating hydrodynamics, sediment transport, and substance transport in complex hydrological and coastal systems. Unlike traditional structured or curvilinear grids, the flexible mesh allows the combination of different element types within a single computational domain, including triangles, quadrilaterals, and general polygons.

Thanks to this flexibility, the user can accurately represent complex geometries such as rivers, channels, estuaries, coastal zones, lagoons, or urban floodplains, while optimizing the total number of grid cells and computational time. The mesh can be locally refined in areas requiring higher resolution (e.g., regions with complex flow patterns, estuaries, or hydraulic structures) and coarsened in less dynamic areas, thereby achieving a balance between computational accuracy and efficiency.

In Delft3D FM, the flexible mesh generation process is carried out through several main steps:

- Define the computational domain boundaries.
- Specify characteristic lines (such as shorelines, river centerlines, and channels).
- Set the desired mesh resolution.
- Generate the mesh automatically or semi-automatically.

The software provides an interactive mesh-editing tool (RGFGrid) that allows the user to inspect and refine mesh quality, ensuring that grid cells have appropriate shapes and avoiding overly distorted elements or unfavorable aspect ratios (**Figure 2.1**).

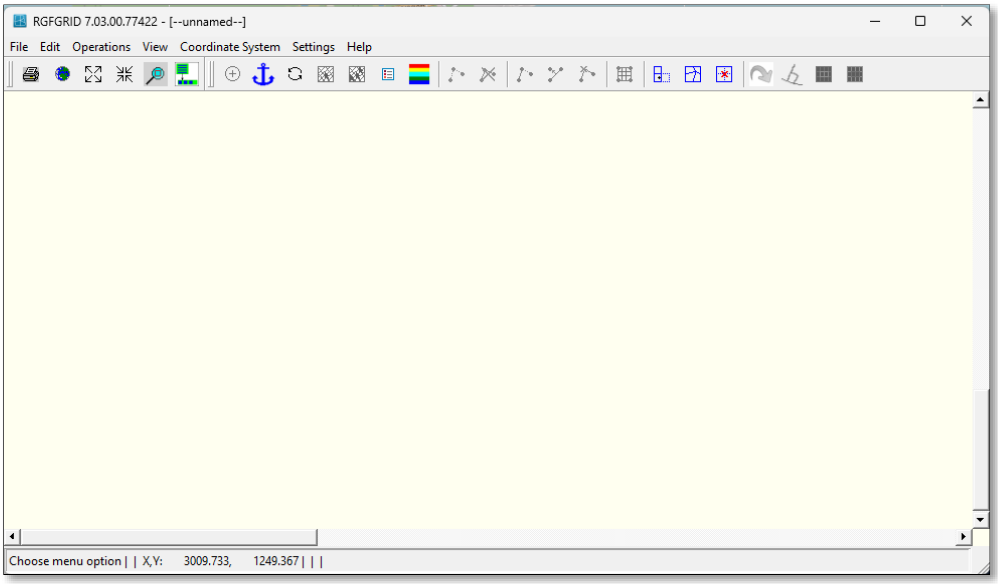


Figure 2.1. Flexible mesh generation wizard

Detailed steps for making a flexible mesh can be found at: https://content.oss.deltares.nl/delft3d4/RGFGRID_User_Manual.pdf.

NOTE: For this manual, a flexible mesh for the Brusdalsvatnet area is provided.

2.2. Setting up a HYD scenario

To open a new HYD project, click on the "***Project Management***" option and then select "***HYD Scenario***". This window also allows the user to create a new scenario, open an existing scenario to modify, clone an existing scenario, and delete a scenario (**Figure 2.2**).

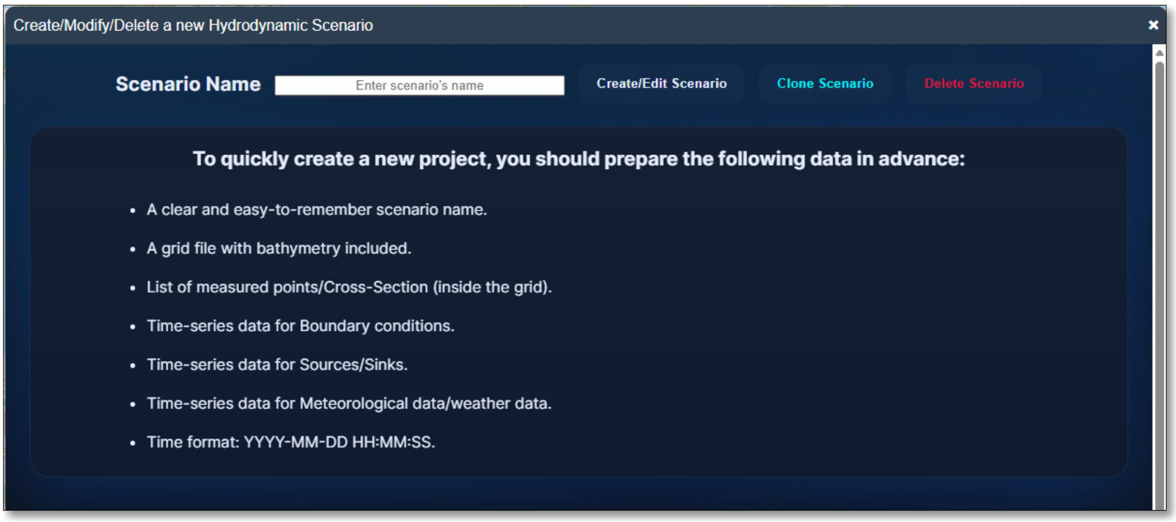


Figure 2.2. Create a new HYD scenario

- *To create a new HYD scenario*: provide the name of the scenario, then hit the "***Create/Edit Scenario***" button.
- *Open an existing scenario*: type the name of the scenario in the text, select the scenario from the list, and then hit the "***Create/Edit Scenario***" button (**Figure 2.3**).

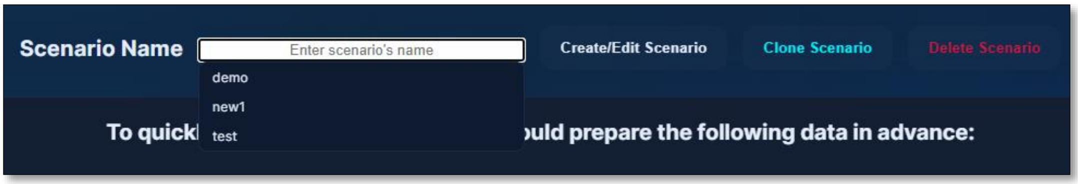


Figure 2.3. Open an existing HYD scenario

After finishing the above steps, a new window will appear where the user can initiate new parameters for a new scenario or modify parameters of the existing scenario. This window has three main tabs: ***Model Settings***, ***Hydro and Meteo Parameters***, and ***Output Parameters***.

2.2.1. Model Settings

The "***Model Settings***" tab contains several sub-tabs that allow the user to initiate basic configurations for the scenario, which are: ***General***, ***Time Frame***, ***Observations***, ***Boundary Conditions***, and ***Other Parameters***.

- *General*: This sub-tab allows the user to define the approximate location of the modelling area, upload a pre-defined flexible grid, and initiate some sigma layers (**Figure 2.4**). To understand more about the sigma layer, please refer to this manual: https://content.oss.deltares.nl/delft3d/D-Flow_FM_User_Manual.pdf.

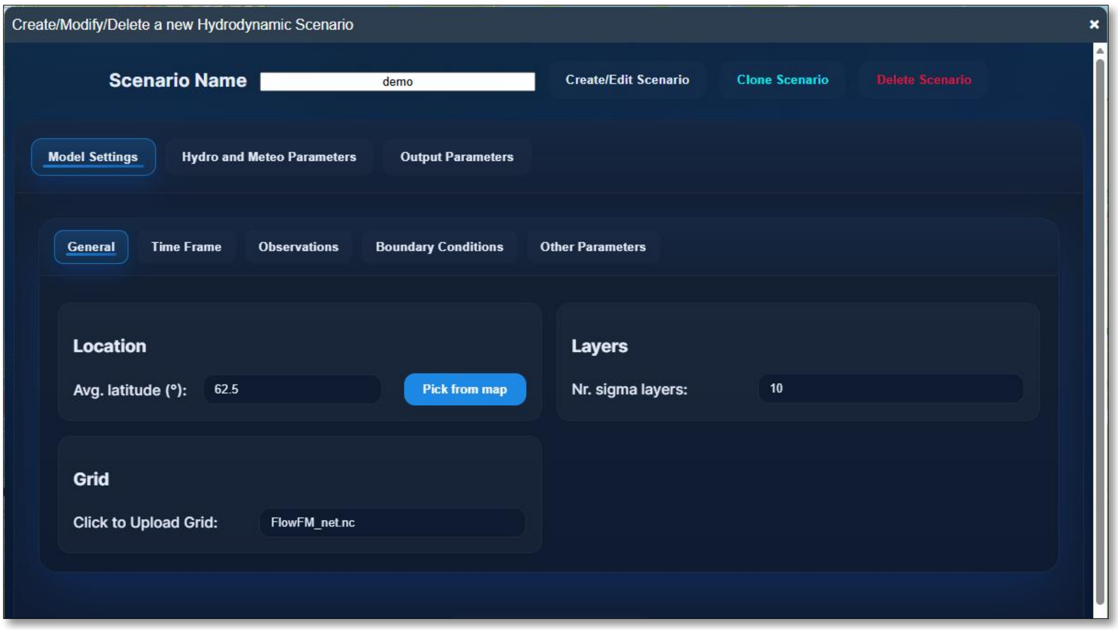


Figure 2.4. Set up general options for the HYD scenario

❖ *Time Frame*: The user can select the simulation period and setup time interval during the simulation using this sub-tab (**Figure 2.5**).

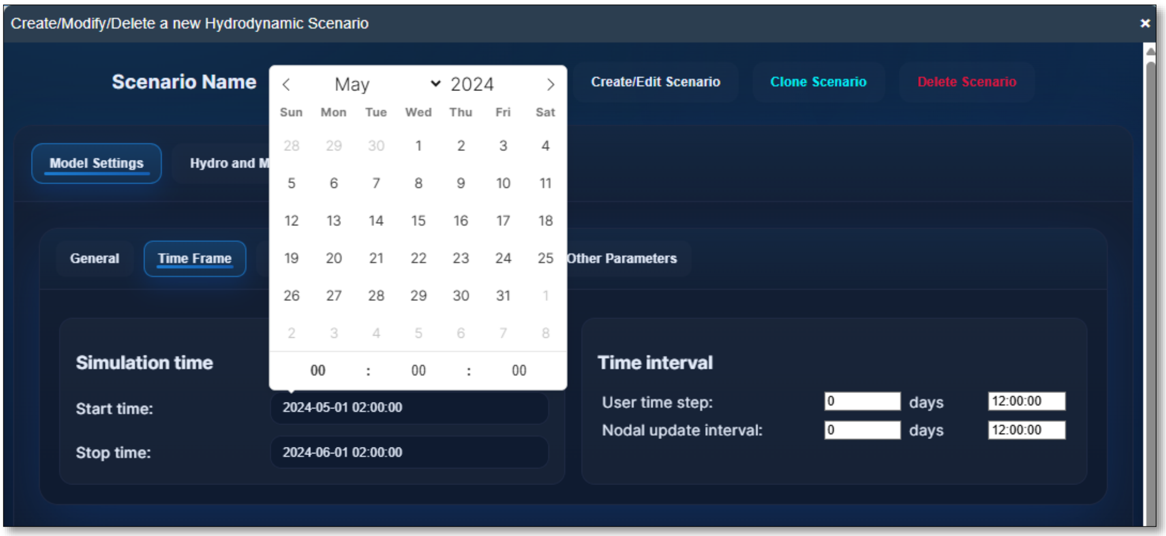


Figure 2.5. Set up the simulation period

❖ *Observations*: This sub-tab allows the user to define observation points and cross-sections (if needed) to monitor simulated information during the simulation period (**Figure 2.6**). The user can either define observation points by selecting them from the map directly, uploading them from a CSV file, or typing their attributes (i.e., name, latitude, and longitude) on this sub-tab directly.

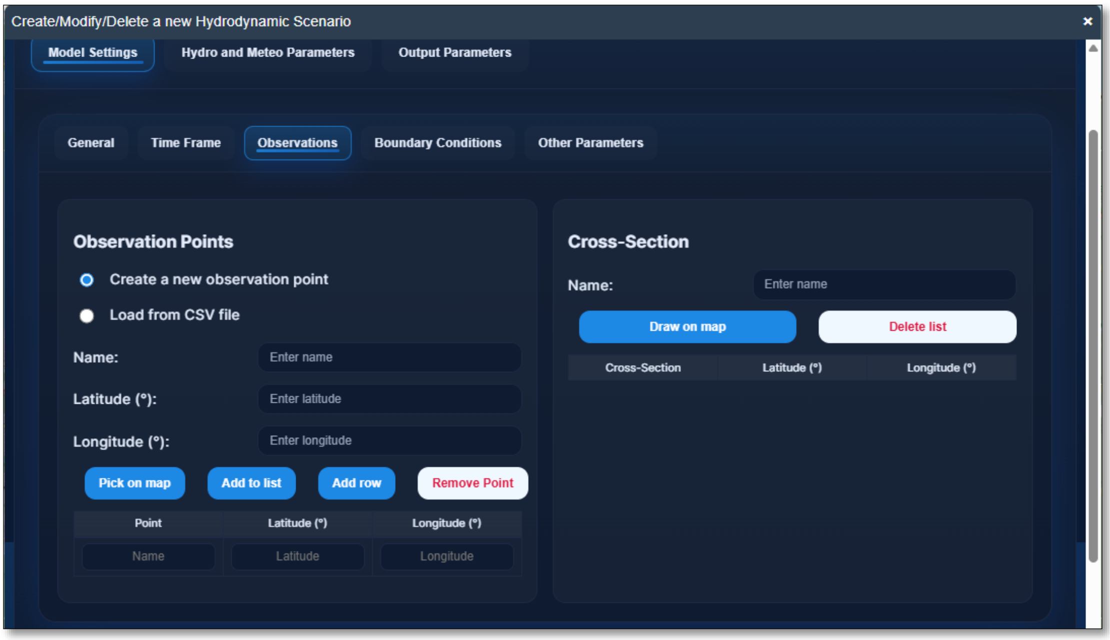


Figure 2.6. Set up observations

➢ *Select observation points from the map*: The user can follow the steps below to create an observation point:

- *Step 1*: Active radio "***Create a new observation point***" button.
- *Step 2*: Click the "***Pick on map***" button.
- *Step 3*: This window is temporarily hidden, and the map appears. Select one point on the map (inside the grid area) by using the left mouse button.
- *Step 4*: This window will display again, then hit the "***Add to list***" button to add the selected point to the list (**Figure 2.7**).

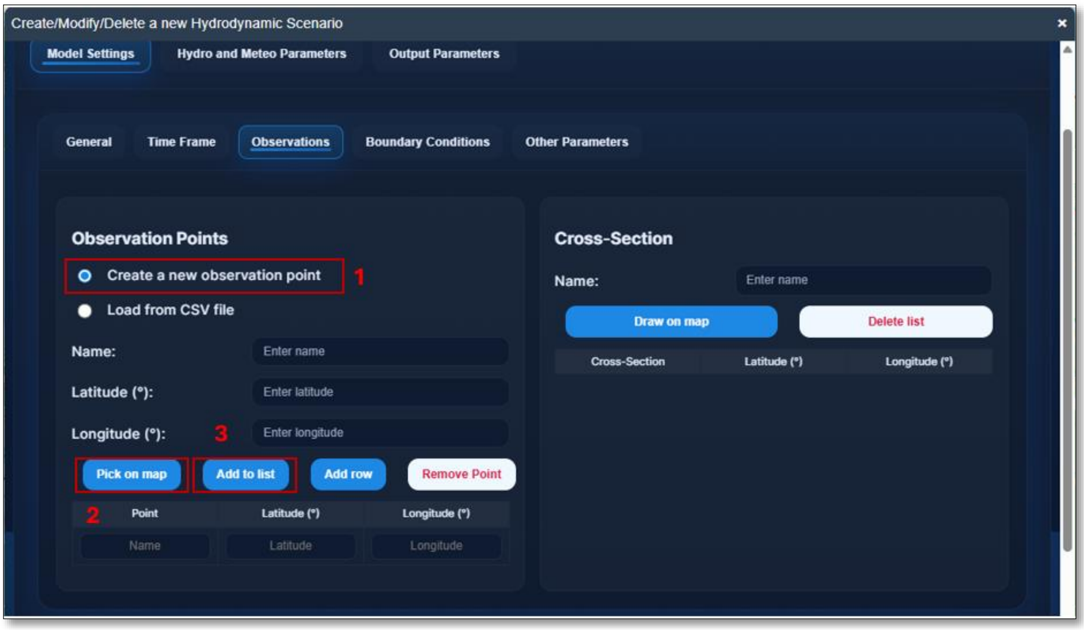


Figure 2.7. Add an observation point to the list

Repeat steps 2-4 to add other points to the list (**Figure 2.8**). To remove an observation from the list, copy its name and paste it into the field "***Name***" and then click the "***Remove Point***" button.

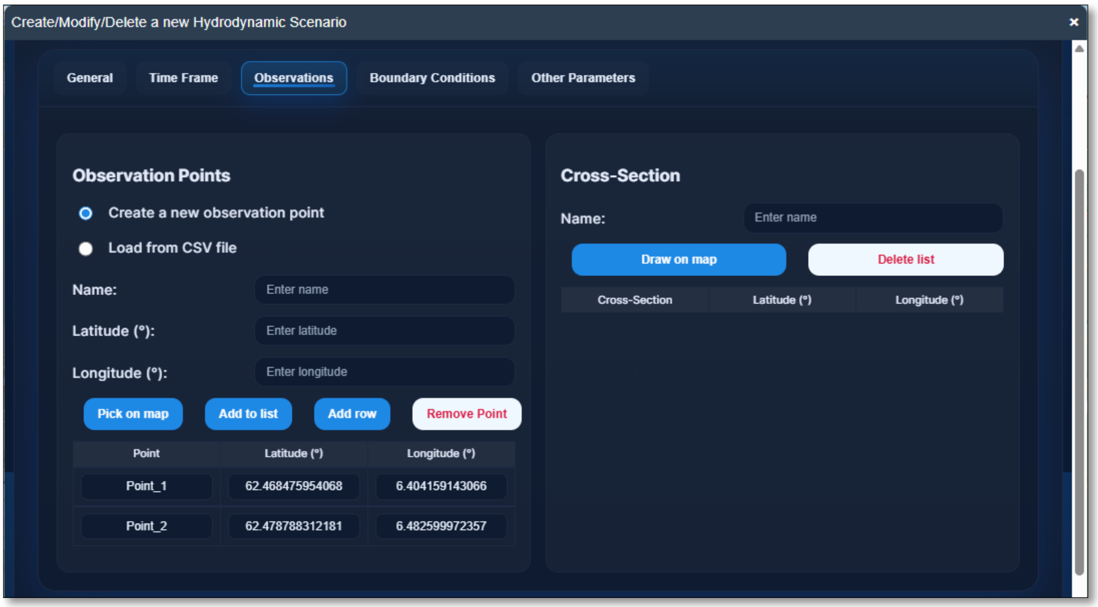


Figure 2.8. Select observations from the map

➢ *Upload observation points from the CSV file:* This is a convenient way to add a pre-defined list of observations to the list. The user can follow the steps below to upload a CSV file:

- *Step 1:* Active radio "***Load from CSV file***" button to display an area to upload a file (**Figure 2.9**). If the user has already added observation points, these points will be kept in the list. The user can add other points from a CSV file independently.

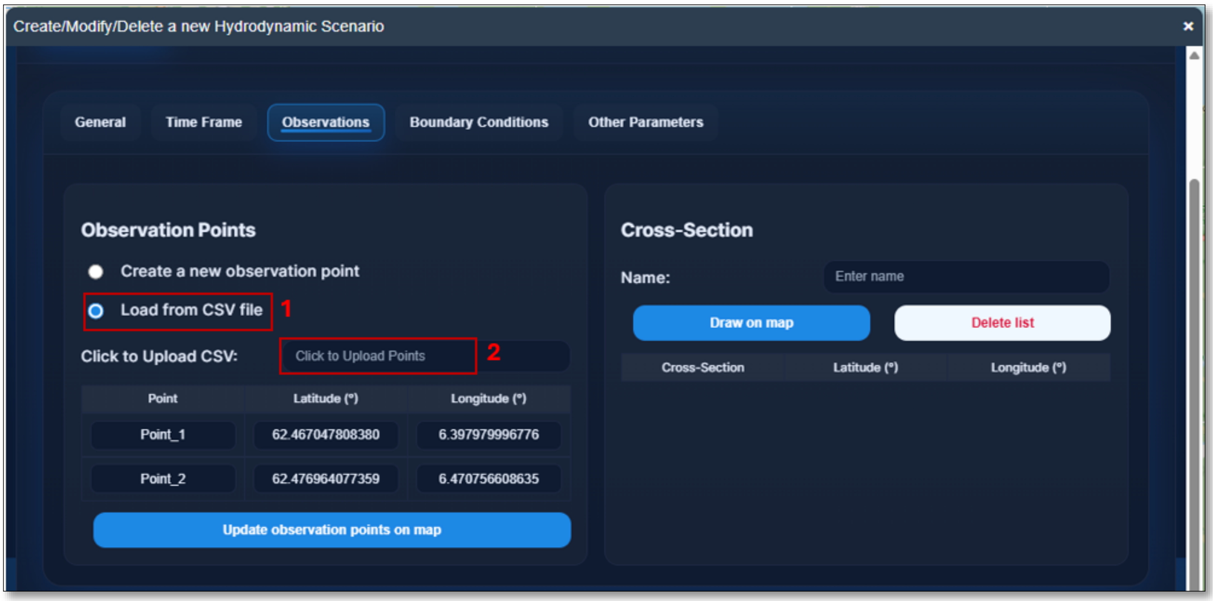


Figure 2.9. Upload observations from a CSV file

- *Step 2:* Click in the "***Click to Upload CSV***" text and then select a CSV file from the computer. A list of all observation points is listed in **Figure 2.10**.

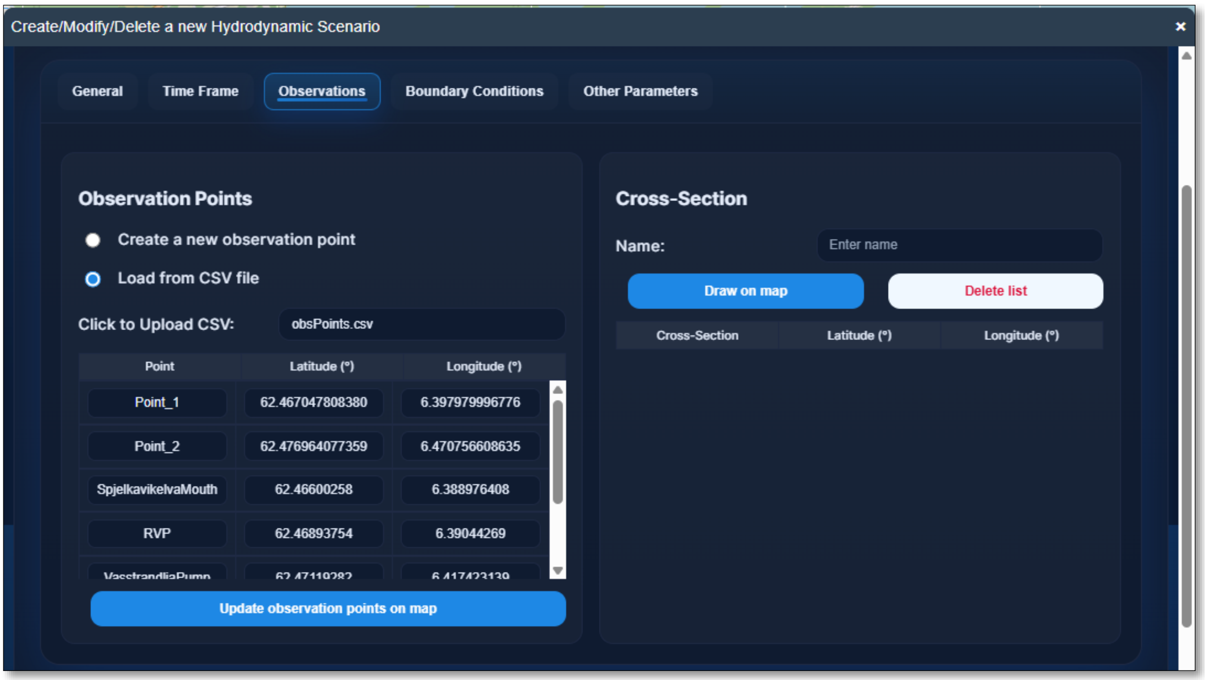


Figure 2.10. List of selected observation points

The structure of the pre-defined list is shown in **Figure 2.11**.

Name	Lat	Long
SpjelkavikelvaMouth	62.46600258	6.388976408
RVP	62.46893754	6.39044269
VasstrandliaPump	62.47119282	6.417423139
Profiler	62.47452771	6.461336385
Norebotnen	62.48383188	6.565308454

Figure 2.11. List of observation points (in CSV format)

The "***Update observation points on map***" button allows the user to see the distribution of all selected observation points on the map (**Figure 2.12**).

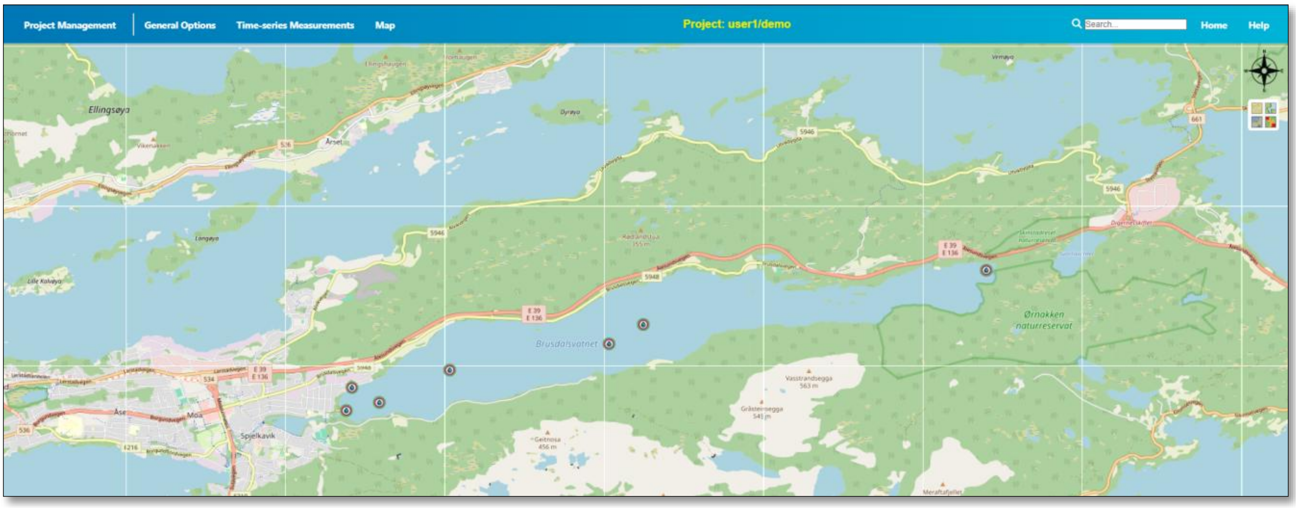


Figure 2.12. Map of observation points

- *Input points directly:* This method can be used to add observation points to the list directly. The user can copy and paste attributes of one point in the first row, then click the "***Add row***" button to add a new row and provide attributes of a new point.

- *Create a cross-section:* A cross-section is used to describe the geometric characteristics of rivers, channels, or flow branches, and is especially important for 1D hydrodynamic simulations and 1D-2D coupled models. Cross-sections define the relationship between water level, flow area, wetted perimeter, and discharge, thereby directly influencing the computed water levels and flow velocities. The steps below show how to create a cross-section:

- *Step 1:* Provide the name of the cross-section in the field "***Name***" (**Figure 2.13**). If the user does not specify the name of the cross-section at this step, the application will automatically get a default name of "***Cross-Section***".

- *Step 2:* Click the "***Draw on map***" button. This window temporarily disappears, the map is shown, and the user can draw a cross-section (**Figure 2.14**).

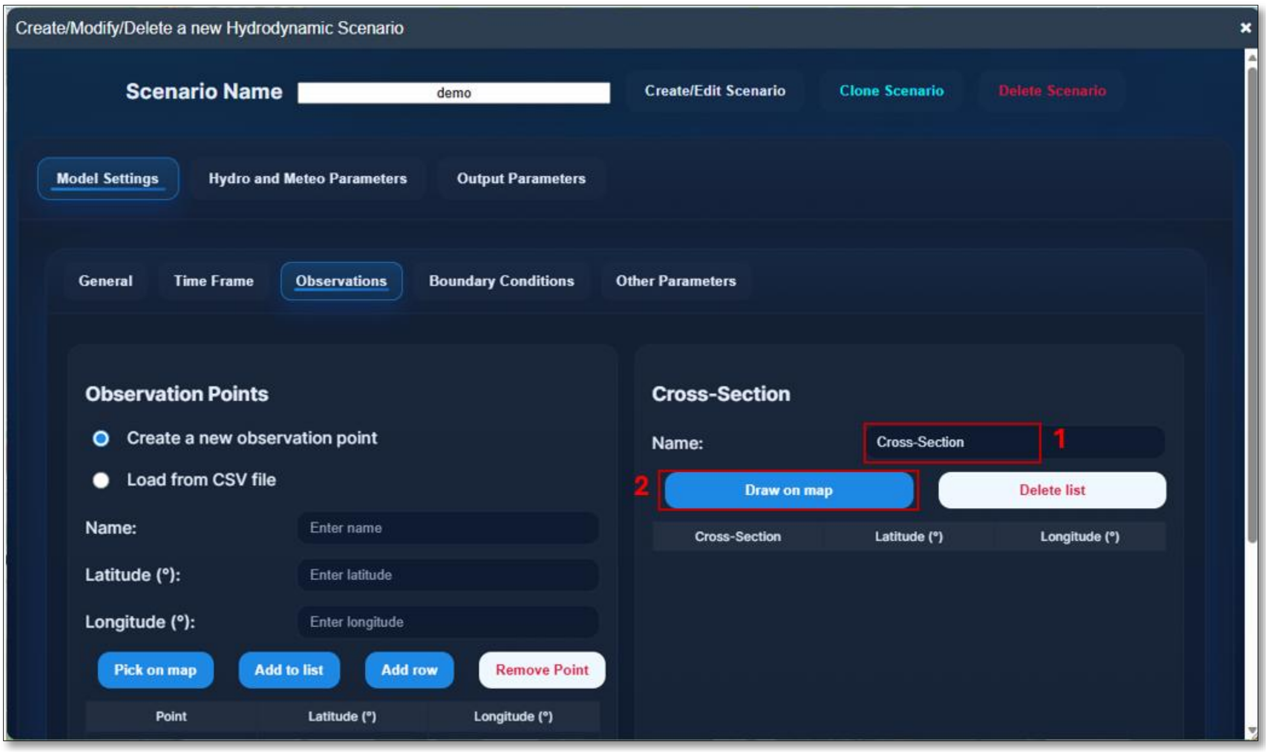


Figure 2.13. Create a cross-section

- *Step 3:* The user starts selecting the first point of the cross-section by using the left mouse button, selects other vertices, and finishes by using the right mouse button. The user can see a pop-up menu describing how to create a cross-section that is shown in **Figure 2.14**.

Figure 2.14. Draw a cross-section on the map

NOTE: In some (rare) cases, users may occasionally see that the number of points selected on the map does not match the number of points displayed in the list in Figure 2.15. For

example, the user selects 03 points on the map, but the number of points displayed in the list is more than 3. In this case, the best solution is to clear the list and select the points again.

- *Step 4:* After the user finishes, the window in *step 2* will appear with the vertices of the cross-section selected (**Figure 2.15**). If the user wants to entirely delete the existing cross-section, hit the "***Delete list***" button and draw a new one.

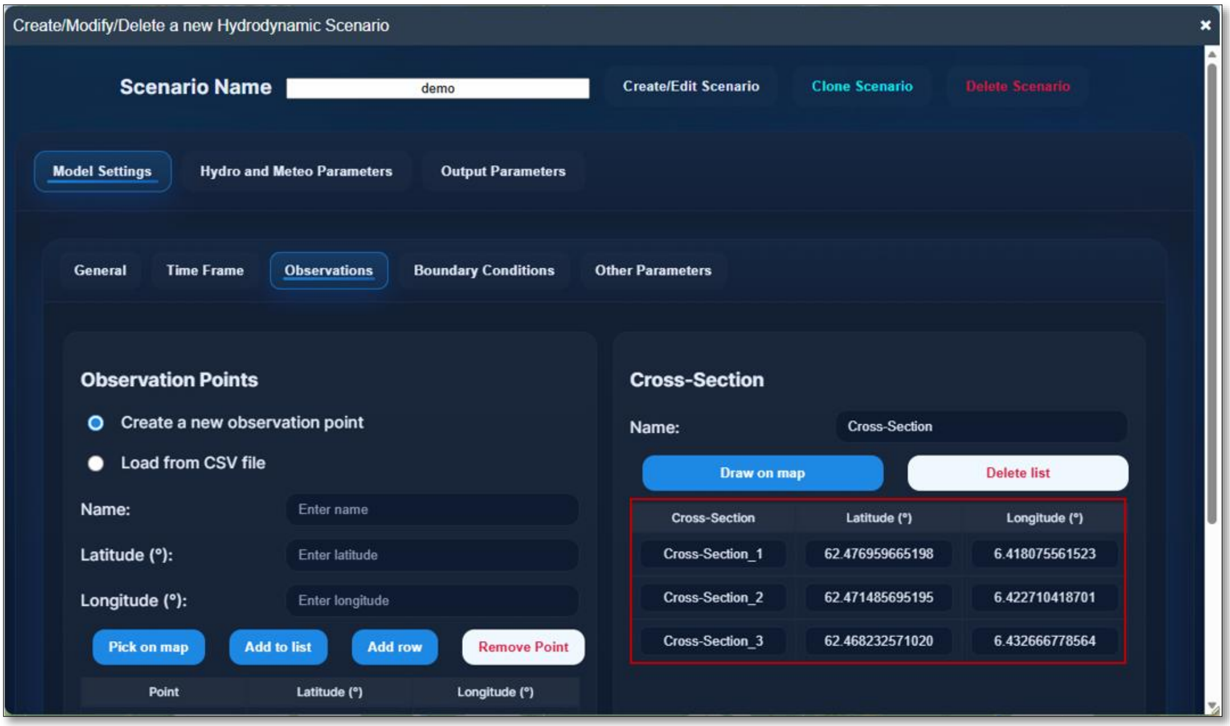


Figure 2.15. Finish creating a cross-section

❖ *Boundary Conditions*: These conditions are used to define external forcing acting on the computational domain, such as water levels, discharges, velocities, or wave- and constituent-related variables. Boundary conditions are essential for the model to function properly and determine how the system interacts with its surrounding environment.

Boundary conditions can be applied at open boundaries of 1D or 2D domains and may include water level boundaries, discharge boundaries, rating curves, or time-varying series. Proper specification of boundary conditions is crucial to ensure numerical stability, simulation accuracy, and a realistic representation of hydrodynamic processes.

➢ *Create a boundary condition:* Steps to create a boundary condition are similar to steps for creating a cross-section. The selected boundary conditions' vertices are shown in **Figure 2.16**. **If a similar error occurs when selecting boundary conditions (like when creating a cross-section), it is recommended to clear the list (by hitting the "*Delete BC*" button) and redraw a new boundary condition**.

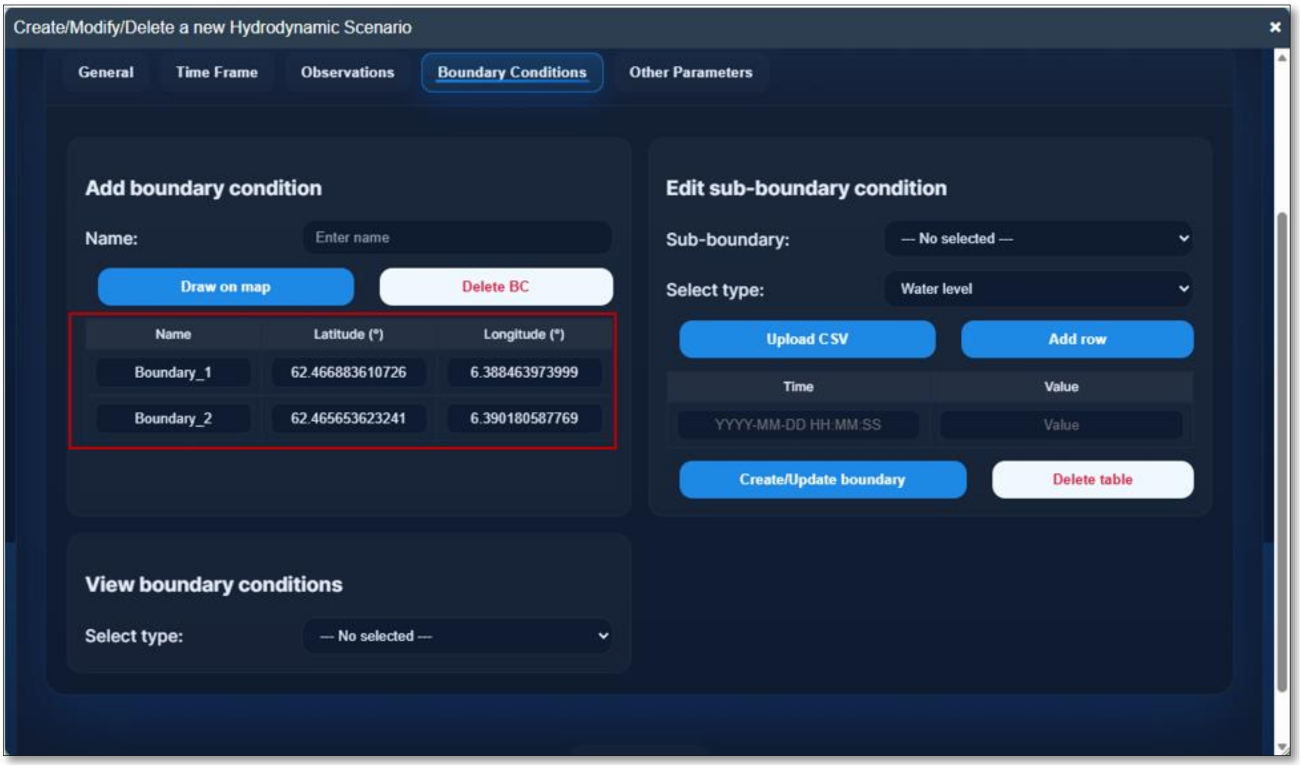


Figure 2.16. Defining a boundary condition

➢ *Add data to the boundary condition*: After the user successfully adds boundary conditions, a list of created boundaries will be added to the list in the field "***Sub-boundary***" (**Figure 2.17**).

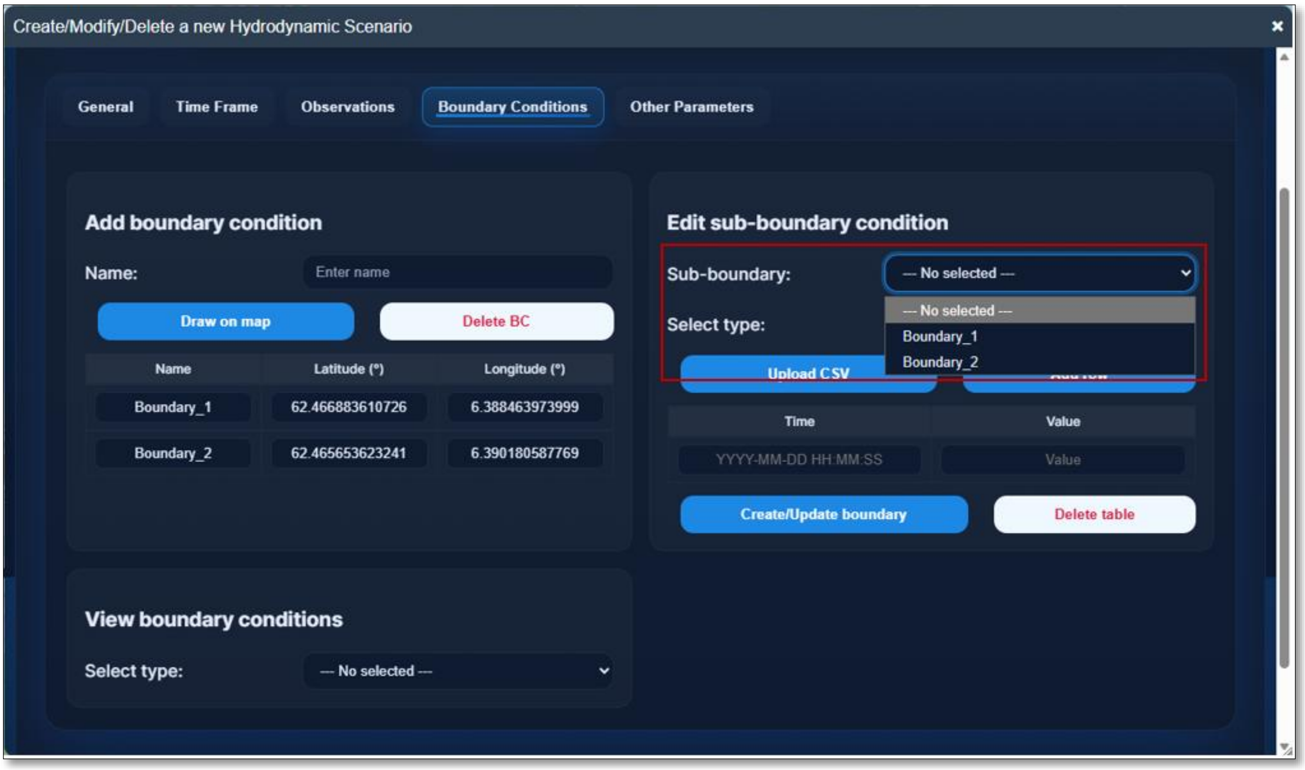


Figure 2.17. Add data to a boundary condition

The steps below show how to add a water level/contaminant for each sub-boundary condition (**Figure 2.18**):

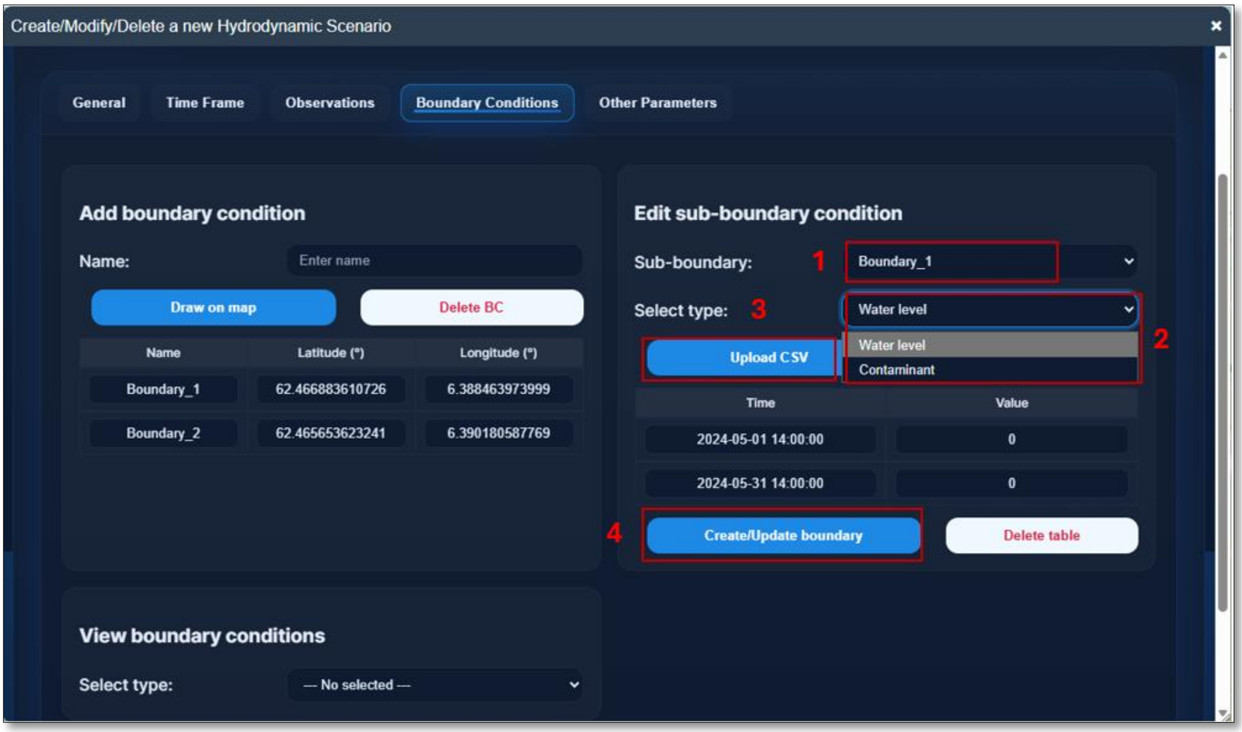


Figure 2.18. Define data for a sub-boundary condition

- *Step 1*: Select a specific sub-boundary from the list in the field "***Sub-boundary***".
- *Step 2*: Select either "***Water level***" or "***Contaminant***" in the field "***Select type***".
- *Step 3*: The user can either upload this data from a CSV file or copy and paste data into this table. The structure of the data used for the sub-boundary condition is described in **Figure 2.19**. **If the data for these options is not available, set them all to zero**. The user can use the "***Add row***" button or the "***Delete table***" button in **Figure 2.16** to add a new row or delete the table entirely.

Time	Value
2024-05-01 12:00:00	0
2024-05-31 12:00:00	0

Figure 2.19. Data used for a sub-boundary condition

NOTE: By default, the application is using the time zone in UTC (Coordinated Universal Time). The time difference is for display only and does not affect the calculation results.

- *Step 4*: After time-series data for each sub-boundary is added properly, the user can hit the "***Create/Update boundary***" button to save data into the database.

Repeat all the abovementioned steps to add or modify data for each sub-boundary condition.

- *View added sub-boundary condition*: After all data for the sub-boundary condition is prepared, the user can recheck these boundary conditions via the "***View boundary conditions***" panel (**Figure 2.20**).

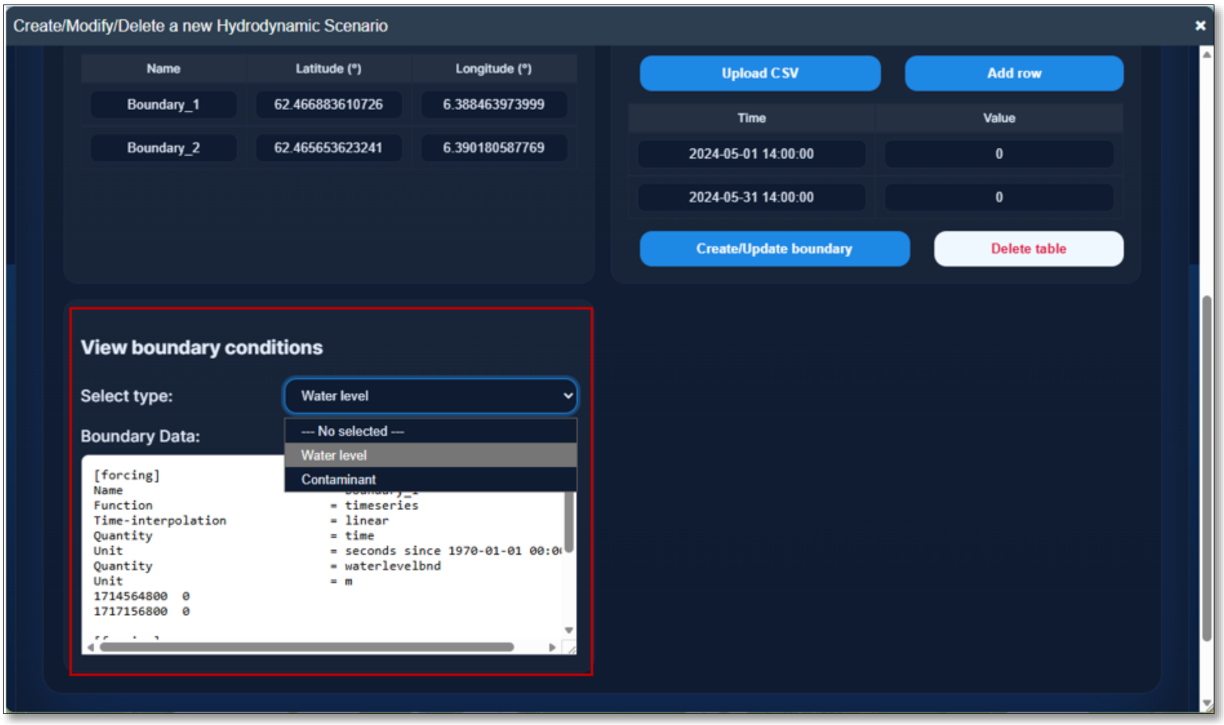


Figure 2.20. View assigned sub-boundary conditions

- *Other Parameters*: The user can define other initial parameters, such as defining a temperature model, assigning an initial water level, salinity, or temperature for the whole area in this sub-tab (**Figure 2.21**).

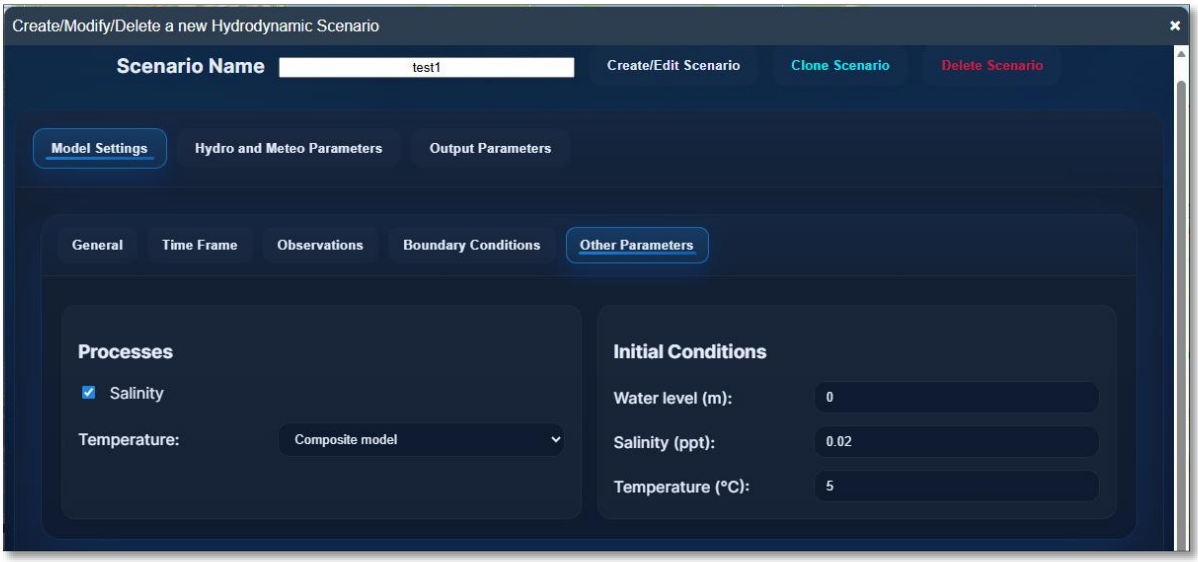


Figure 2.21. Set up other initial parameters

2.2.2. Hydro and Meteo Parameters

Hydrological data and meteorological data provide essential external forcing for hydrodynamic simulations. Hydrological data typically includes river discharges, water levels, inflows, and upstream or downstream boundary conditions, which control the flow regime and water level variations within the river and channel network.

Meteorological data represent the atmospheric forcing acting on the water surface, including wind speed and direction, air pressure, rainfall, evaporation, and heat fluxes. These data influence surface stresses, water level setup, circulation patterns, and, when relevant, water temperature and transport processes. Accurate and consistent hydrological and meteorological data are therefore crucial to ensure realistic and reliable Delft3D FM simulation results.

❖ *Hydrological Parameters*: The user can create a new data source for hydrological parameters directly from the map or upload data from a CSV file.

➢ *Create a data source from the map:* The user can follow the steps below to add a new data source (**Figure 2.22**):

- *Step 1*: Activate the "***Create a new source/sink***" option.

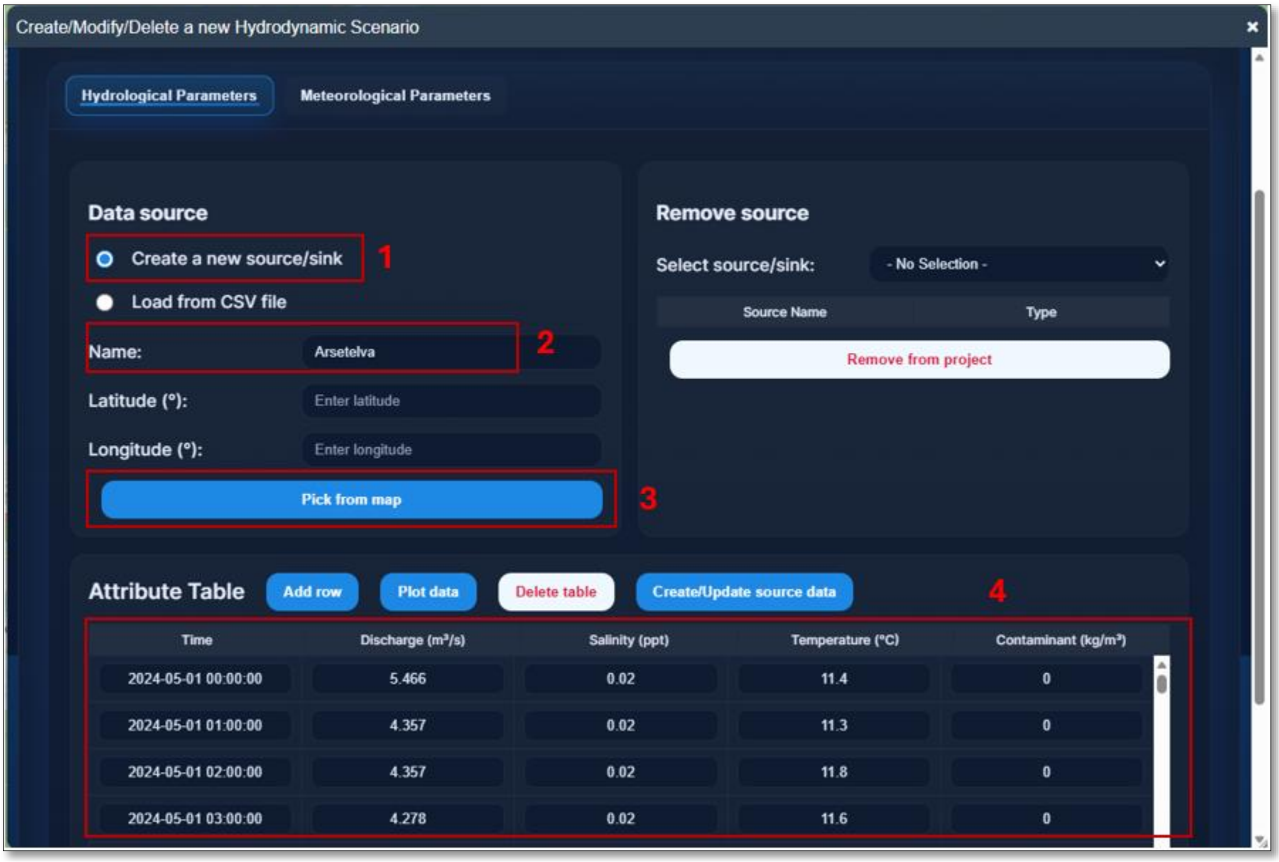


Figure 2.22. Set up the hydrological parameters from the map

- *Step 2*: Define the wanted source's name in the field "***Name***".
- *Step 3*: Click the "***Pick from map***" button, which temporarily hides this window and shows the map where the user can select the point with the left mouse button. Once the user completes selecting a point on the map, its locations will be added in the "***Latitude***" and "***Longitude***" fields.
- *Step 4*: The user must fill out the necessary information in the "***Attribute Table***" field. The data for this area can be added manually or copied and pasted from a CSV file or an Excel sheet. The user can use the "***Add row***" button to add a new observation or hit the "***Delete table***" button to clear the data of the selected source.

➢ *Upload data source from a CSV file*: The following steps show how to upload a data source from a CSV file (**Figure 2.23**):

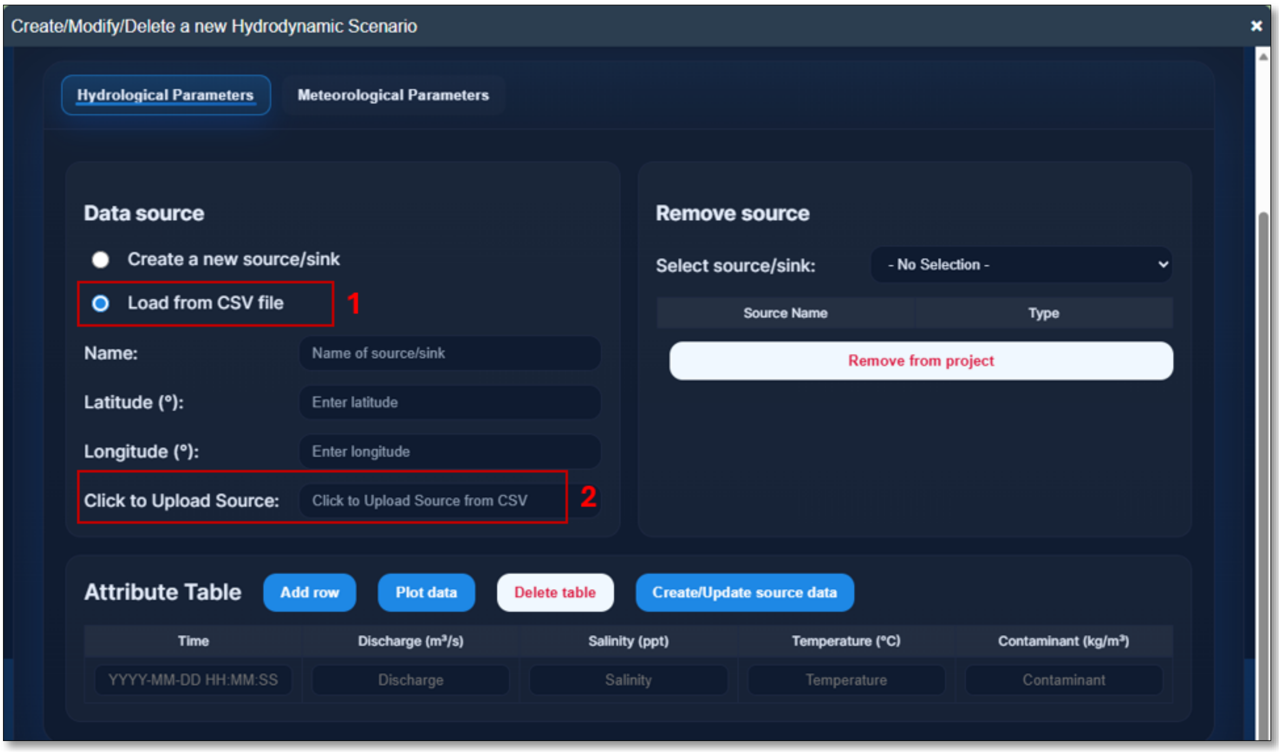


Figure 2.23. Set up the hydrological parameters from a CSV file

- *Step 1*: Activate the "***Load from CSV file***" option.
- *Step 2*: Click on the title "***Click to Upload Source***" to open and upload a CSV file. Once the CSV file is uploaded, all the necessary information from this source will be added automatically (**Figure 2.24**).

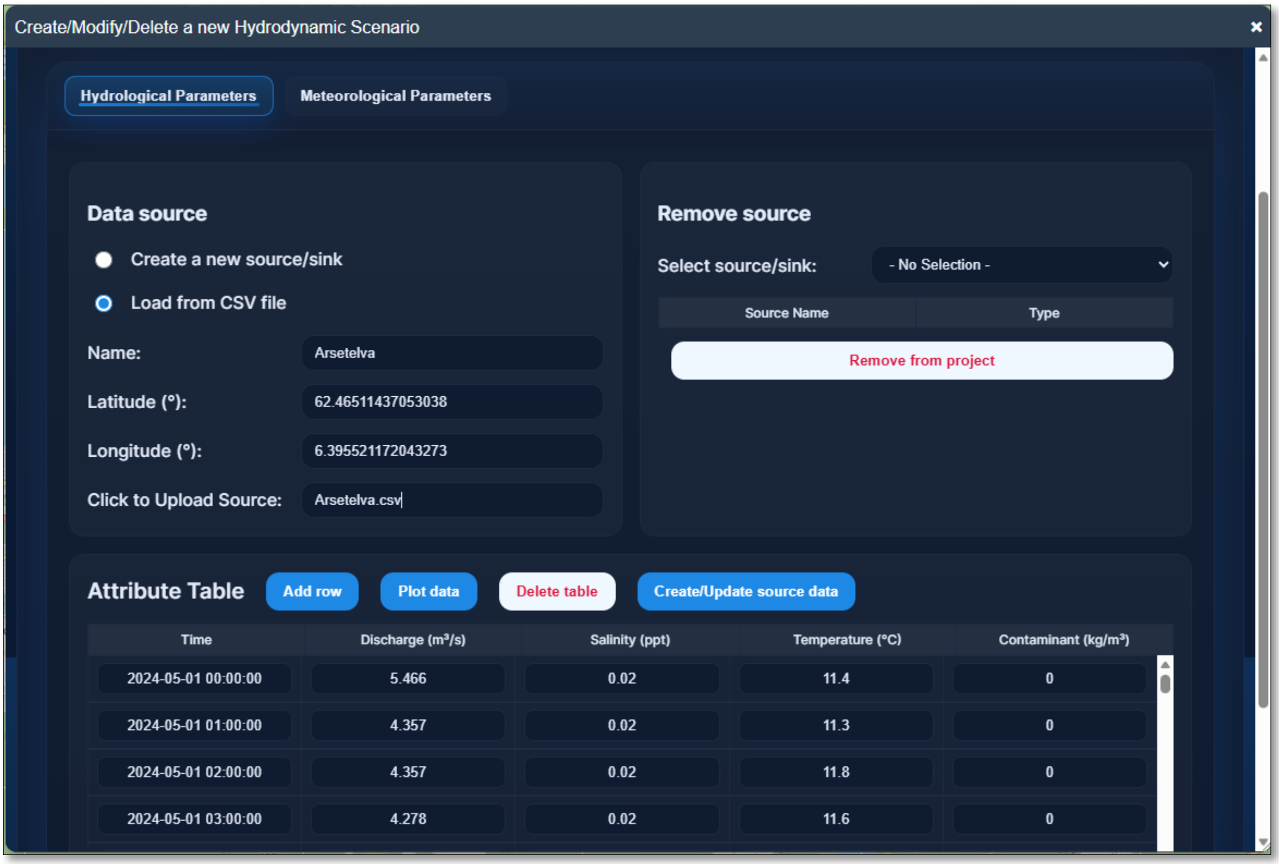


Figure 2.24. Fill hydrological parameters from a CSV file

Once data is loaded into the "***Attribute table***", the user can hit the "***Plot data***" button to draw all data (**Figure 2.25**) before deciding to save it into the project by hitting the "***Create/Update source data***" button.

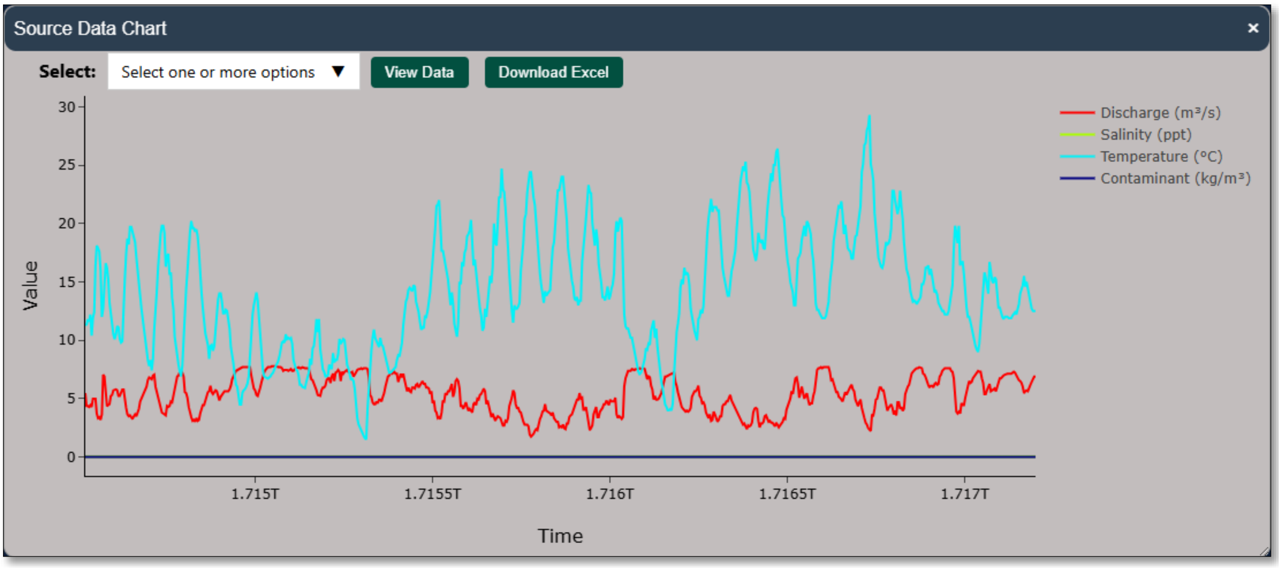


Figure 2.25. Plot hydrological parameters

All assigned data sources for hydrological parameters are listed in the panel "***Remove source***".

From this panel, the user can remove any source/sink by selecting it from the list "***Select source/sink***" and then hitting the "***Remove from project***" button to delete the selected source/sink (**Figure 2.26**).

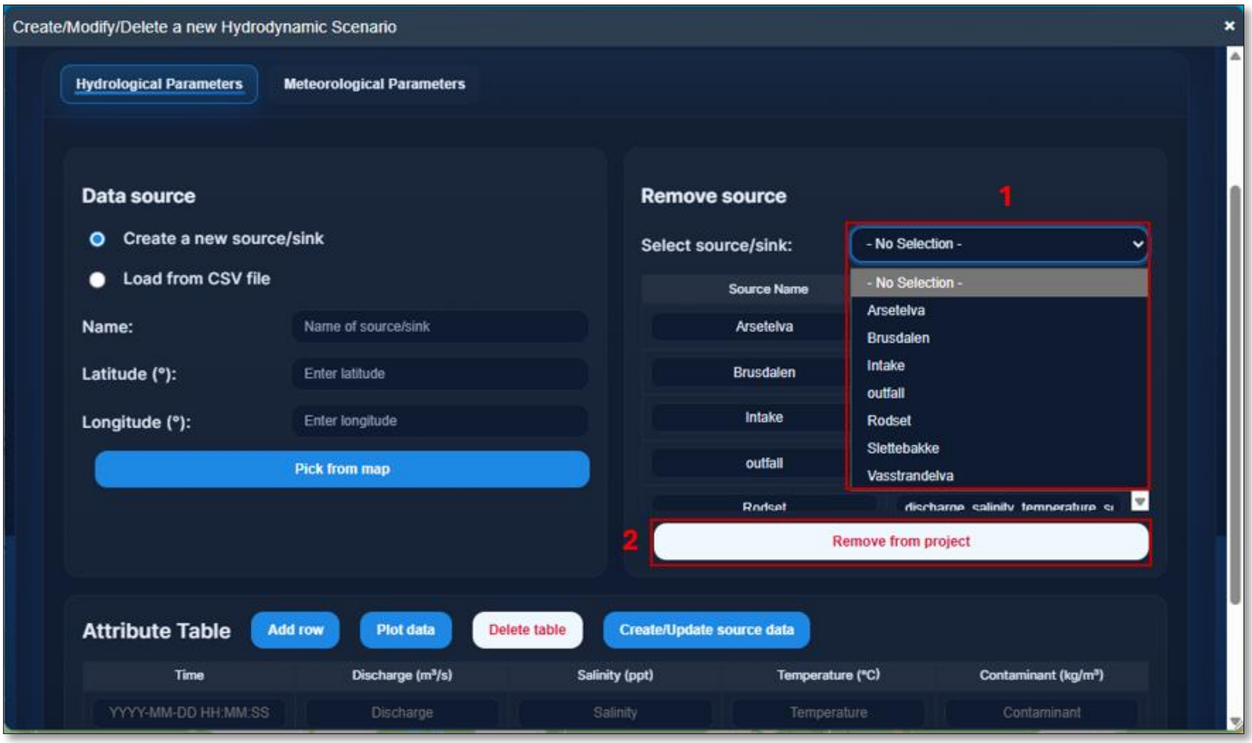


Figure 2.26. Remove source/sink from project

❖ *Meteorological Parameters*: The user can create meteorological data by importing it from a CSV file (**Figure 2.27**). The steps below show how to upload a CSV file:

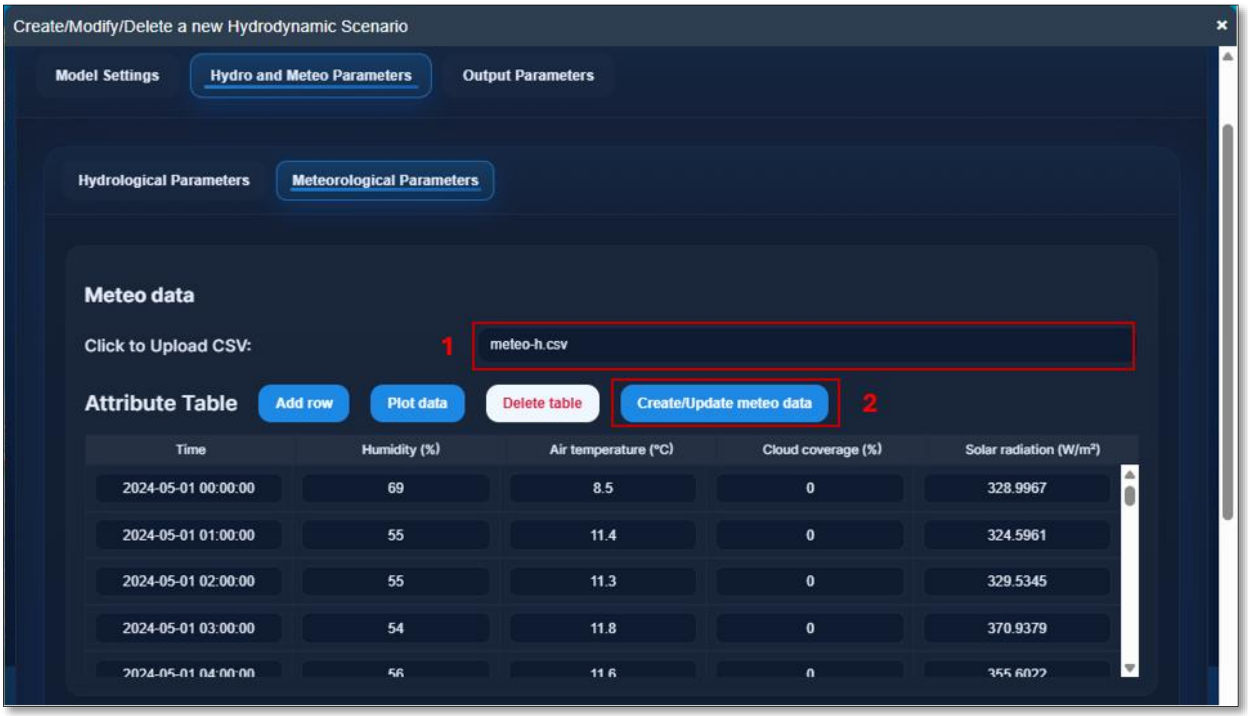


Figure 2.27. Upload meteorological data

- *Step 1*: Click on the field "***Click to Upload CSV***" and then select data from a CSV file. The structure of the meteorological data is described in **Figure 2.28**.

Time [-]	Humidity [%]	Air temperature [Â°C]	Cloud coverage [%]	Solar radiation [W/m2]
2024-05-01 00:00:00	69	8.5	0	328.9967
2024-05-01 01:00:00	55	11.4	0	324.5961
2024-05-01 02:00:00	55	11.3	0	329.5345
2024-05-01 03:00:00	54	11.8	0	370.9379
2024-05-01 04:00:00	56	11.6	0	355.6022
2024-05-01 05:00:00	54	12.2	0	370.0475
2024-05-01 06:00:00	63	10.4	0	336.4317
2024-05-01 07:00:00	63	11.9	0	367.3785
2024-05-01 08:00:00	63	12.4	0	349.7933
2024-05-01 09:00:00	47	16.4	0	370.534
2024-05-01 10:00:00	42	18.1	0	345.1109
2024-05-01 11:00:00	43	17.9	0	356.1852
2024-05-01 12:00:00	40	17.6	0	397.3989
2024-05-01 13:00:00	46	16	0	395.6157
2024-05-01 14:00:00	89	12	0	405.4828

Figure 2.28. The structure of meteorological data

- *Step 2*: Once the meteorological data is loaded successfully, the user can plot to check data by using the "***Plot data***" button and save data to the project by using the "***Create/Update meteo data***" button (**Figure 2.29**). The user also clicks on the "***Delete table***" button to entirely remove the existing data and repeat the above steps to upload a new dataset.

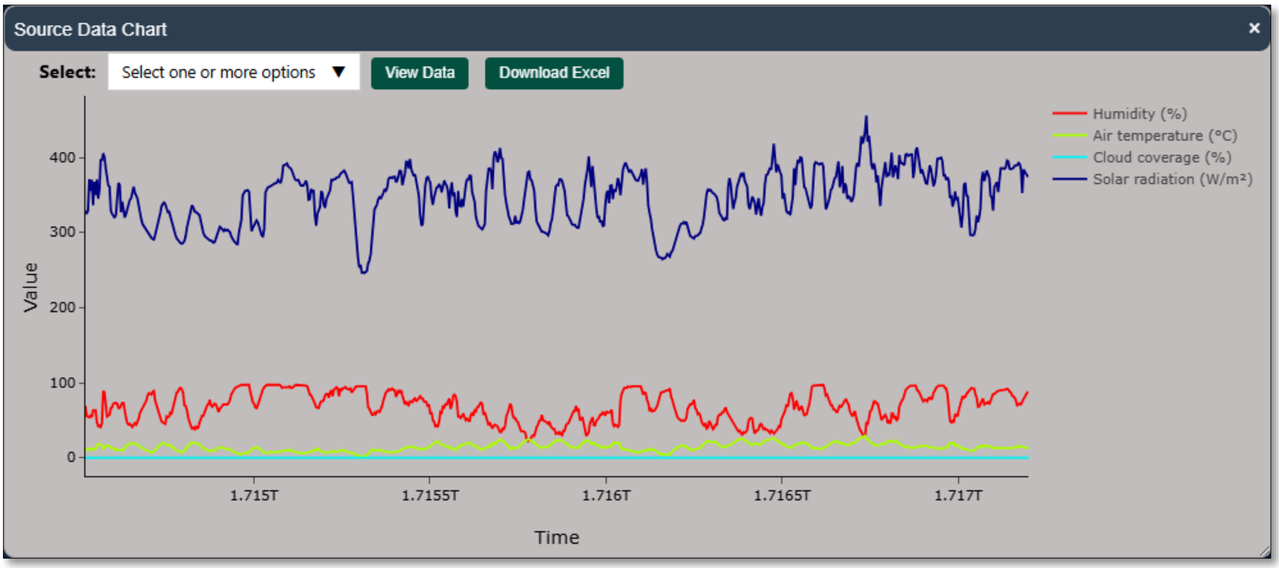


Figure 2.29. Plot meteorological parameters

Weather data are used to represent atmospheric forcing acting on the computational domain, including wind speed and direction, air pressure, rainfall, evaporation, and other surface-related fluxes. These inputs influence surface stresses, water level variations, circulation patterns, and, in some applications, water temperature and transport processes. Proper specification of weather data helps ensure a realistic interaction between the atmosphere and the water system, contributing to the accuracy and reliability of Delft3D FM simulation results.

To add weather data to the project, the user can follow the steps below (**Figure 2.30**):

- *Step 1*: Click on the field "***Select item***" in the panel "***Weather data***" and select "***Wind: Magnitude and Angle***" from the dropdown list.
- *Step 2:* Click on the field "***Upload CSV***" and select a CSV file. After an appropriate CSV file format is selected, the time-series data is automatically loaded into the table.
- *Step 3:* Click on the "***Create/Update Weather data***" button to save data into the project.

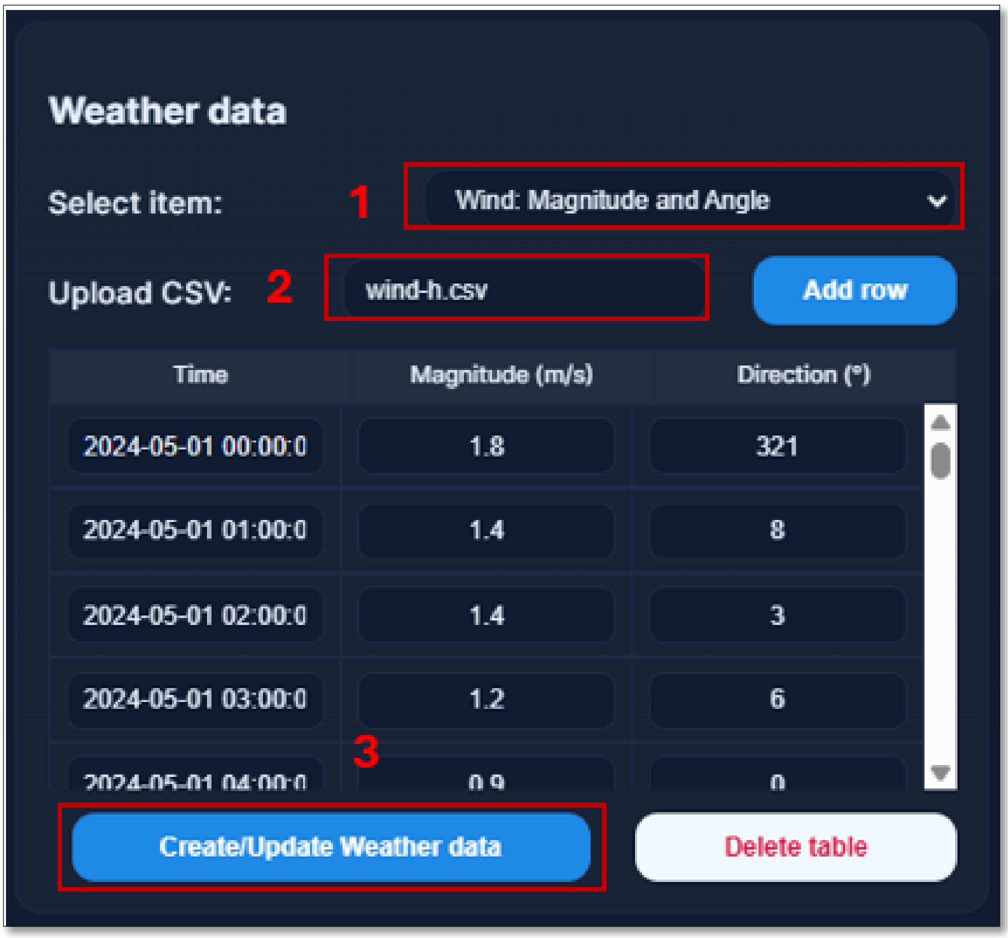


Figure 2.30. Create weather data

If a CSV is unavailable, the user can click the "***Add row***" button to add a new row, then provide the corresponding data in the table, and complete the process by clicking the "***Create/Update Weather data***" button. The "***Delete table***" button can be used to clear the data in the table entirely.

2.2.3. Output Parameters

This tab allows the user to set up outputs and their properties (i.e., start time, stop time, and interval of each time stamp) during the simulation. There are three important sub-panels in this tab: ***History***, ***Map***, and ***Water Quality*** (**Figure 2.31**). The ***History*** and ***Map*** options allow (or don't allow) Delft3D FM to write time-series-based and spatial measurements and information during a HYD simulation. The "***Water Quality***" option is very important and must be selected if the user wants to run a water quality (WAQ) simulation later, because this will be input to run the WAQ simulation. If both ***History*** and ***Map*** options are NOT selected, the HYD simulation still runs, but there is no output for HYD visualization.

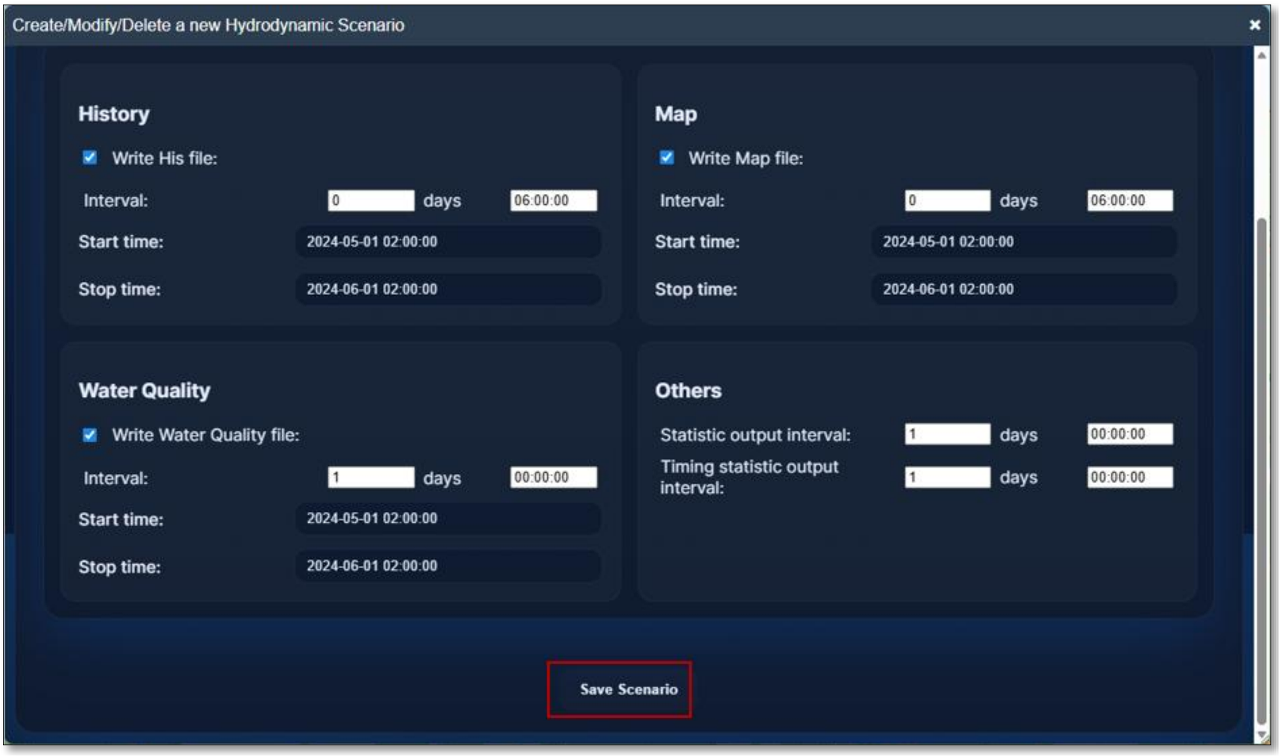


Figure 2.31. Set up output parameters for a HYD simulation

After all the above processes are finished, the user must click the "***Save Scenario***" button to save the parameters into the project. Now, the scenario is ready to run the HYD simulation.

2.3. HYD scenario modification

Once a HYD scenario is available, the user can clone it to a new scenario or delete it from the project destination by using the "***Clone Scenario***" and "***Delete Scenario***" buttons, respectively (**Figure 2.32**).

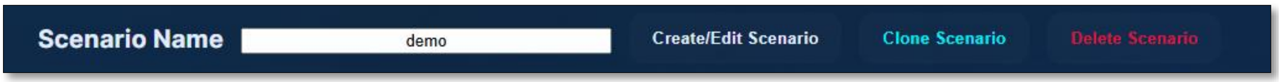


Figure 2.32. HYD scenario modification

In general, cloning or deleting a scenario will take time. Therefore, after the user uses these functions, the title of the corresponding buttons will change, showing that the progress is being processed. It is recommended to wait until the user sees a notification that shows the progress has finished to avoid unwanted problems (**Figure 2.33**).

Figure 2.33. Clone a HYD scenario

2.4. Running a HYD simulation

To run a HYD scenario, the user can click on "***Project Management***", then select "***Run HYD Simulation***". The user can see a window allowing them to select a scenario to run (**Figure 2.34**).

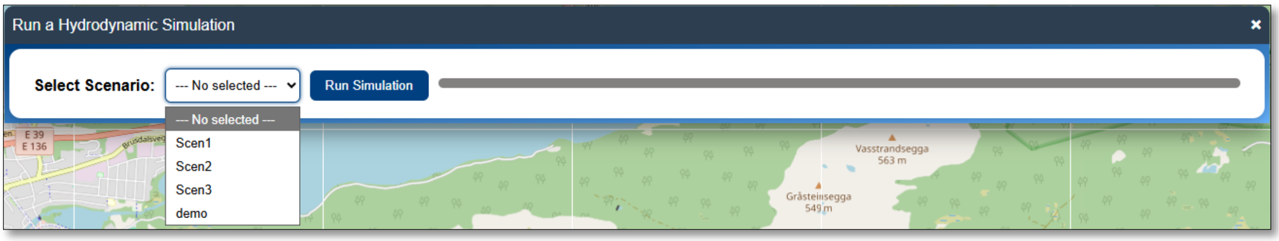


Figure 2.34. Select a HYD scenario

After selecting a scenario, click the "***Run Simulation***" button to run a HYD simulation. You can see the process of simulation (for example, percentage completed, time used, and time left for this simulation) (**Figure 2.35**).

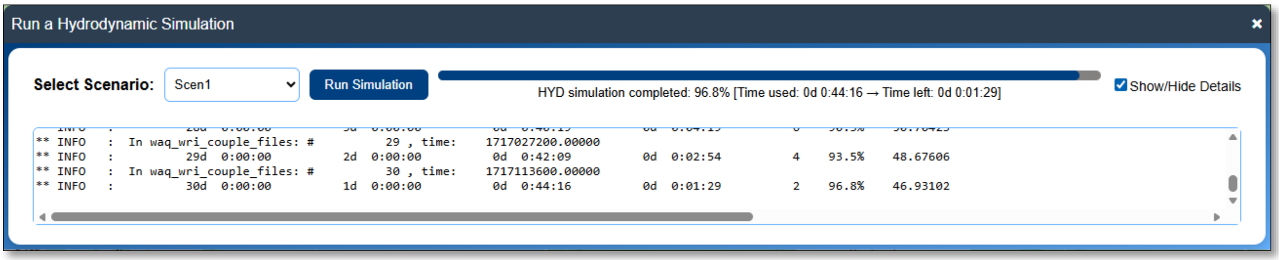


Figure 2.35. Run a HYD simulation

Running a HYD scenario will take time, depending on the size of the scenario. The user can close this window, perform other tasks, and return whenever they want to check the percentage completed. After the simulation is completed, the user can see the status as shown in **Figure 2.36** or **Figure 2.37** (depending on whether the window for running a HYD simulation is displayed or not at the time the simulation is finished).

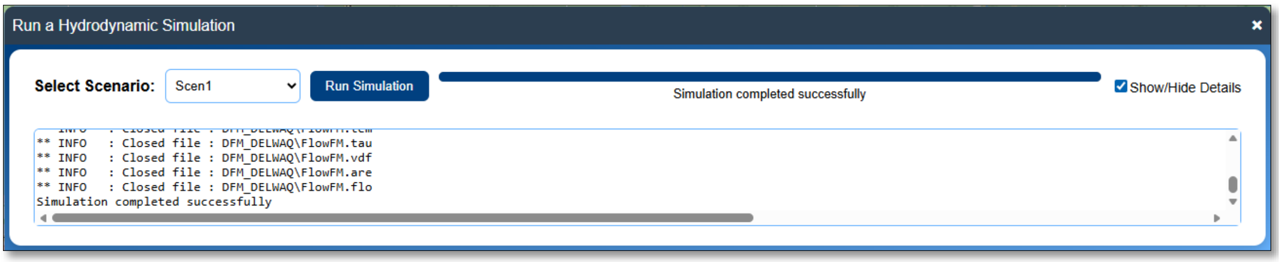


Figure 2.36. A HYD simulation is completed while the window is being opened

Figure 2.37. A HYD simulation is completed before the window is opened

3. Building and running a water quality simulation

The water quality module is a powerful tool for simulating the transport, dispersion, and transformation of substances in aquatic environments. These substances can include organic and inorganic pollutants, nutrients, dissolved oxygen, suspended solids, microorganisms, or other chemical compounds. The module is fully integrated with the 1D/2D hydrodynamic model, using water levels, flow velocities, and discharge data to simulate advection, lateral and longitudinal diffusion, and physicochemical and biological reactions in the water.

Key features of the water quality module in Delft3D FM include:

- ❖ *Transport models*: Calculate the movement of substances with the flow, including advection and lateral/longitudinal dispersion.
- ❖ *Reaction models*: Simulate in-water chemical and biological processes such as oxidation-reduction, nitrification/denitrification, organic matter decomposition, absorption, and release to the environment.
- ❖ *Sources and sinks*: Allow specification of pollutant sources, natural loads, or boundary concentrations to simulate the impacts of human activities and natural processes.
- ❖ *1D-2D interaction*: The module can operate in coupled 1D river networks and 2D floodplains or coastal areas, enabling the exchange of water quality variables between different domains.
- ❖ *Real-time analysis or forecasting*: Simulation results can be used to assess current conditions, predict pollutant behavior, support environmental management, and evaluate the impact of construction or hydraulic projects.

In this platform, we use several water quality models, grouped into three main categories:

- ❖ *Physical models*: Simulate the transport, dispersion, and spatial distribution of substances within the flow. Some models are used in the platform:
  - *Conservative and decaying tracers*: Simulate the transport of substances that are either stable or subject to decay over time.
  - *Suspended sediment (three fractions)*: Simulate the transport and deposition of suspended sediments, divided into three size fractions to reflect different settling and transport behaviors.
- ❖ *Chemical models*: Simulate chemical reactions in water, such as oxidation-reduction, neutralization, and other chemical-environment interactions. Some models are used:
  - *Simple oxygen*: A basic model tracking changes in dissolved oxygen concentrations.
  - *Oxygen and BOD (water phase only)*: Tracks dissolved oxygen along with biochemical oxygen demand (BOD) in the water column, reflecting organic matter decomposition processes.
  - *Cadmium*: Simulates the transport and transformation of cadmium into water, including adsorption and interactions with sediments.
  - *Eutrophication (Eutrof 1a)*: Models nutrient dynamics, algal growth, and oxygen variations, supporting assessment and prediction of eutrophication in aquatic systems.
  - *Trace metals*: Simulates the transport, accumulation, and redistribution of other heavy metals in water, including settling, adsorption, and resuspension processes.
- ❖ *Microbial models*: Simulate the growth, decay, and interactions of microorganisms or biological substances in the water. In this platform, a *coliform bacteria* model, which simulates the transport, decay, and growth of coliform bacteria in water, supporting microbial water quality assessment and public health risk evaluation, is used.

By simulating transport, chemical reactions, and biological processes, the water quality module of Delft3D FM is a critical tool for environmental assessment, water resources management, and the study of complex aquatic systems. Accurate specification of sources, boundary conditions, and reaction parameters is essential to ensure the reliability and accuracy of simulation results.

3.1. Setting up a WAQ scenario

A WAQ simulation can be executed with a HYD scenario prepared. To create a WAQ scenario, click on the option "***Project Management***" and then click "***WAQ Scenario***". Then, a window appears, allowing the user to select a prepared HYD scenario (**Figure 3.1**).

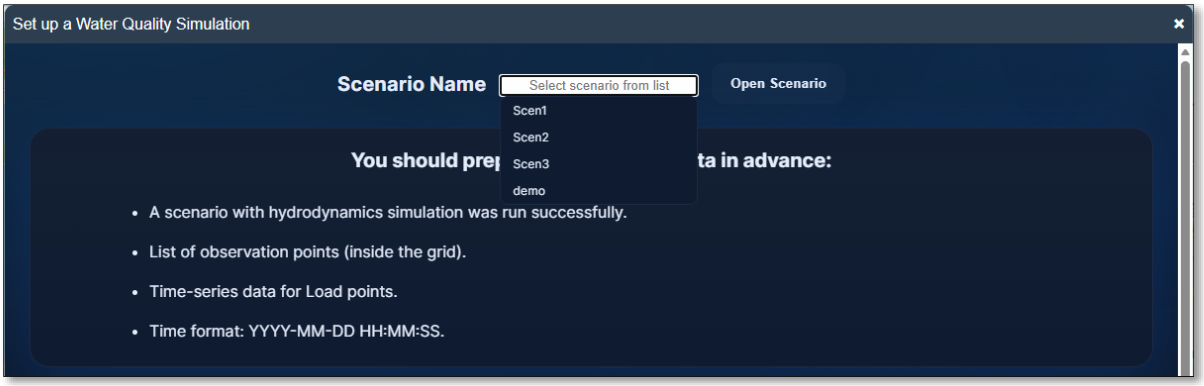


Figure 3.1. Select a prepared HYD scenario

After a HYD scenario is selected, if a WAQ scenario has been created before, the user can see the window as **Figure 3.2**.

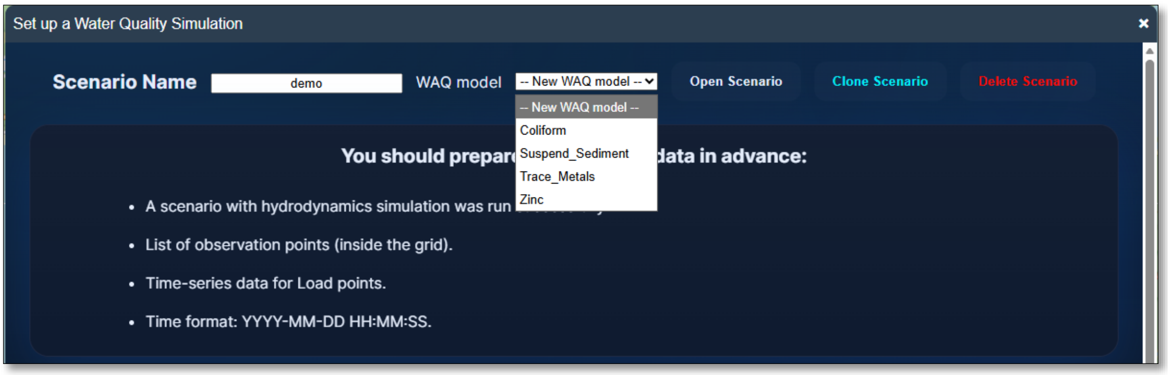


Figure 3.2. A HYD simulation with previous WAQ scenarios

Three tabs in this window allow the user to check and set up a new WAQ scenario. They are ***Model Overview***, ***Point Settings***, and ***Model Settings*** (**Figure 3.3**).

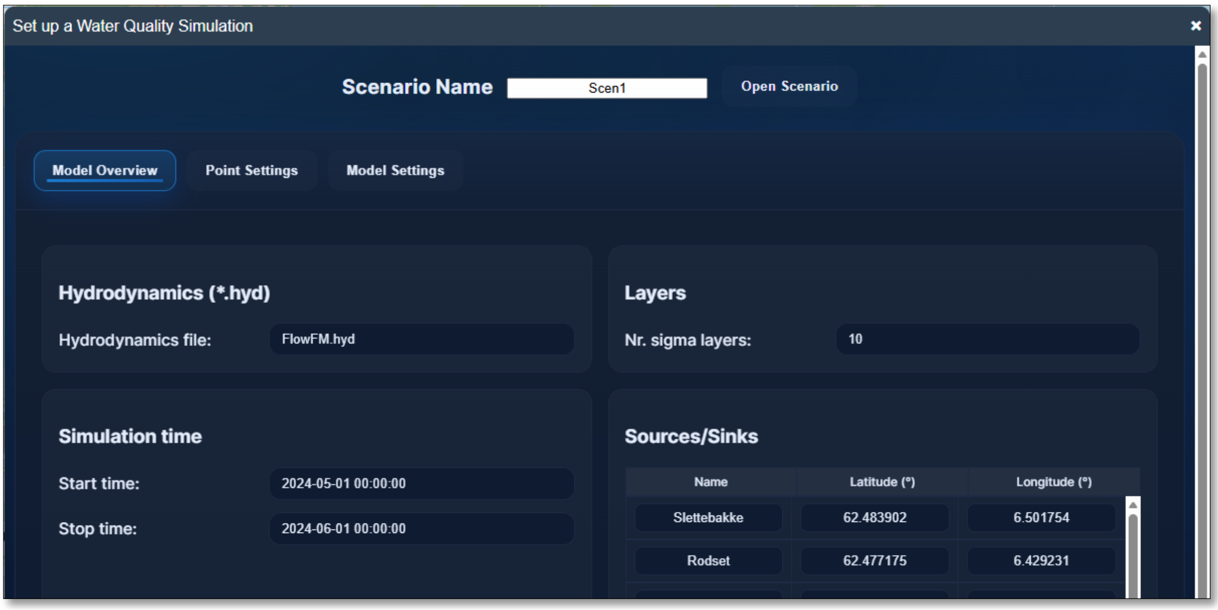


Figure 3.3. Create a WAQ scenario

3.1.1. *Model Overview*

This tab summarizes general information obtained from the selected HYD scenario, such as the hydrodynamic file (*.hyd), the number of layers, the simulation period, and sources/sinks. In general, the user doesn't need to change any information in this tab.

3.1.2. *Point Settings*

This tab allows the user to add observation points and define loads for the simulation. The user can follow these steps below to add or delete observations and/or loads:

- *Step 1*: Provide the name of an observation point (or a load). *Name of the load (s) in this tab must be the same name as the field "**Location**" in the table "**Time-Series Preparation**" in the tab "**Model Settings**".*
- *Step 2*: Click the "***Pick on map***" button, then this window disappears, allowing the user to select an observation (or a load) on the map. If the user does not specify the name, a default name is used (**Figure 3.4**).

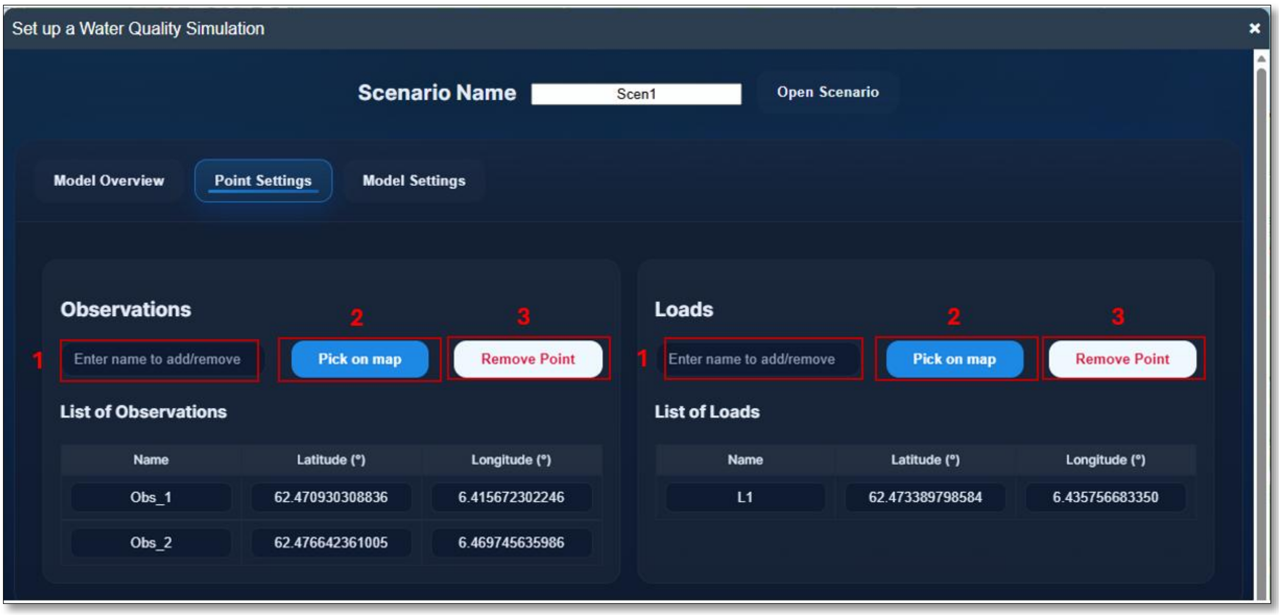


Figure 3.4. Define observations and loads for a WAQ scenario

- *Step 3*: If the user wants to delete an observation point (or load) from the table, type the name of the observation point (or load) from the list into the input text (in *step 1*), and then click on the "***Remove Point***" button.

3.1.3. *Model Settings*

In this tab, the user can define the model's parameters for a water quality simulation. The steps below show how to set up the WAQ parameters:

- *Step 1*: Select the category model used (physical, chemical, or microbial model).
- *Step 2*: Select the WAQ sub-model to use from the list.
- *Step 3*: After the user selects a WAQ model, the name of the model is generated automatically, and the associated values of this model will be automatically initiated in the fields "***Assigned Substance***" and "***Initial Value***". The user can change this name to another name; otherwise, the previous WAQ model (with the same name) will be overwritten (**Figure 3.5**).

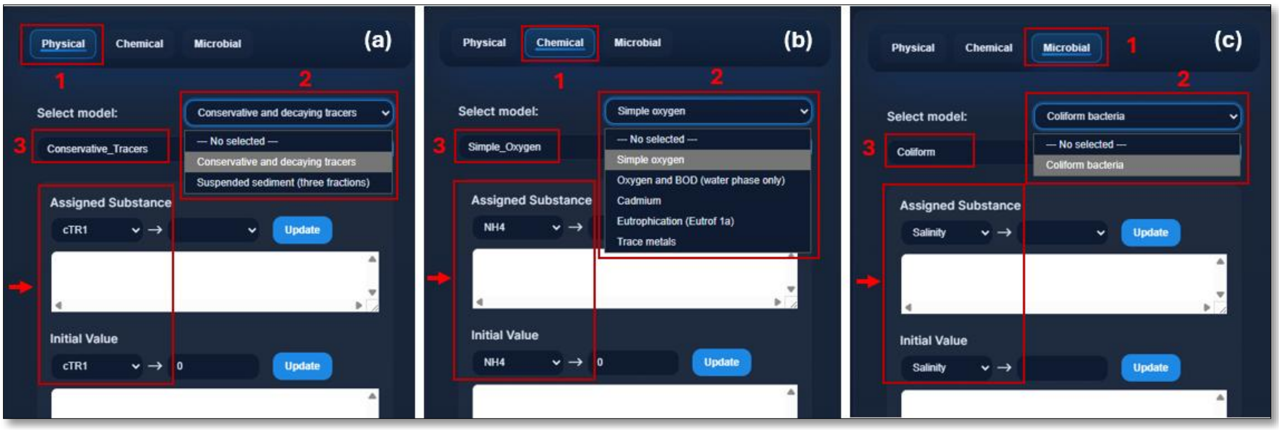


Figure 3.5. Define WAQ model: (a) Physical model; (b) Chemical model; (c) Microbial model

- *Step 4*: Provide time-series data for the WAQ model by uploading a CSV or typing data manually. The structure of a time-series for the WAQ model is described in **Figure 3.6**.

➢ *Add data manually*: Type data in the same format described in **Figure 3.6**. Click the "***Add row***" button to insert a new row.

➢ *Upload data from a CSV file*: Select the "***Upload CSV***" button and then select an appropriate CSV file from the user's machine.

timeLinear	Location	substance	value
2024-05-01 12:00:00	L1	ZNWTOT	0.154
2024-05-30 12:00:00	L1	ZNWTOT	0.154
2024-05-01 12:00:00	L2	ZNWTOT	0.154
2024-05-30 12:00:00	L2	ZNWTOT	0.154
2024-05-01 12:00:00	L3	ZNWTOT	0.154
2024-05-30 12:00:00	L3	ZNWTOT	0.154
2024-05-01 12:00:00	L4	ZNWTOT	0.154
2024-05-30 12:00:00	L4	ZNWTOT	0.154

Figure 3.6. Time-series for a WAQ model

- *Step 5*: Before selecting the "***Process TS Data***" button to process time-series data, the user has to make sure that the name of the field “***Location***” in the table “***Time-Series Preparation***” must be included in the table “***List of Loads***” in the tab “***Point Settings***” (**Figure 3.4**). Otherwise, the user will get a window showing the wrong data format for the selected WAQ model (**Figure 3.7**).

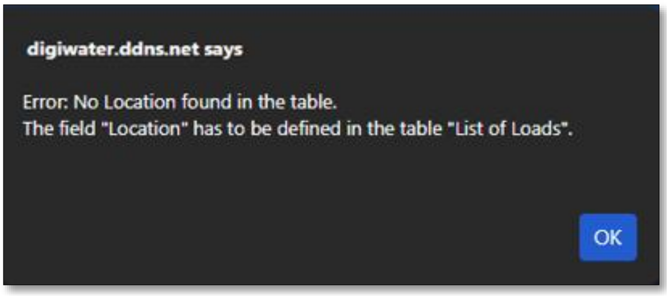


Figure 3.7. Undefined load for the WAQ model

- *Step 6*: After the user clicks on the “***Process TS Data***” button, the processed time-series data is shown in the field “***Post-process Data***” and other associated values for this time-series data are added into the list in the field “***Assigned Value***” (**Figure 3.8**).

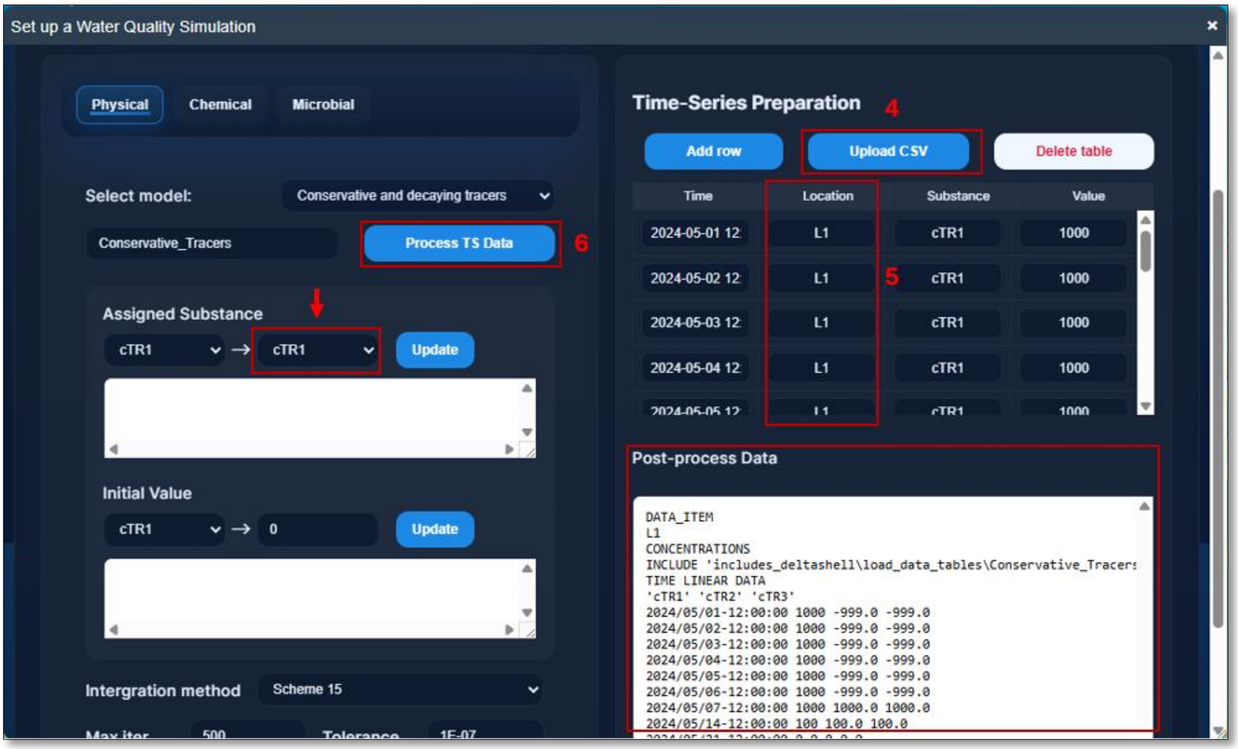


Figure 3.8. Process time-series data for the WAQ model

- *Step 7*: Once the time-series data is processed, the user can assign values for substances in the field “***Assigned Substance***” and their initial values in the field “***Initial Value***”. The user can use the “***Update***” button to add values from the dropdown list in the fields “***Assigned Substance***” and “***Initial Values***”. Values in these fields can be changed (*remember to include a space between all variables and operators*); an example of setting these values is shown in **Figure 3.9**. Finally, the user selects the button "***Save physical model***" (or "***Save chemical model***", "***Save microbial model***", depending on which category they use) to save the WAQ model into the project.

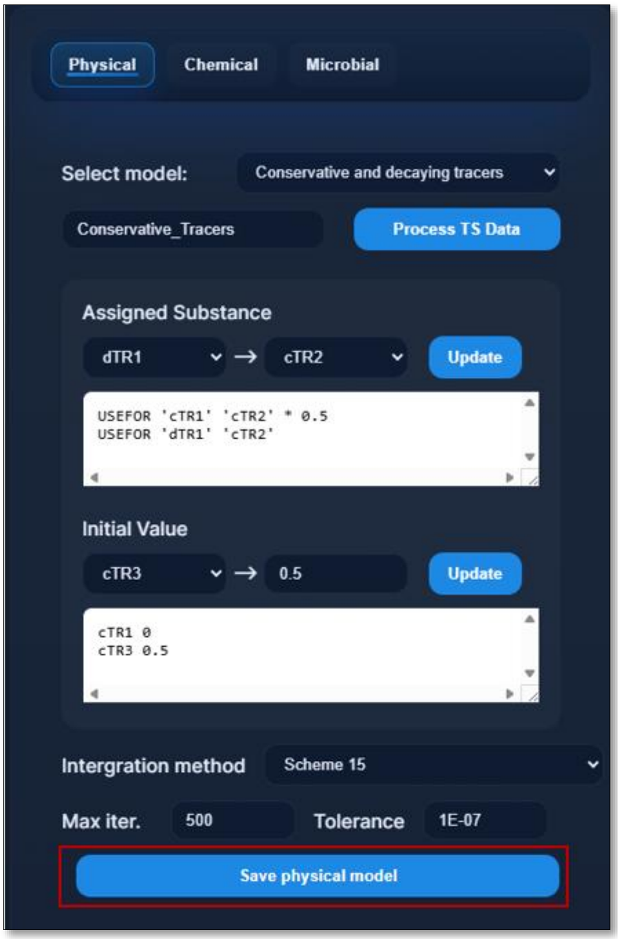


Figure 3.9. Define values and save the WAQ model

3.2. Running a WAQ simulation

A WAQ scenario only runs based on a prepared HYD scenario; therefore, the user must run a HYD scenario completely. To run a WAQ scenario, the user can click on "***Project Management***", then select "***Run WAQ Simulation***". The user must first select a HYD scenario, then select a WAQ scenario (**Figure 3.10**).

Figure 3.10. Select a WAQ scenario

After selecting a scenario, click the "***Run Simulation***" button to run a WAQ simulation. You can see the process of simulation (for example, percentage completed, status for this simulation) (**Figure 3.11**).

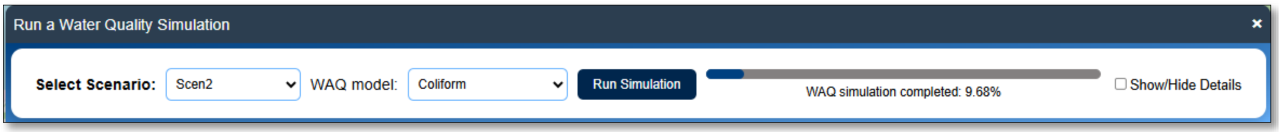


Figure 3.11. Run a WAQ simulation

After a WAQ scenario is completed, the user can see information in the window (**Figure 3.12**).

Figure 3.12. Complete a WAQ simulation

4. Visualization

To open HYD and/or WAQ simulations, click on the option "***Project Manager***" and then select "***Visualization***". The user can select a project from a drop-down list "***Scenario***" (**Figure 4.1**).

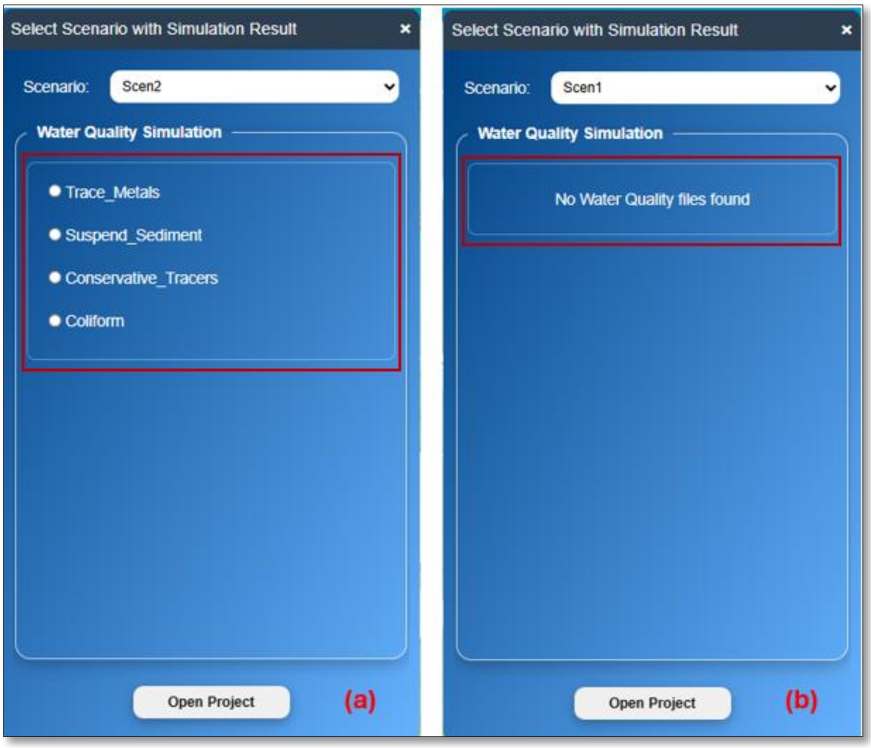


Figure 4.1. A scenario with (a) and without (b) WAQ simulation

If the selected project has both HYD and WAQ simulations, the names of all WAQ simulations will be listed in the field "***Water Quality Simulation***". Otherwise, the user can see a text "*No Water Quality files found*". The platform allows the user to visualize the HYD and the WAQ simulations simultaneously or separately by selecting or deselecting the WAQ simulation while opening a project. After selecting the specific scenario (with or without WAQ simulation), click the "***Open Project***" button to visualize the simulation(s).

Once a project is loaded, the user can use the menu to visualize different properties of the outputs. There are three main items that the user can select from the menu: ***General Options***, ***Time-Series Measurements***, and ***Map***. Items in each submenu will be different depending on each HYD and/or WAQ simulation imported.

4.1. General Options

Some options included in the "General Options" are shown in **Figure 4.2**.

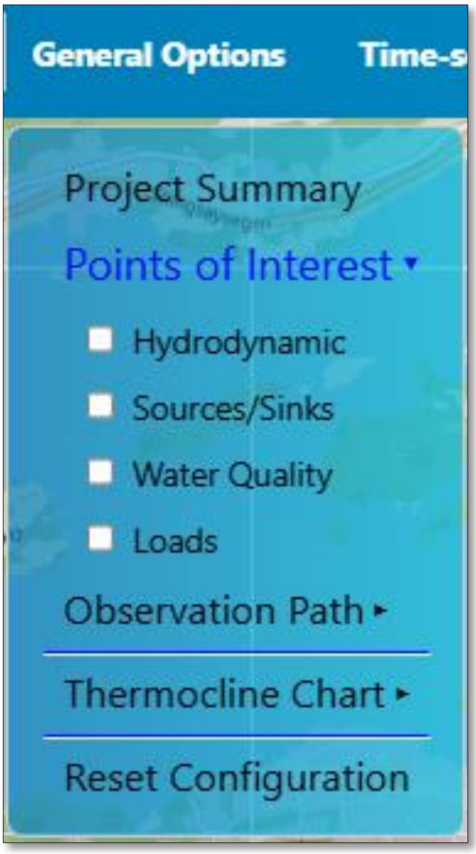
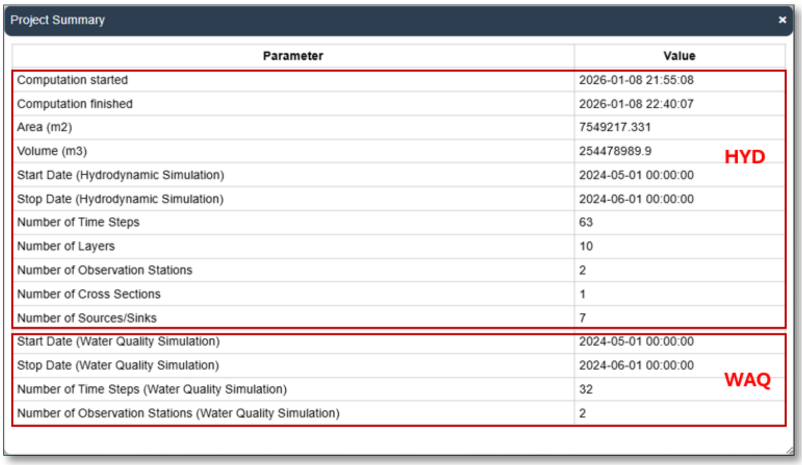


Figure 4.2. General Options menu

- *Project Summary*: This option summarizes important properties of HYD and/or WAQ simulations. If the user opens both the HYD and the WAQ simulations, two parts in this summary window are shown in **Figure 4.3**. If the user does not select a WAQ simulation while opening the project, the WAQ-related simulation part is not included in this window.



Project Summary

Parameter	Value
Computation started	2026-01-08 21:55:08
Computation finished	2026-01-08 22:40:07
Area (m2)	7549217.331
Volume (m3)	254478989.9
Start Date (Hydrodynamic Simulation)	2024-05-01 00:00:00
Stop Date (Hydrodynamic Simulation)	2024-06-01 00:00:00
Number of Time Steps	63
Number of Layers	10
Number of Observation Stations	2
Number of Cross Sections	1
Number of Sources/Sinks	7
Start Date (Water Quality Simulation)	2024-05-01 00:00:00
Stop Date (Water Quality Simulation)	2024-06-01 00:00:00
Number of Time Steps (Water Quality Simulation)	32
Number of Observation Stations (Water Quality Simulation)	2

Figure 4.3. Project summary option

❖ *Points of Interest*: This option allows the user to show or hide points of interest on the map (**Figure 4.4**). If the user does not select WAQ simulation, the options "***Water Quality***" and "***Loads***" are not available in this list.

Figure 4.4. Location of interest points

Once the option "***Hydrodynamic***" is selected, the user can click on the specific point to see more detailed information (**Figure 4.5**).

Figure 4.5. In-situ observations

❖ *Observation path*: This option allows the user to show or hide "***Cross-Sections***" (if HYD simulation has this in output) and define "***Profile***" from an available map (**Figure 4.6**).

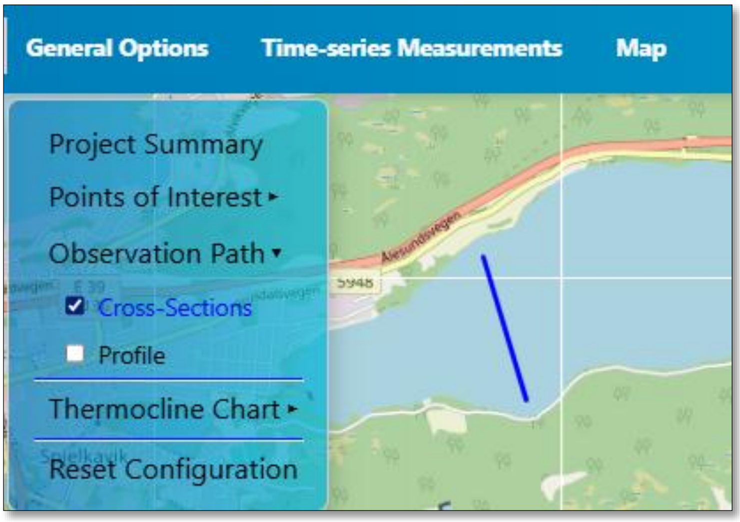


Figure 4.6. Cross-Sections option

By selecting a cross-section on the map, the user can select and visualize properties associated with this cross-section (**Figure 4.7**).

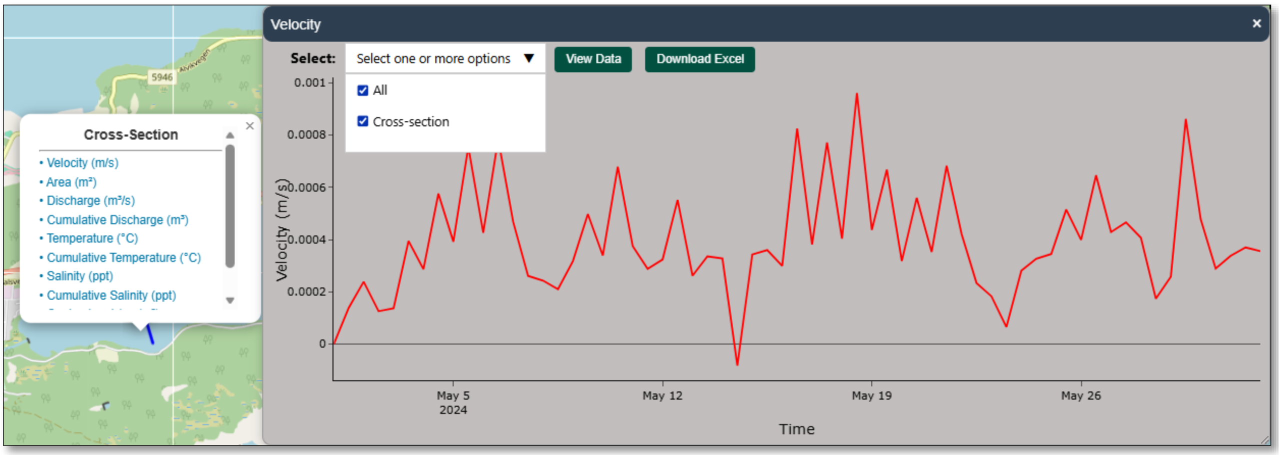


Figure 4.7. Cross-Sections properties

To use the option "***Profile***", one map layer must be available on the map. The use of this option will be described in the section map below.

❖ *Thermocline Chart*: The user can visualize a thermocline chart for a HYD simulation or a vertical profile for a WAQ simulation at a specific location by using the option "***Thermocline Chart/Depth vs. Temperature***" and "***Thermocline Chart/Depth vs. Water Quality***", respectively (**Figure 4.8**).

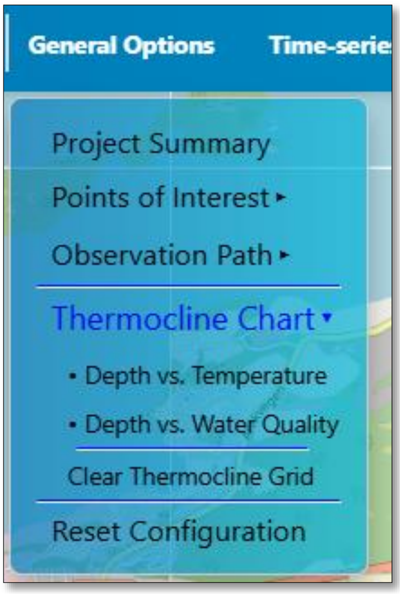


Figure 4.8. Thermocline Chart option

To activate a thermocline chart, follow the steps below:

- *Step 1*: Select the option "***Depth vs. Temperature***" or "***Depth vs. Water Quality***". If the user clicks on the option "***Depth vs. Water Quality***", a small area appears that allows the user to select which option in the WAQ simulation to draw a thermocline chart (**Figure 4.9**).

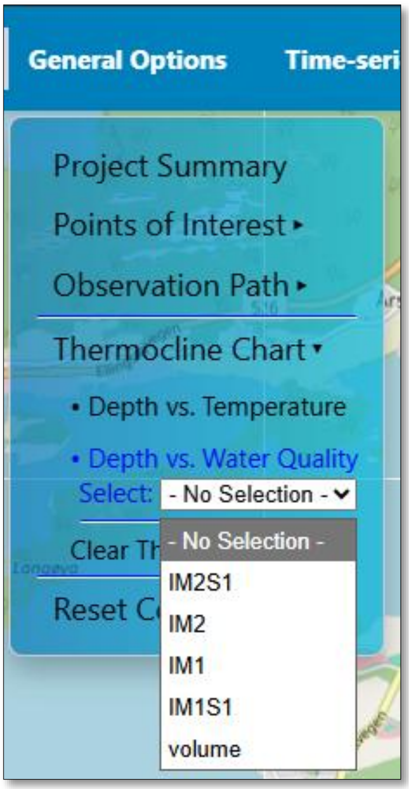


Figure 4.9. Thermocline Chart option for WAQ simulation

- *Step 2*: Select a location from the grid printed on the map (**Figure 4.10**). Change the name of the point if you prefer another name. Then, click the "***Plot Chart***" button to draw.

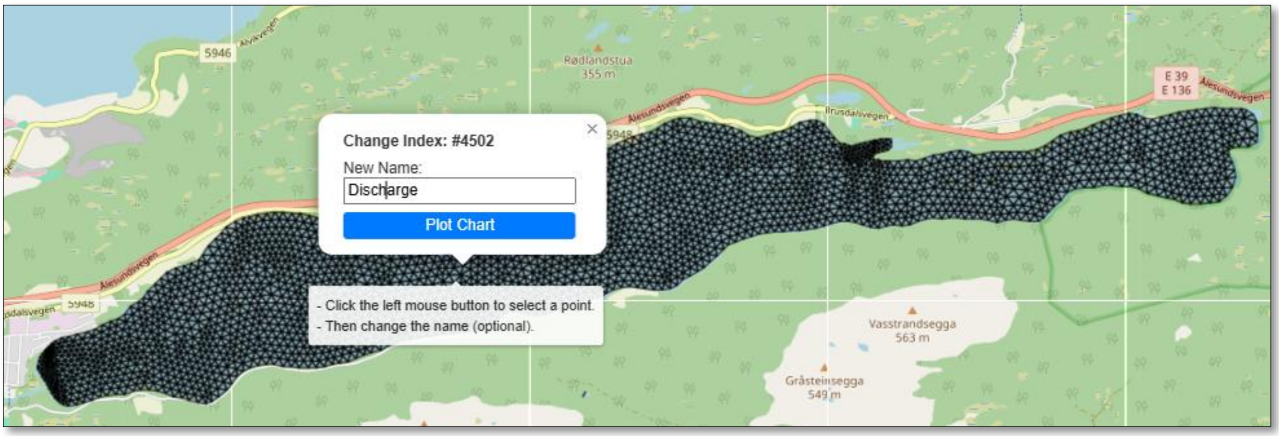


Figure 4.10. Select a location for the thermocline chart

- *Step 3*: A window containing a dynamic chart for HYD simulation (**Figure 4.11**) or WAQ simulation (**Figure 4.12**) appears, and the user can click the "***Play***" button to see the animation. The user can change the value in the dropdown menu in the field "Duration" to adjust the animation speed.

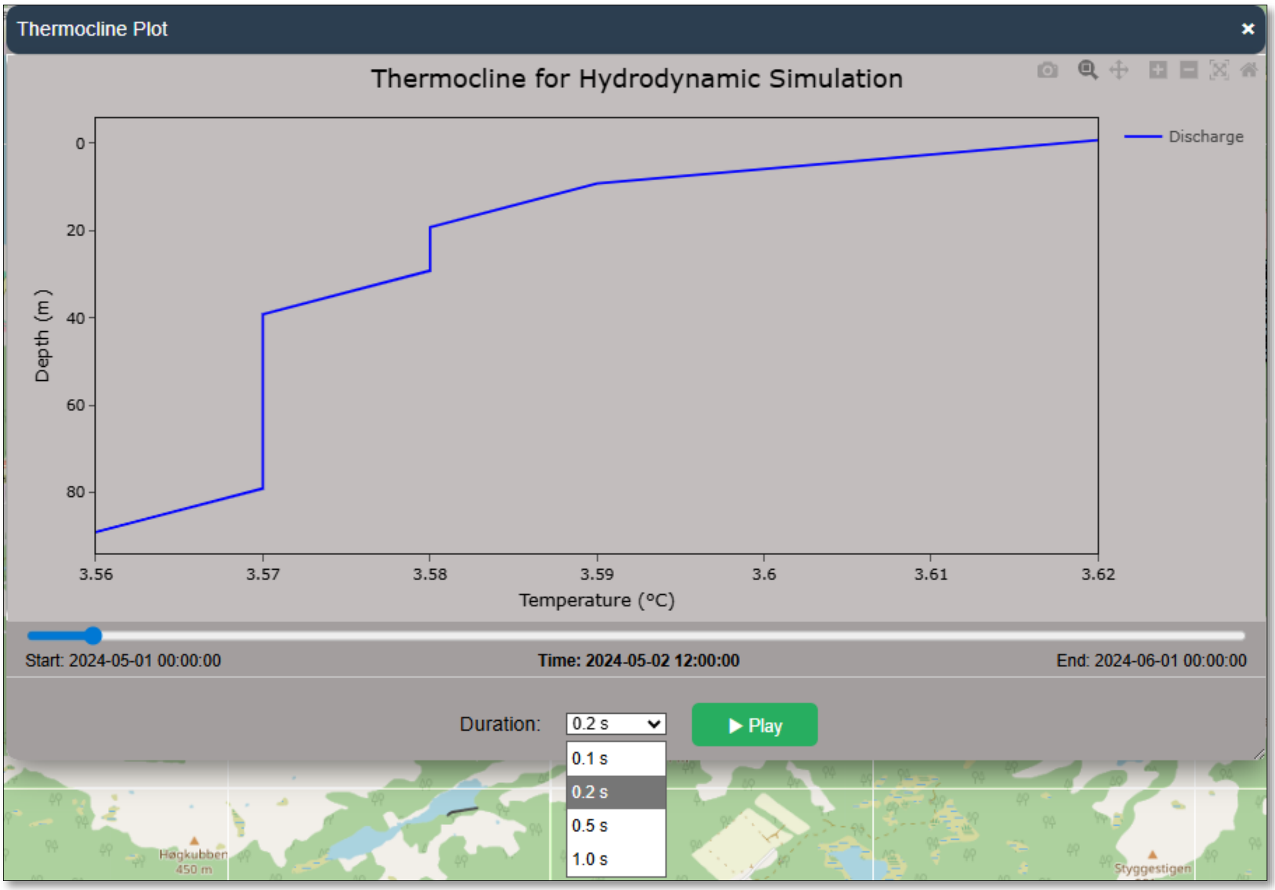


Figure 4.11. Thermocline chart for the HYD simulation

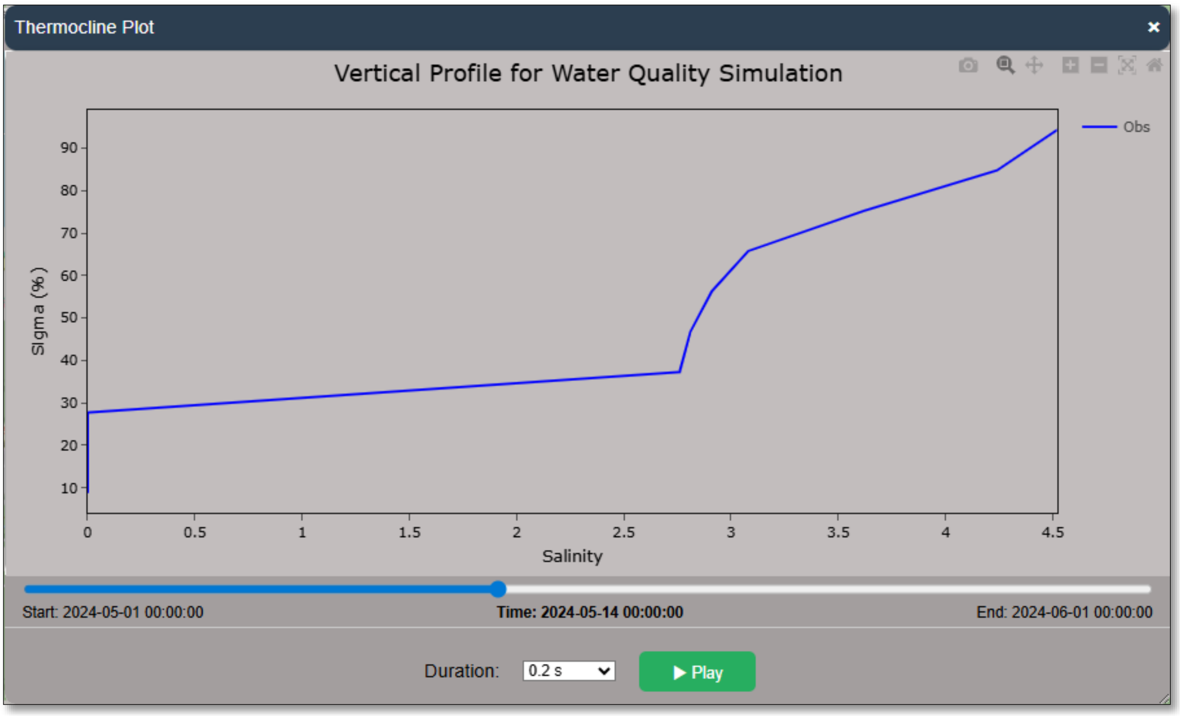


Figure 4.12. Thermocline chart for the WAQ simulation

The user can delete the thermocline chart by using the "***Clear Thermocline Grid***" on this submenu (**Figure 4.8**).

4.2. Time-Series Measurements

Figure 4.13. Time-series Measurement option

This option enables the user to view all time-series-based outputs generated from HYD and WAQ simulations. There are two parts in this submenu: the first part contains options regarding the HYD simulation, and the second part is for the WAQ simulation (**Figure 4.13**). If the user does not import the WAQ simulation while opening a project, the second part will be hidden. An example of a time-series chart for the HYD simulations is shown in **Figure 4.14**.

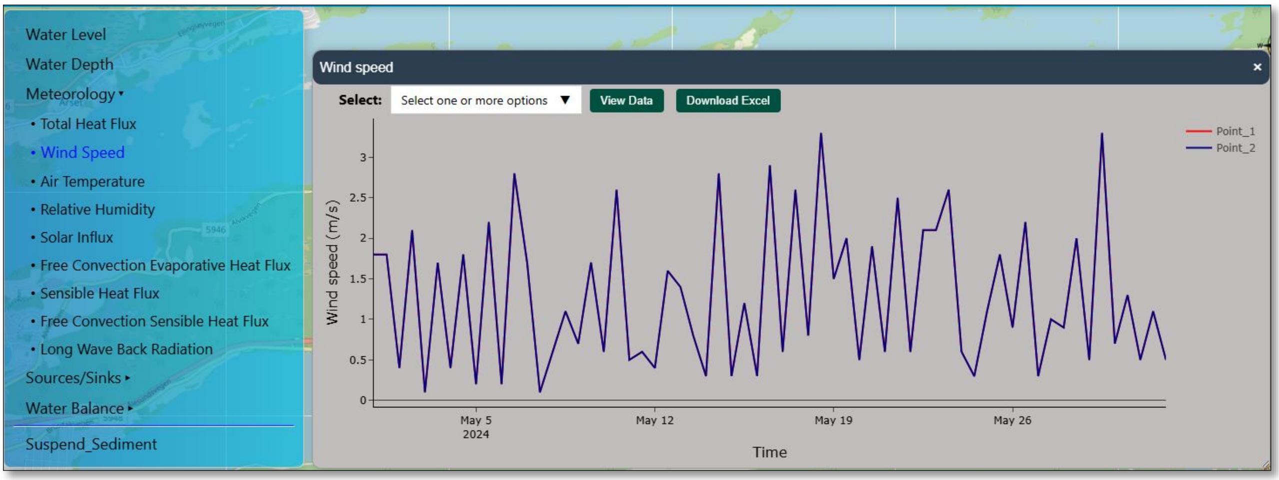


Figure 4.14. Time-series chart for a HYD simulation

When a WAQ simulation is imported, the user has a small window that allows them to toggle between available substances (**Figure 4.15**).

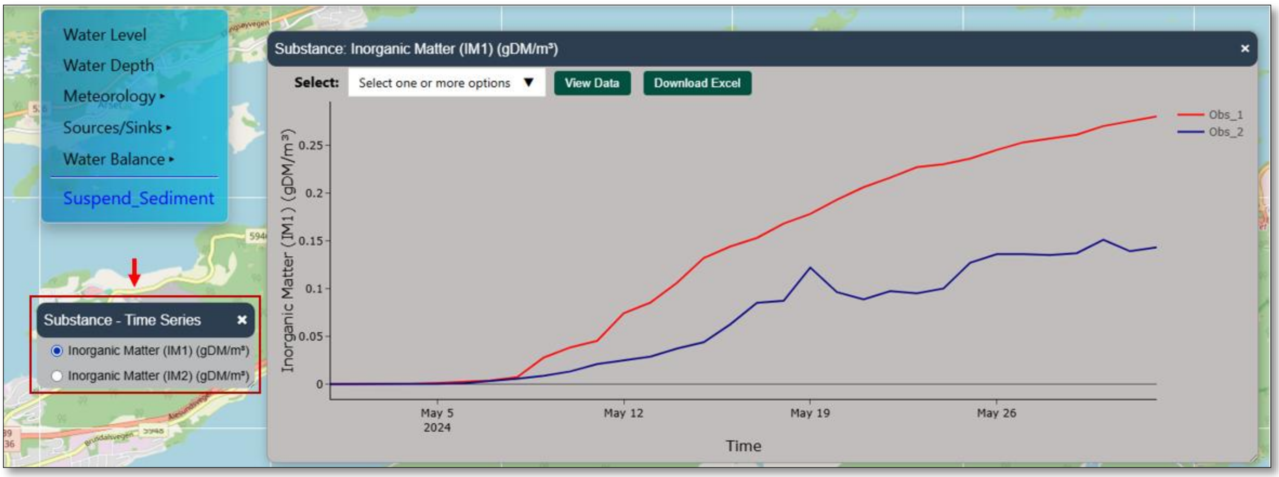


Figure 4.15. Time-series chart for a WAQ simulation

A time-series chart allows the user to view simulated data in text format or download it as an Excel format by clicking on the "***View Data***" and "***Download Excel***" buttons. When the user clicks on the button "***View Data***", a new tab opens showing the simulation time-series data (**Figure 4.16**). To save this data in Excel format, click on the "***Download Excel***" button and then choose a place to store the output.

```
Time,Obs_1,Obs_2
2024-05-01 00:00:00,0,0
2024-05-02 00:00:00,1.4200e-5,2.6000e-6
2024-05-03 00:00:00,1.2800e-4,2.9000e-5
2024-05-04 00:00:00,4.5100e-4,1.0300e-4
2024-05-05 00:00:00,0.0012,3.2400e-4
2024-05-06 00:00:00,0.0022,8.9400e-4
2024-05-07 00:00:00,0.0037,0.0034
2024-05-08 00:00:00,0.0073,0.0056
2024-05-09 00:00:00,0.0278,0.0088
2024-05-10 00:00:00,0.0385,0.0133
2024-05-11 00:00:00,0.0452,0.0210
2024-05-12 00:00:00,0.0742,0.0249
2024-05-13 00:00:00,0.0853,0.0288
2024-05-14 00:00:00,0.1060,0.0372
2024-05-15 00:00:00,0.1320,0.0438
2024-05-16 00:00:00,0.1440,0.0626
2024-05-17 00:00:00,0.1530,0.0850
2024-05-18 00:00:00,0.1680,0.0872
2024-05-19 00:00:00,0.1780,0.1220
2024-05-20 00:00:00,0.1930,0.0962
2024-05-21 00:00:00,0.2060,0.0887
2024-05-22 00:00:00,0.2160,0.0974
2024-05-23 00:00:00,0.2270,0.0951
2024-05-24 00:00:00,0.2300,0.1000
2024-05-25 00:00:00,0.2360,0.1270
2024-05-26 00:00:00,0.2450,0.1360
2024-05-27 00:00:00,0.2530,0.1360
2024-05-28 00:00:00,0.2570,0.1350
2024-05-29 00:00:00,0.2610,0.1370
2024-05-30 00:00:00,0.2700,0.1510
2024-05-31 00:00:00,0.2750,0.1390
2024-06-01 00:00:00,0.2800,0.1430
```

Figure 4.16. Simulated time-series data during the simulation

4.3. Map

This menu allows the user to see many types of dynamic maps, including spatial single-layer and multiple-layer maps (**Figure 4.17**).

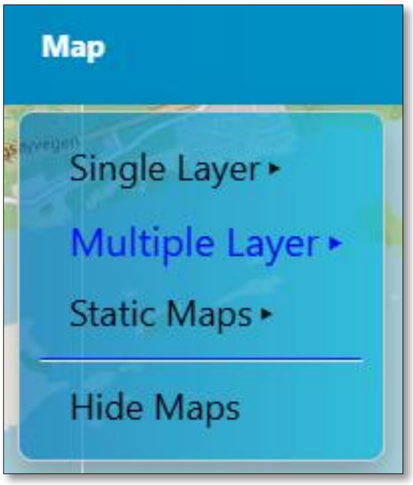


Figure 4.17. Map option

❖ *Single Layer*: This option contains some type of dynamic single-layer maps, including Water

Level, Water Depth (for a HYD simulation), and another layer map for a WAQ simulation (this option will be hidden if the user does not import a WAQ simulation while opening the project) (**Figure 4.18**).

Figure 4.18. Single Layer option

An example of a dynamic map for a single-layer map of water depth is shown in **Figure 4.19**. The user can click the button "***Play***" to see the animation and change the animation speed by specify value in the field "***Speed***".

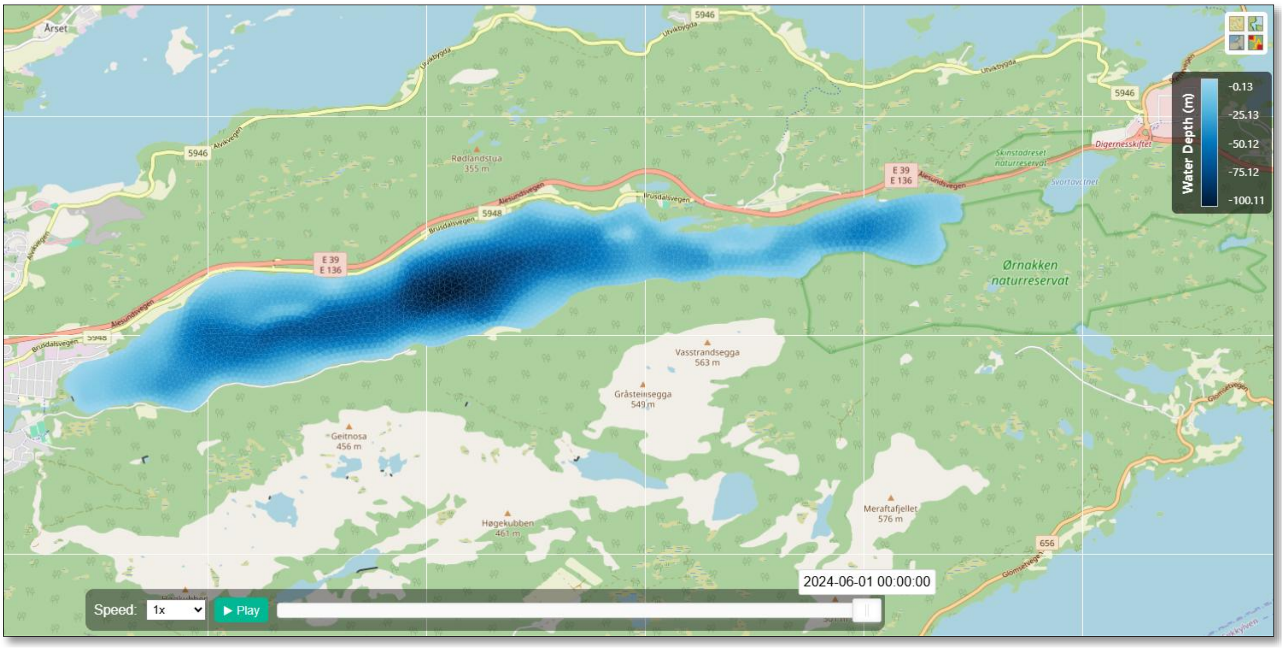


Figure 4.19. Dynamic water depth map

For a WAQ dynamic single-layer map, the user can find a small window that allows them to toggle which substance is shown on the map (**Figure 4.20**).

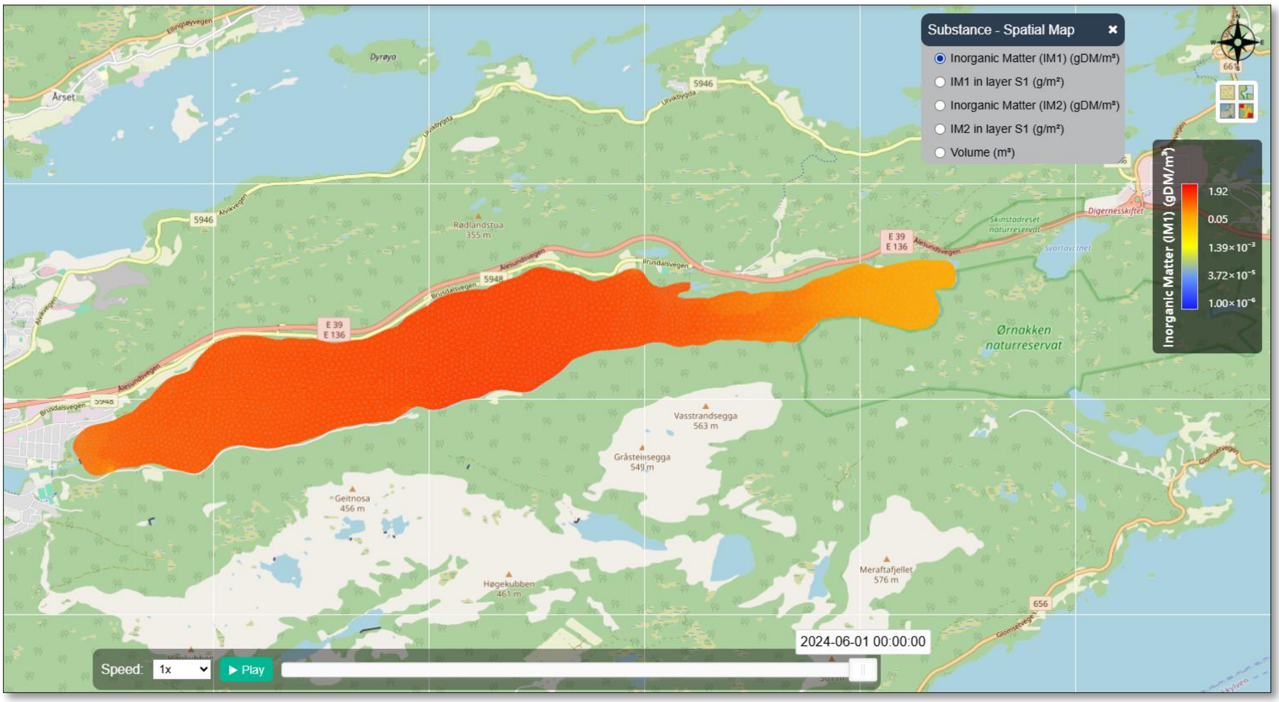


Figure 4.20. Dynamic single-layer map for WAQ simulation

❖ *Multiple Layer*: This option allows the user to visualize objects in many layers (**Figure 4.21**). There are two parts shown in this submenu: the first part (with the field "***Depth Layer***") is for a HYD simulation, and the second part (with the field "***Sigma Layer***") is for a WAQ simulation.

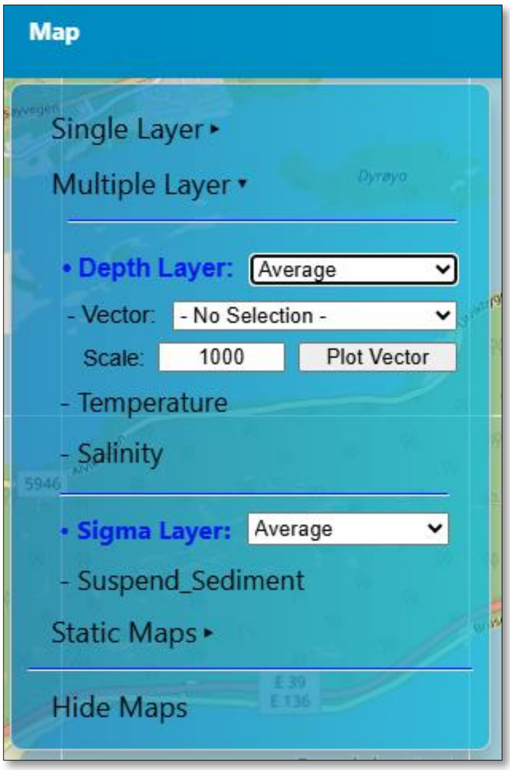


Figure 4.21. Multiple Layer option

The user can decide which layer to visualize by selecting it from a dropdown list in the field "***Depth Layer***" (**Figure 4.22**).

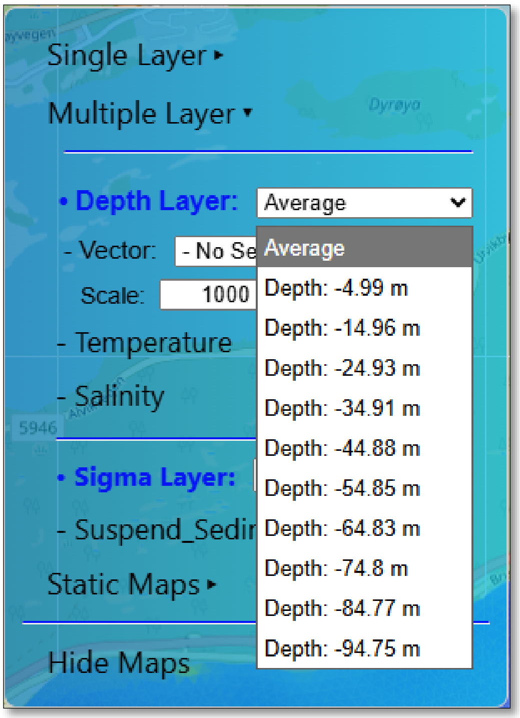


Figure 4.22. Select the layer for the object

The user can visualize the average vector (such as velocity) on the map by selecting "***Velocity***" from the dropdown list in the field "***Vector***" (**Figure 4.23**), change the scale in the field "***Scale***", and then click on the "***Plot Vector***" button to show the vector on the map.

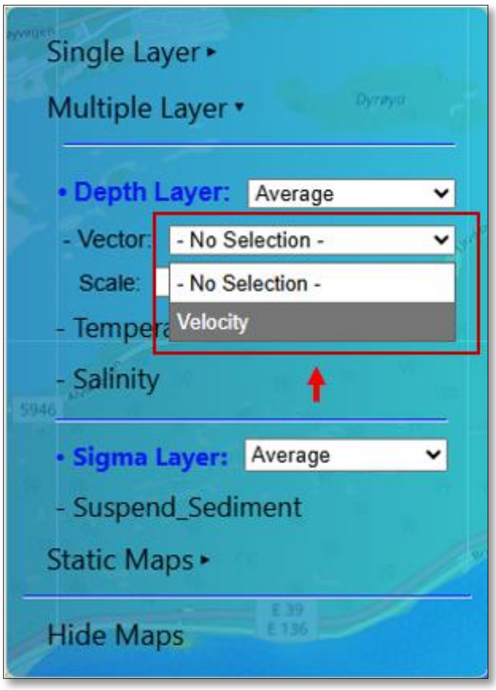


Figure 4.23. Set up a vector map

An example of a vector map for velocity during a HYD simulation is shown in **Figure 4.24**. To plot a vector map for a specific layer, change the value in the dropdown list in **Figure 4.22**.

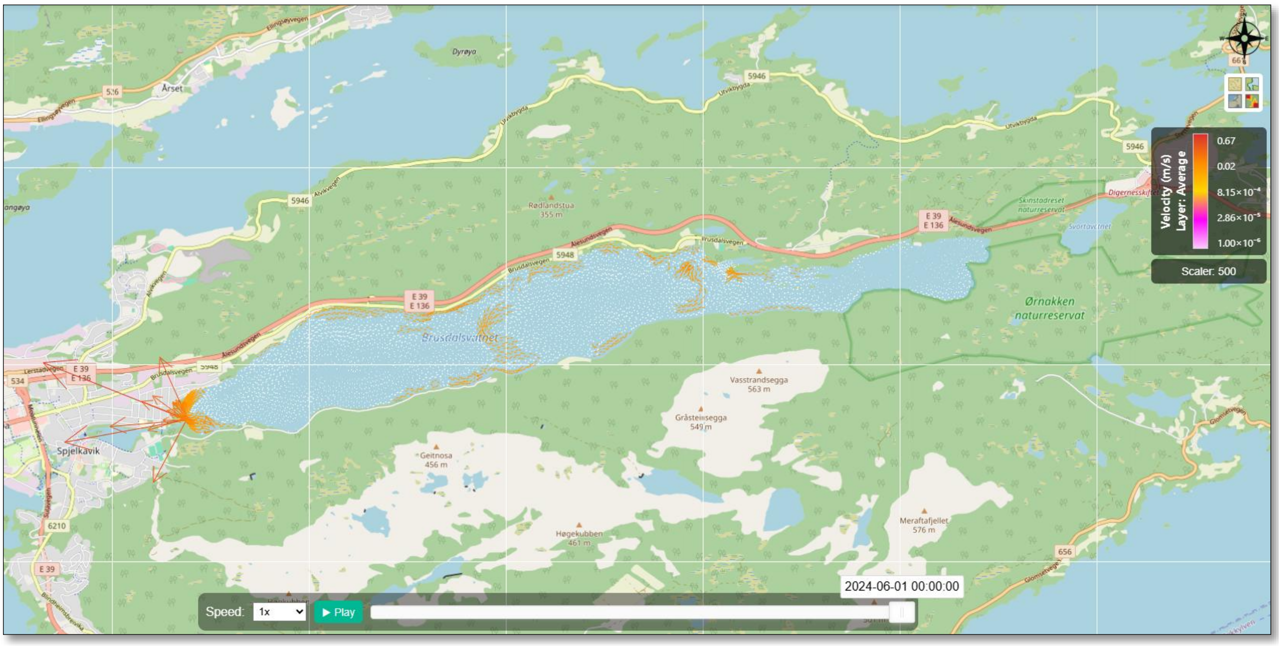


Figure 4.24. Plot vector map

If the user wants to independently visualize another map (for example, temperature or salinity), change the value in the dropdown list in the field "***Vector***" to "- *No selection* -", then click on the temperature or salinity option on the menu to visualize it on the map. An example of a temperature during a HYD simulation is shown in **Figure 4.25**.

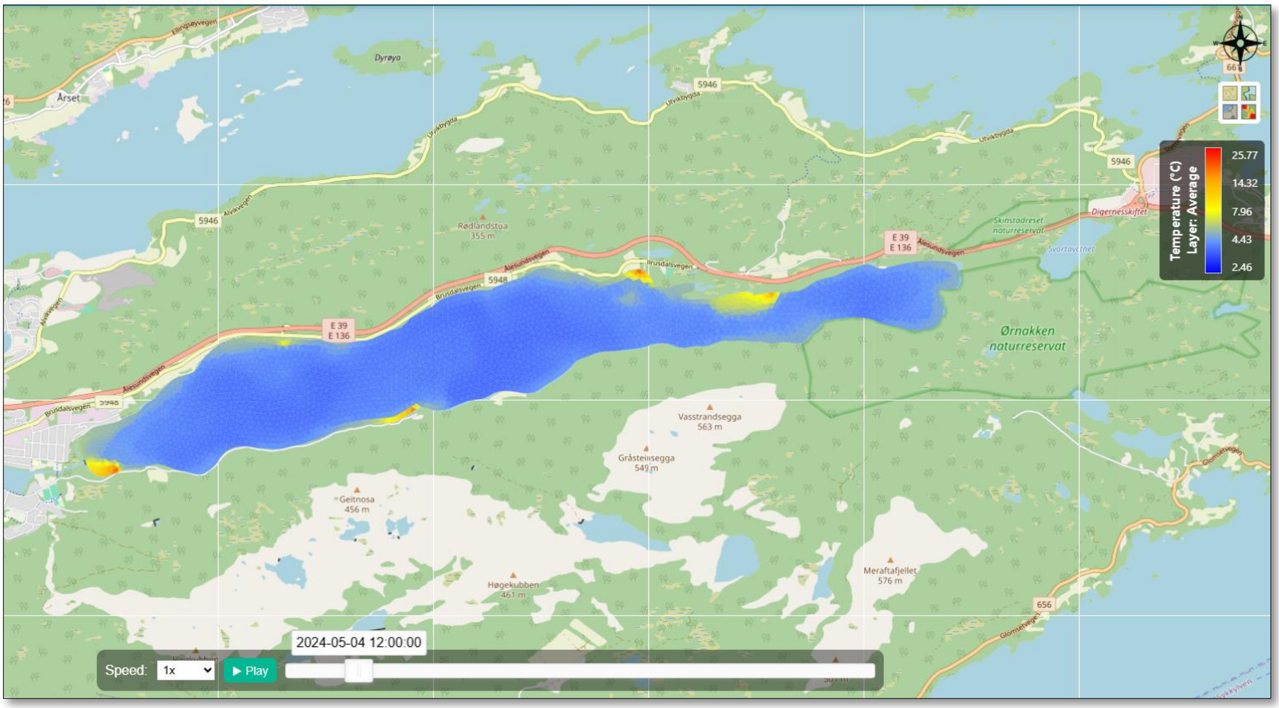


Figure 4.25. Multiple-layer dynamic map for average temperature

In case the user wants to see both velocity and temperature on the same map, select a specific layer from the dropdown list in the field "***Depth Layer***", change the value in the dropdown list in the field "***Vector***" to "*Velocity*", then click on the temperature option on the menu to visualize it on the map. An example of a velocity and temperature map during a HYD simulation at a depth of 5m is shown in **Figure 4.26**.

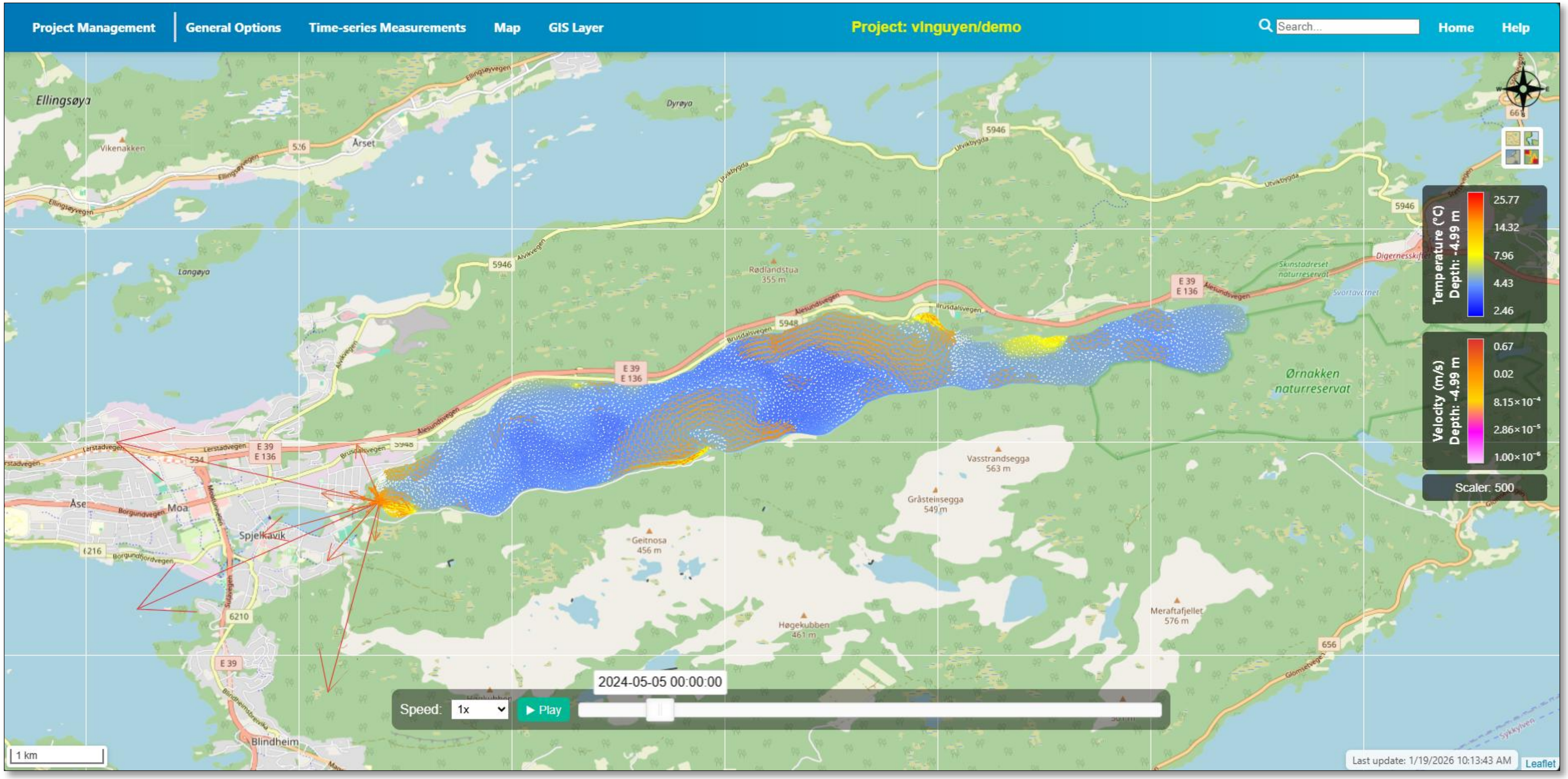


Figure 4.26. Velocity and temperature maps in a specific depth layer

If the user selects a WAQ simulation while opening the project, the second part in **Figure 4.21** will appear. After the user selects a specific sigma layer for a WAQ simulation, select the object to visualize on the map. The user can change the type of substance from an additional window (**Figure 4.27**).

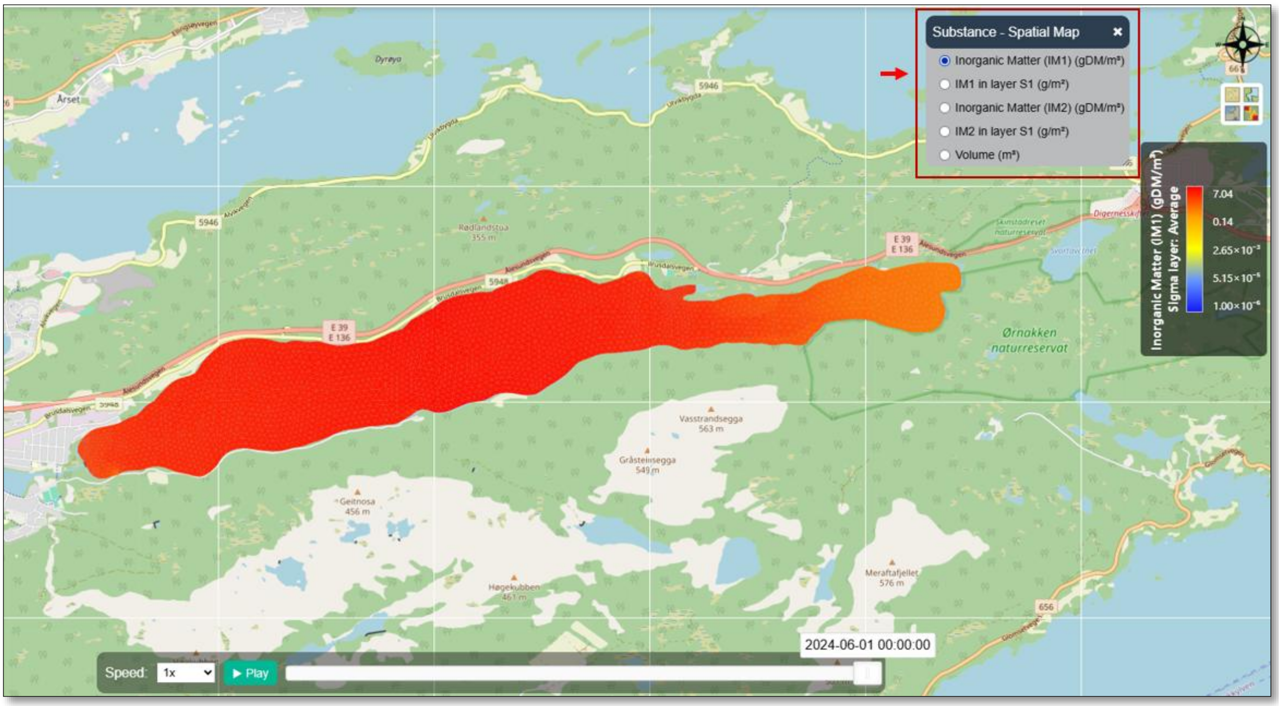


Figure 4.27. A dynamic map for the average suspended sediment scenario

❖ *Static Map*: This option allows the user to visualize a static map. One example of a bed depth map is shown in **Figure 4.28**.

Figure 4.28. A bed depth map

5. Uploading GIS data

The platform allows the user to upload and view GIS-based data. The supported formats are GeoJSON (.geojson) and shapefiles (.shp). The user can upload a GeoJSON file directly inside the platform. However, a .shp file and its associated files must be grouped in a folder and compressed as a ZIP file (.zip). *The user can add many .shp files (and their associated files) into one single folder*. To upload GIS data, the user clicks on "***Project Management***", then selects "***Utilities/GIS Uploader***" (**Figure 5.1**). A new window appears, allowing the user to select GeoJSON or a ZIP file.

Figure 5.1. Upload GIS data

After the user imports GIS data successfully, there is a new item that appears on the menu that contains imported files. The user can show or hide GIS files on the map by checking or unchecking the corresponding files in the list (**Figure 5.2**).

Figure 5.2. GIS menu

An example of GIS files on the map is shown in **Figure 5.3**. The user can delete the GIS files from the list (and the project) by clicking on the "***Delete***" button beside.

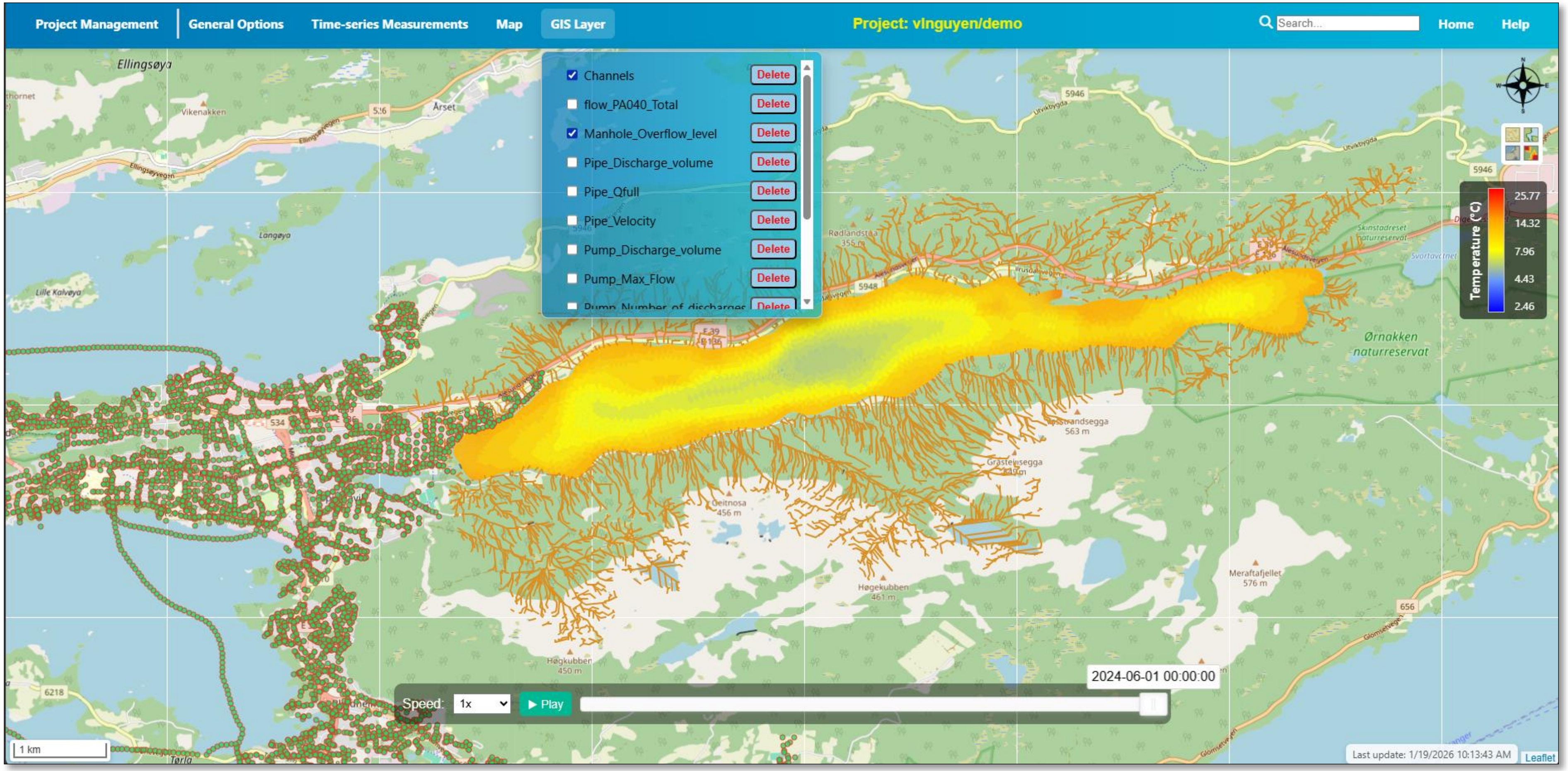


Figure 5.3. GIS layers in the map

When the user clicks on the object on the map, they can see the attributes of the selected object (**Figure 5.4**).

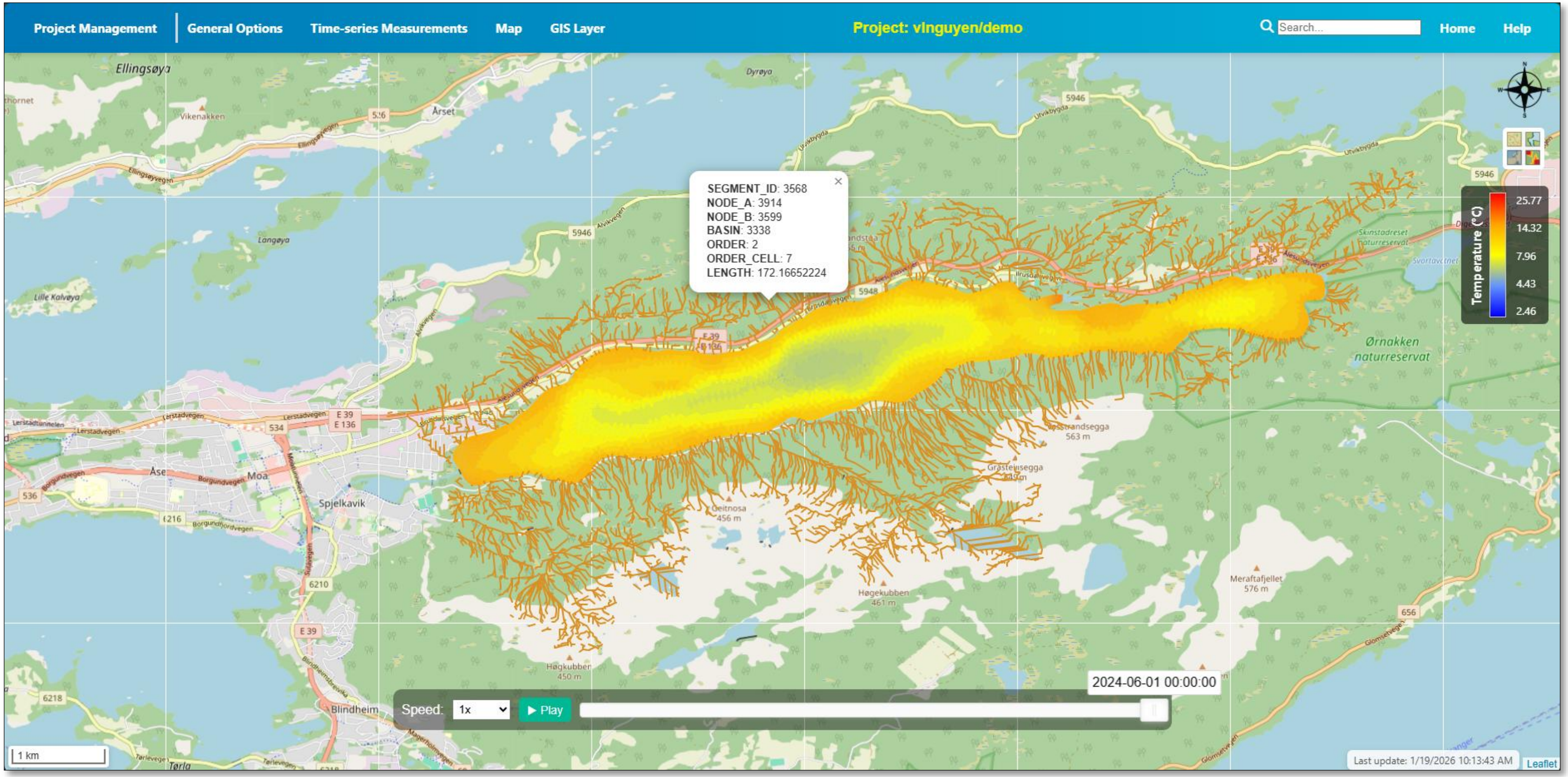


Figure 5.4. View object's attributes

6. References

Manuals, https://oss.deltares.nl/web/delft3dfm/downloads

D-Flow FM User Manual, https://content.oss.deltares.nl/delft3d/D-Flow_FM_User_Manual.pdf

RGFGRID User Manual, https://content.oss.deltares.nl/delft3d4/RGFGRID_User_Manual.pdf