MIKE URBAN+

User Guide

2D Overland

PLEASE NOTE

CONTENTS

1 General Settings

MIKE URBAN+ is a flexible system for modelling and design not only of water distribution and wastewater and storm water collection systems but also of 2D overland and integrated network-surface modelling for flood analysis.

The 2D Overland module in MU+ allows for:

- 2D modelling of free-surface flows and pollution transport
- Coupled 1D network and 2D overland system modelling for integrated surface flow and pollution transport analysis

2D Overland modelling capabilities are made available in MIKE URBAN+ in relation to Collection System models.

Access the Modules and Description editors for 2D overland modelling under the General Settings section for Collection System models.

1.1 Modules

Define the computation modules needed for a MIKE URBAN+ project in the Modules editor (Figure 1.1). MIKE URBAN+ allows coupling the 2D overland model to a collection system network and/or a river network, where the river network is either defined within MIKE URBAN+ or in an external MIKE HYDRO River model setup file.

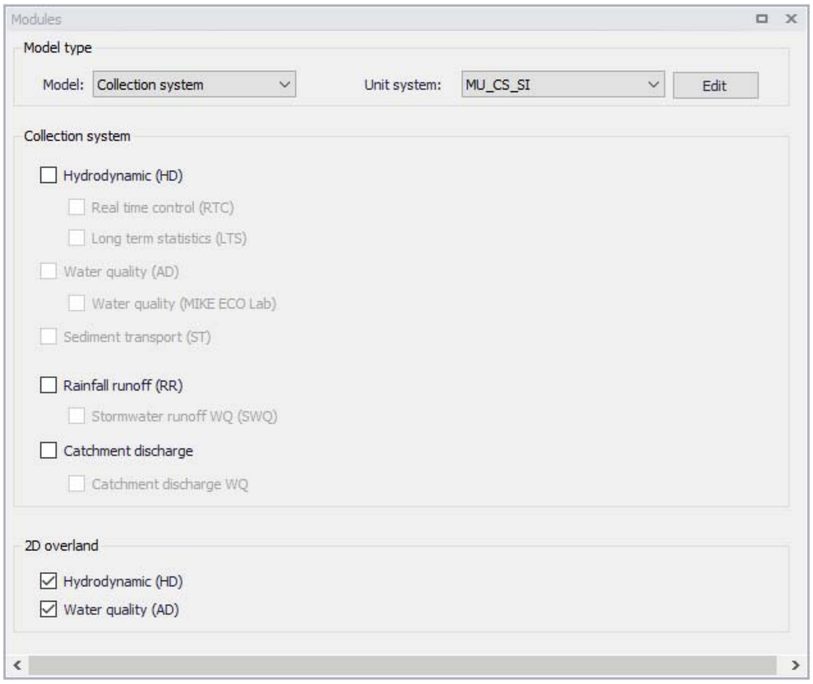


Figure 1.1 The Modules Editor showing 2D Overland analysis options at the bottom

When 'Coupling to MIKE HYDRO River' is active, the desired MIKE HYDRO model setup file must be selected in the 'River model' page. Note that coupling AD module with MIKE HYDRO is allowed and automatically enabled when this module is also active for 2D overland and/or collection system. AD components names in MIKE HYDRO must, however, be identical to the components names in MIKE URBAN+.

The editor provides an ‘at a glance’ view of the MU+ CS analysis modules active for a project. Activate options under the 2D Overland module if 2D surface flow (HD) and pollution transport modelling capabilities are needed in the MU+ project.

Activating both the Collection System HD and 2D Overland HD options offers options for coupling 1D network and 2D overland systems for integrated (1D/2D) modelling.

The 2D Overland modules include:

- Hydrodynamic (HD)
- Water Quality (AD)

The Collection System modules include:

- Hydrodynamic (HD)

  - Real Time Control (RTC)
  - Long Term Statistics (LTS)
- Water Quality (AD)
  - Water Quality (MIKE ECO Lab)
- Sediment Transport (ST)
- Rainfall Runoff (RR)
  - Stormwater Runoff Water Quality (SWQ)
- Catchment Discharge
  - Catchment Discharge WQ

1.2 Description

Access the Description editor from the General Settings section. This editor allows addition of information about the project and a free text description of the model (Figure 1.2).

It may also be used as a model build log to make notes on updates and amendments.

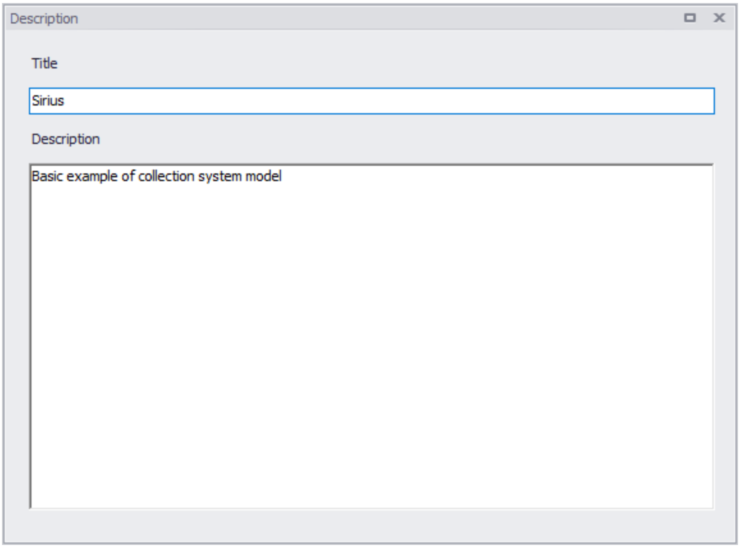


Figure 1.2 The Description Editor

2 Map Configuration

The Map Configuration section contains information on the coordinate system used in the MIKE URBAN+ project and presents options for customising the map background image.

2.1 Coordinate System

The Coordinate System dialog (Figure 2.1) displays the Projection system used in the project.

Figure 2.1 The Coordinate System dialog showing information on the projection system used in the project

The Projection system shown matches the specified Projection in the 'New Module Setup' window when the MIKE URBAN+ project was created (Figure 2.2).

It is currently not possible to modify the projection system for a project in MIKE URBAN+.

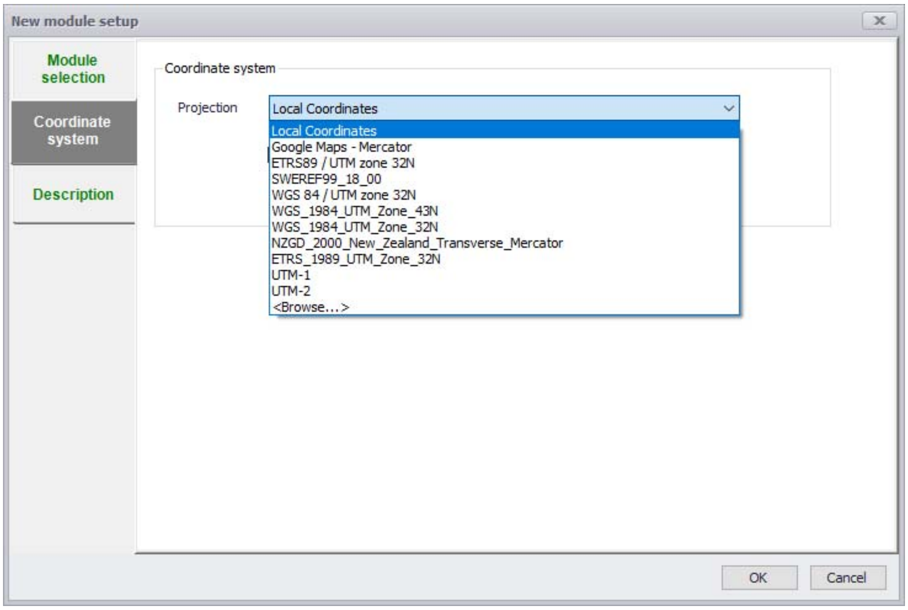


Figure 2.2 Specify the projection system for a new MU+ project on the New Module Setup window.

2.2 Background Map

The Background Map editor allows the user to select a background image to show in the Map View in MIKE URBAN+ (Figure 2.3).

Activate a background map overlay by ticking the 'Visible' checkbox on the editor.

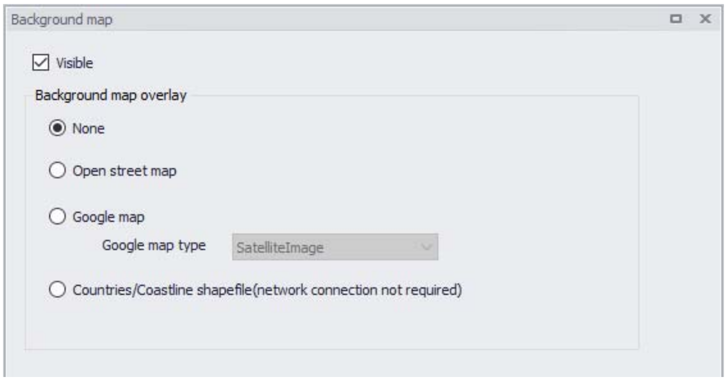


Figure 2.3 The Background Map Editor

The following background map overlay options are available:

- **None**
- **Open Street Map**
- **Google Map**. Select the Google map type to display (i.e. Street map, Satellite image, Terrain, or Hybrid).
- **Countries/Coastline Shapefile**. Polygon features that demarcate coastline or country boundaries.

An internet connection must be available for Open Street Map and Google Map overlays (Figure 2.4).

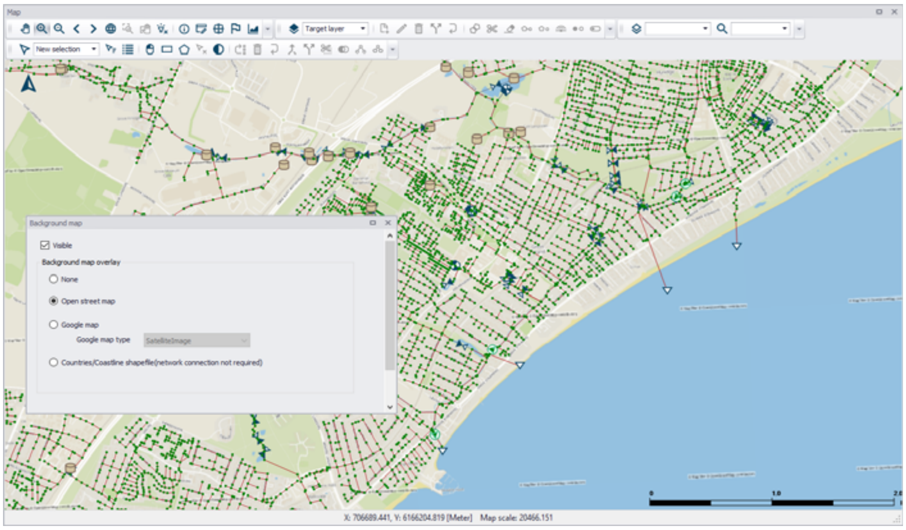


Figure 2.4 An example Open Street Map background on the Map View in MU+

3 2D Overland Hydraulic Modelling

The 2D Overland module in MIKE URBAN+ is used for modelling 2D free-surface flows. It allows simulation of hydraulic phenomena on land surfaces, overland water bodies, as well as coastal areas, and may be applied wherever flow stratification can be neglected.

The MIKE URBAN+ 2D Overland Flow module may be applied to overland flow phenomena including:

- Surface flows
- Overland flooding in pluvial, fluvial, and coastal environments

The 2D Overland Hydrodynamic (HD) module simulates water level variations and flows in response to a variety of forcing functions. It provides the basis for water quality (AD) computations also offered under the module.

Activating the Hydrodynamic (HD) module provides access to 2D Overland model editors in the Setup tree view (Figure 3.1).

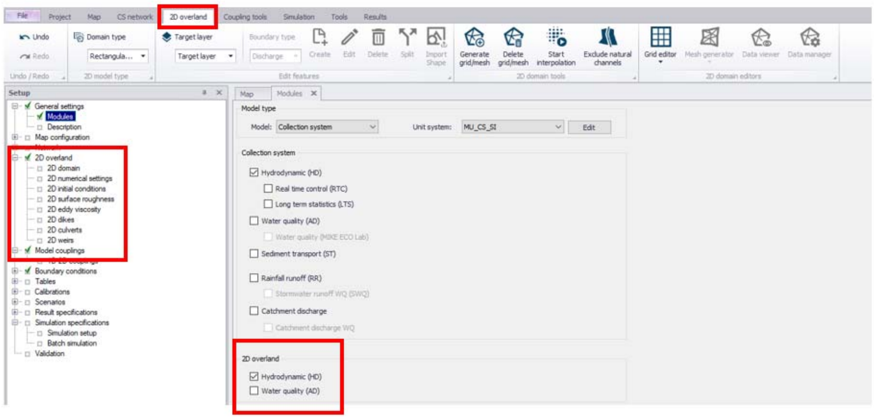


Figure 3.1 Activate the Hydrodynamic (HD) 2D Overland module to access editors for building 2D Overland models in the MIKE URBAN+)Setup Tree View.

The various 2D Overland model editors include:

- 2D Domain. For defining or configuring the 2D computational grid or mesh.
- 2D Numerical Settings. For defining 2D model numerical solution settings.
- 2D Initial Conditions
- 2D Surface Roughness

- 2D Eddy Viscosity
- 2D Dikes
- 2D Culverts
- 2D Weirs

These editors are described further in subsequent sections.

3.1 2D Domain

The 2D Domain editor is where the computational grid or mesh for the 2D model is configured or defined.

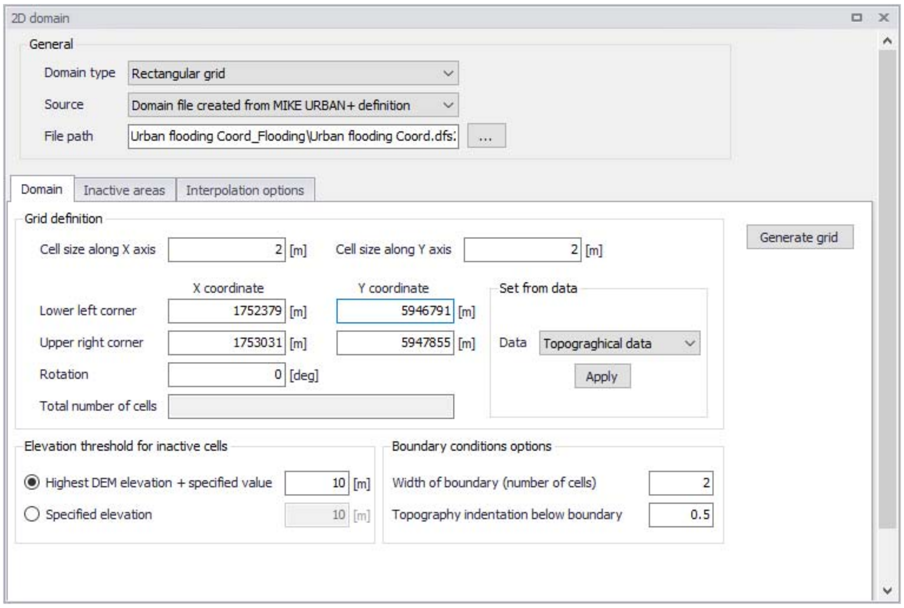


Figure 3.2 The 2D Domain editor

3.1.1 General

Define general properties for the 2D model computational grid under the General group box at the top of the editor.

Domain Type

Select the type of 2D computation grid to use in the project from the drop-down menu. In MIKE URBAN+, computational grids may be:

- **Rectangular grid**. Made up of orthogonal, uniformly-spaced grid points representing the model domain. Related to MIKE *.DFS2 file types.

- **Flexible mesh**. Normally made up of non-uniform triangular or quadrangular elements representing the model domain. Related to MIKE *.MESH file types.

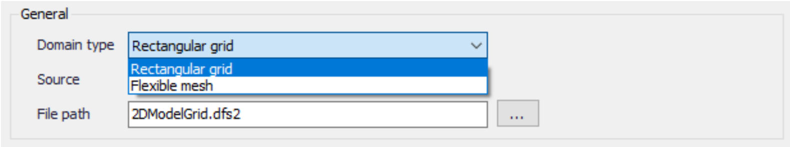


Figure 3.3 2D domain type options

Source

The 2D computational grid may be may be directly specified in the editor with an existing file or may be configured and generated from MIKE URBAN+. The Source dropdown menu offers options for using a:

- Domain file created from MIKE URBAN+ definition
  With this option, it is required to specify the extend of the 2D domain and its resolution, and finally interpolate topographical data onto the computational points.
- Existing domain file
  With this option, an input 2D domain file must be supplied and will be used as is.

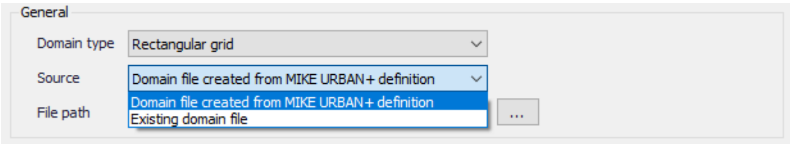


Figure 3.4 2D domain source options

File Path

Depending on the Source and Domain Type specified, the File Path will refer to either the:

- **Path and file name for the domain file to be generated from MIKE URBAN+.** This is relevant for when the Source is ‘Domain file created from MIKE URBAN+ definition’.
- **Path to the existing domain file**. This is relevant for when Source is ‘Existing domain file’.

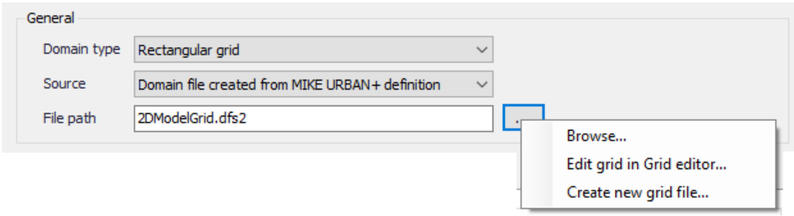


Figure 3.5 File path options

The ellipsis button (...) offers the following functions:

- **Browse**. Launches the file explorer for navigating to the desired location of the domain file to be created or the location of the existing domain file, when using the ‘Rectangular grid’ domain type.
- **Browse mesh**. Launches the file explorer for navigating to the desired location of the domain file to be created or the location of the existing domain file, when using the 'Flexible mesh' domain type.
- **Browse dfsu**. Launches the file explorer for navigating to the desired location of the existing domain file, when using the 'Flexible mesh' domain type and when the 2D domain should preferably be defined with element-centered topographical values. Only the first item from the selected dfsu file can be used.
- **Edit grid in Grid Editor**. Launches the Grid Editor to edit existing or previously-created rectangular grids.
- **Edit in Mesh Generator**. Launches the Mesh Generator to examine/edit existing or previously-created *.MESH files.
- **Open in Data Viewer**. Launches the Data Viewer to examine/edit existing or previously-created *.MESH files.
- **Create new grid file**. Launches the Grid Editor to create a new 2D rectangular *.DFS2 grid.
- **Crete new using Mesh Generator**. Launches the Mesh Generator to create a new flexible mesh file.

3.1.2 Domain

When creating new computational grids from MIKE URBAN+, the Domain tab is where basic properties for grid/mesh generation are defined.

Tab contents in the editor change depending on whether a rectangular grid or a flexible mesh shall be created.

Rectangular Grid

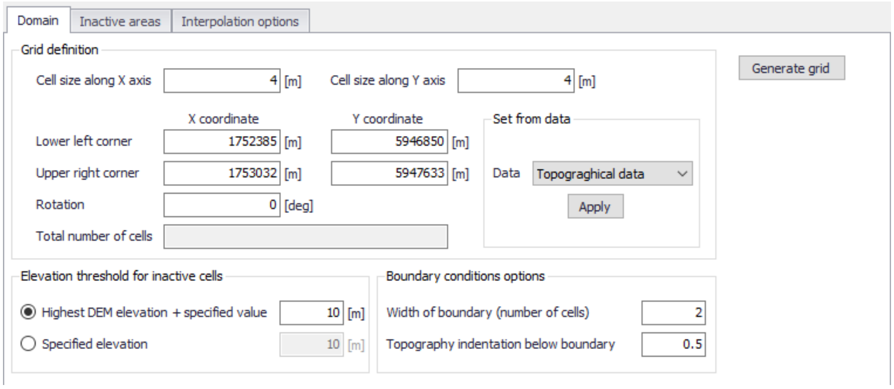


Figure 3.6 The Domain tab for configuring new rectangular grids in MIKE URBAN+

- **Grid definition**: Define the grid cell size along the x- and y-directions, and the extent of the 2D model domain. The extent may also be derived from available data layers, such as:
  – **Topographical data**. Based on the data layer(s) to be used in 2D topography interpolation. The data layers must be activated in the Interpolation Options tab to be properly detected as ‘Topographical Data’.
  – **Collection system network**. The extent of the whole collection system model.
  – **Active selection**. The maximum extent of currently-selected model elements.

- **Elevation threshold for inactive cells**: Define the elevation value over areas that should remain inactive during 2D model computations. Inactive grid cells are excluded from computations.
  – Highest DEM elevation + specified value. The specified value is added to the maximum elevation value detected from the 2D domain file.
  – Specified elevation. Areas with terrain levels equal to or higher than the defined value are considered inactive.

- **Boundary conditions options:** Specify the width and grid cell levels at 2D boundary sections. 2D boundaries (i.e. closed and open boundaries) are located along edges of the 2D domain in MIKE URBAN+.

  – **Width of boundary (number of cells)**. Define the width (in the direction transversal to the border of the domain) of 2D boundary edges in terms of number of grid cells. Grid size is defined under the ‘Grid definition’ section. This width is used for the indentation described below.

– **Topography indentation below boundary**. The depth subtracted from the lowest cell level adjacent to the boundary. The resulting value is the adjusted level assigned to the open 2D boundary area (Figure 3.7). This is used to apply a constant topography along the boundary and ensure that the boundary is e.g. always flooded.

Along the 2D model perimeter where no 2D boundary condition is applied, the ground elevation is automatically raised to the elevation threshold for inactive cells. This is to represent a closed boundary ensuring zero flow across the perimeter (also see the Raised areas in Figure 3.7).

Along stretches where an open water level or discharge boundary is defined, the ground is lowered to a uniform level to ensure valid numerical computations at the boundary as open boundaries must be wet all the time (Figure 3.7). The lowering (i.e. 'indentation') is 0.5 m by default applied to the rows/columns parallel to the 2D boundary alignment. All 2D cells are lowered to the same level based on the lowest cell level in the first non-adjusted row/column (e.g. by default the 2nd row from the edge) along the 2D boundary (Figure 3.7).

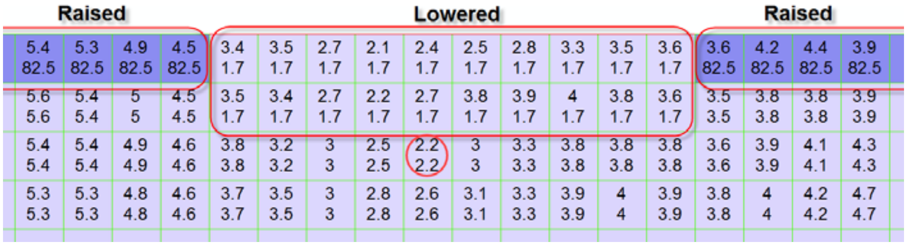


Figure 3.7 Example of automatic terrain level lowering along an open 2D boundary spanning 10 2D cells. In each 2D cell the original levels (upper values) and adjusted levels (lower values) are shown. The value of 1.7 m is taken from the lowest-level unadjusted cell adjacent to the boundary (i.e. 2.2 m encircled) minus 0.5 m.

Flexible Mesh

For flexible meshes, the Domain tab is where mesh resolution, overall extent, and local mesh area properties may be defined.

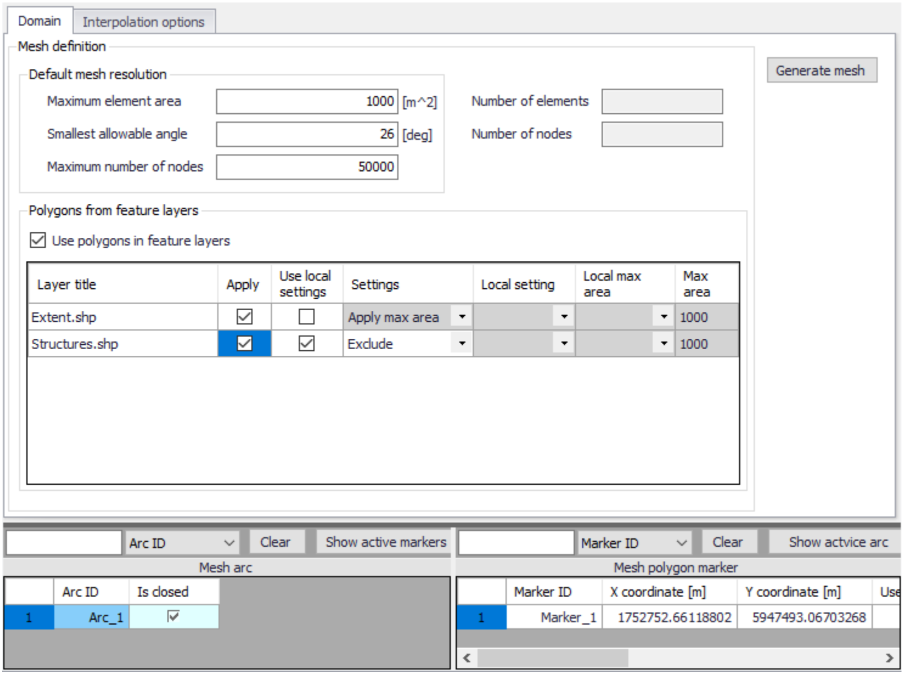


Figure 3.8 The Domain tab for configuring new flexible meshes in MIKE URBAN+

- **Default mesh resolution**. Under this section, set values for various default mesh generation criteria. These include:
  - **Maximum element area**. Area upper bound for elements in the mesh to be generated.
  - **Smallest allowable angle**. Target smallest interior angle for elements in the generated mesh.
  - **Maximum number of nodes**. Upper limit for number of element vertices in the mesh to be generated.
- **Mesh polygons**. This section is where various mesh areas are defined using polygons. This includes the overall extent of the mesh, as well as areas where local mesh settings shall be applied. These areas may be defined using:
  - **Feature layers**. Feature layers loaded to the Map prior to mesh generation. The layers must be added to the Map before use in mesh generation.

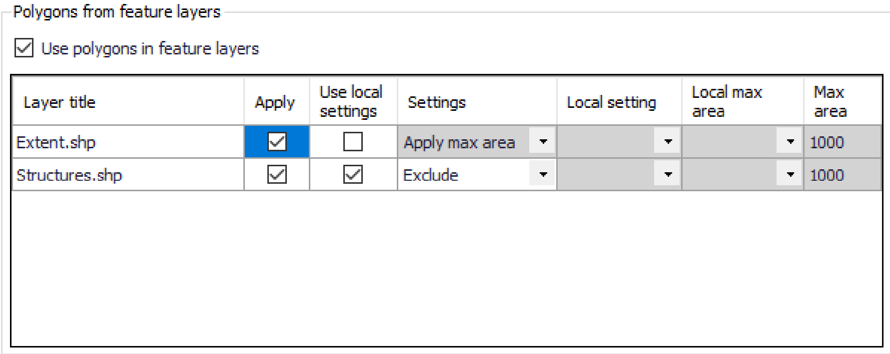


Figure 3.9 List of polygon features that may be used for mesh generation in the Domain tab page

Usable feature layers are automatically recognized by the program and included in the layer list in the editor.

Activate the 'Apply' option for layers to use and tick on the 'Use polygons in feature layers' option to use the applied layers in mesh generation.

- **Mesh arcs and polygon markers**. These are mesh arcs and markers drawn on the Map. Arcs are lines or polylines that are open or closed. Closed arcs have the same first and last nodes, to create a polygon. A polygon marker may be added within a polygon, in order to specify the local properties of the mesh in the polygon (see Figure 3.10).

  Define arcs and polygons on the Map using the Flooding Layer Editing toolbar (Figure 3.10) or the Edit Features toolbox on the 2D Overland menu ribbon (Figure 3.11). Select 'Mesh arc' as the target layer from the toolbar and use the 'Create' tool to draw arcs on the map.

  Mesh polygon markers define local mesh properties in the encapsulating polygon. A polygon may only have one polygon marker.

  Records for defined domain definition layers (i.e. arcs and polygon markers) are shown in the table at the bottom of the 2D Domain editor (Figure 3.12).

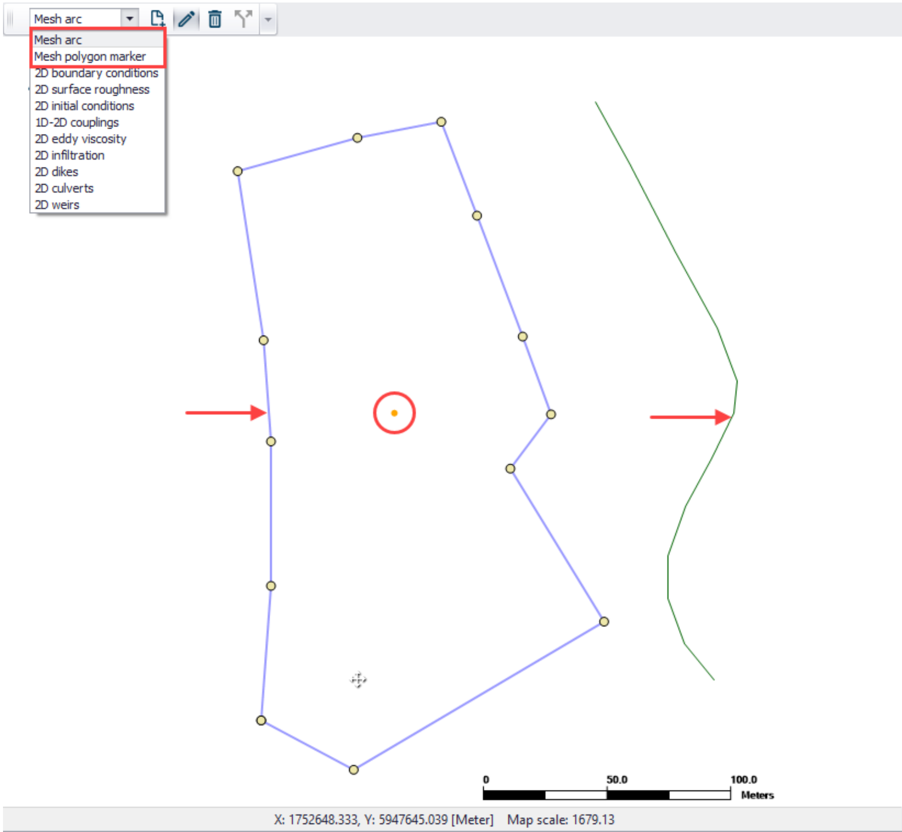


Figure 3.10 Define arcs and polygons on the Map via the Flooding Layer Editing toolbar. The figure shows a closed and open arc indicated by the arrows, and a polygon marker (encircled).

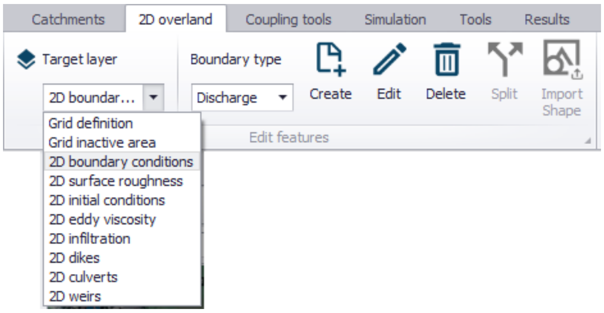


Figure 3.11 The Edit Features toolbox on the 2D Overland menu ribbon

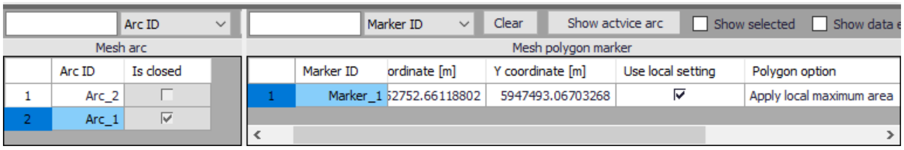


Figure 3.12 Mesh arcs and polygon markers grid table at the bottom of the 2D Domain editor

3.1.3 Inactive Areas

This tab is accessible only when generating Rectangular Grids. It is where areas for inactive cells in the computational grid (other than closed boundaries) are defined.

Inactive cell values, as configured in the Domain tab (i.e. Elevation threshold for inactive cells), are assigned to these cells. These areas may be defined with:

- **Feature Layers**. Polygon feature layers loaded to the Map prior to grid creation. The layers must be added to the Map before use.

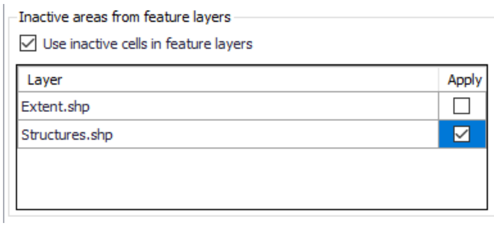


Figure 3.13 List of polygon features that may be used to define inactive cells for rectangular grids

Usable feature layers are automatically recognized by the program and included in the layer list in the editor. Activate the 'Apply' option for layers to use and tick on the 'Use inactive cells in feature layers' option to use the applied layers during grid interpolation. All cells within a polygon from the applied layers will be excluded from the computation.

- **Polygons**. Define inactive polygons on the Map using the Flooding Layer Editing toolbar (Figure 3.10) or the Edit Features toolbox on the 2D Overland menu ribbon (Figure 3.11). Select 'Grid inactive area' as the target layer from the toolbar and use the 'Create' tool to draw polygon features on the map.

  Records for defined polygons are shown in the 'Grid inactive area' secondary table at the bottom of the Inactive Areas tab page.

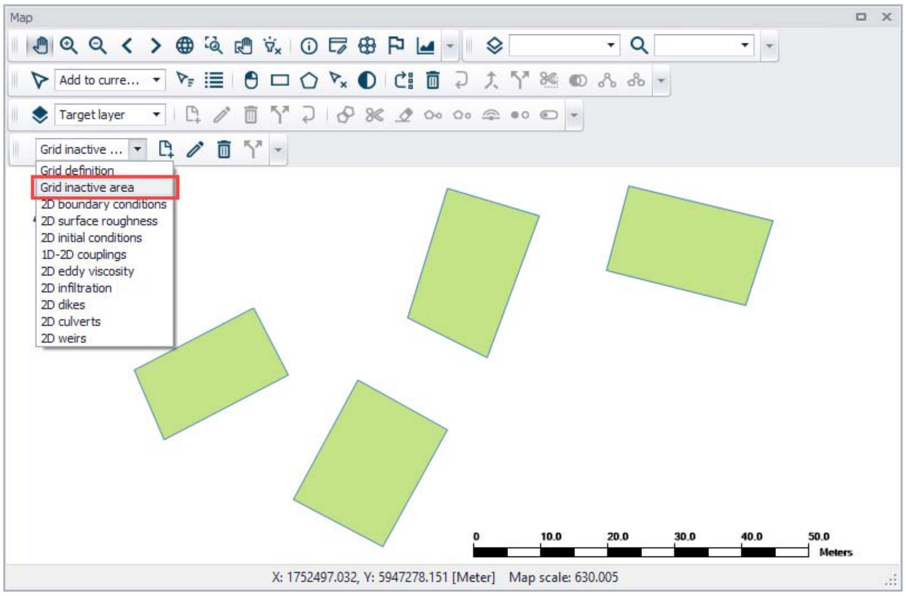


Figure 3.14 Use the Flooding Layer Editing toolbar to create Grid Inactive Area polygons on the map.

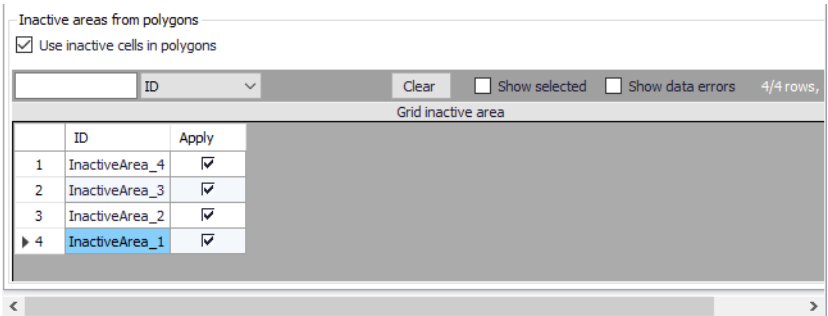


Figure 3.15 Grid inactive area secondary table at the bottom of the Inactive Areas tab page

Inactive area polygons for rectangular grids may be re-used as mesh area polygons for excluded areas during flexible mesh generation. A warning message is issued when switching from rectangular grid to flexible mesh domains after grid inactive areas have been created. Clicking on the 'Yes' button will add the previously-applied grid inactive areas to the mesh domain polygons.

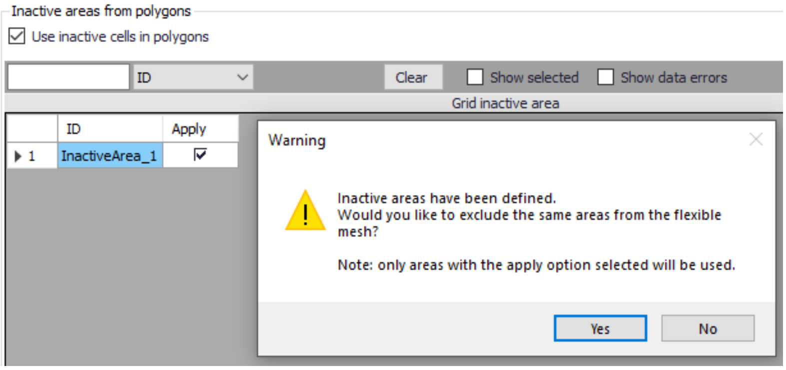


Figure 3.16 Warning message offering to option to use grid inactive areas a mesh domain polygons.

3.1.4 Interpolation Options

The Interpolation Options tab is where topographical data and methods for interpolating topographical values onto the generated computational grid or mesh are configured.

Perform interpolation only after the grid or mesh has been generated.

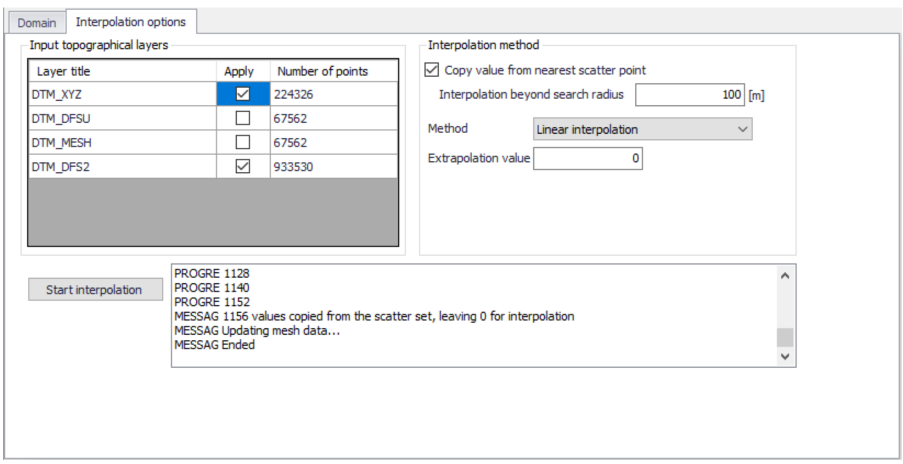


Figure 3.17 The Interpolation Options tab page

Input Topographical Layers

Input topographical layers are sources of scatter data for interpolation of values onto the grid or mesh. The following file types may be used as input topographical layers in MIKE URBAN+:

- **MZ Raster Layer *.DFS2**

- **Feature Layer *.XYZ**
- **MESH Layers *.MESH and *.DFSU**

Load these data layers into MIKE URBAN+ via the ‘Add layer’ function in the ‘Layers and Symbols’ View. These layers are added to the Map, and valid layers are added to the ‘Input topographical layers’ table in the 2D Domain editor.

Click on the ‘Apply’ option if the layer shall be used as a source of scatter data in the interpolation. The ‘Number of points’ column displays the number of scatter data points in the layer. This indicates the number of nodes for a *.MESH file, and the number of cells or elements for *.DFS2 or *.DFSU files, respectively.

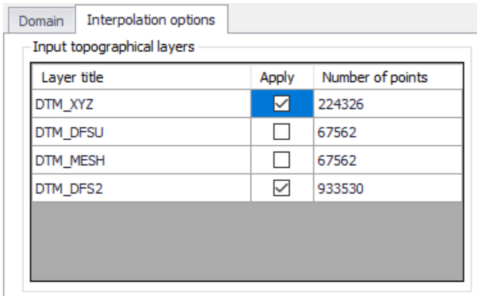


Figure 3.18   The Input Topographical Layers table on the Interpolation Options tab

Interpolation Method

This section offers available options for interpolating values onto the grid or mesh. These include:

- **Copy value from nearest scatter point.** In this option, mesh node values are simply taken from the nearest scatter point (i.e. from active input topographical layers). This option can dramatically decrease interpolation time when a very dense set of scatter data points is available. This is because values are simply set from the nearest scatter points, and true interpolation is only performed for areas where scatter points are beyond the search radius. Mesh node points assigned a value in this way will not be included in the overall interpolation.

  The search radius from a mesh node to the nearest scatter point is set via the ‘**Interpolation beyond search radius’** parameter. Set a reasonable search distance based on scatter data resolution.

- **Method.** The available interpolation methods are:
  - **Linear Interpolation**. For each mesh node to be interpolated, a surrounding triangle based on scatter points is determined, and a linear interpolation based on the scatter point values is made.

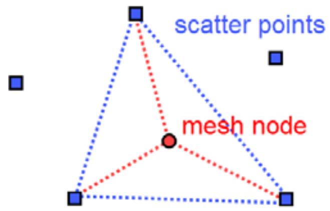


Figure 3.19 Illustration of the linear interpolation method for deriving topography values for mesh nodes from input scatter points

– **Natural neighbour interpolation**. Is a geometric estimation technique that uses natural neighbourhood regions generated around each node. The natural neighbourhood regions are determined by creating a triangulated irregular network from the scatter data points. It is particularly effective in dealing with a variety of spatial data themes exhibiting clustered or highly linear distributions.

  Natural neighbour method extrapolation beyond the extent of the scatter data may be enabled and controlled through the '**Extrapolate to ...% beyond envelope of scatter data**' parameter.

  The points will be placed at the lower left, upper left, lower right and upper right corner of the workspace area, respectively. The parameter specifies the distance from the four points to the data extent area. The water depth value at the four points will be defined as the extrapolation value.

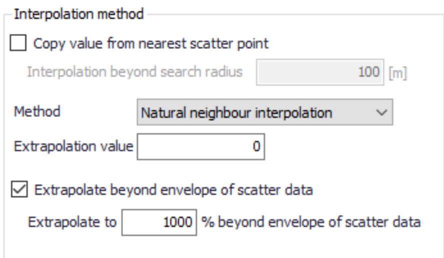


Figure 3.20 Natural neighbour interpolation parameters

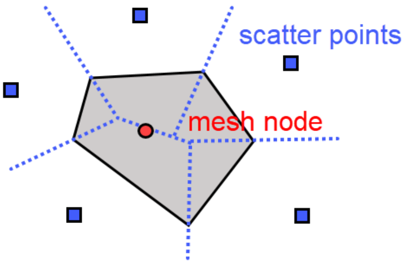


Figure 3.21 Illustration of the natural neighbour interpolation method for deriving topography values for mesh nodes from input scatter points

- **Extrapolation value**. Specify a value to be used when the program needs to extrapolate values beyond configured interpolation settings in the model.

Start Interpolation

Click on the '**Start Interpolation**' button to launch the interpolation process. The status of the interpolation process is indicated in the progress window.

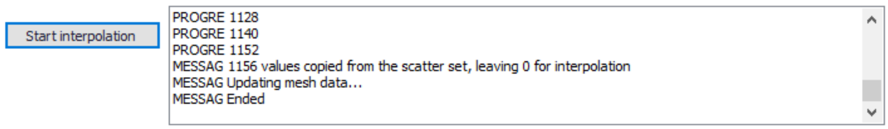


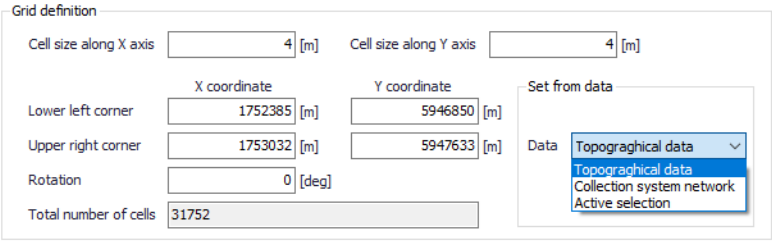
Figure 3.22 The Start Interpolation button and progress log window

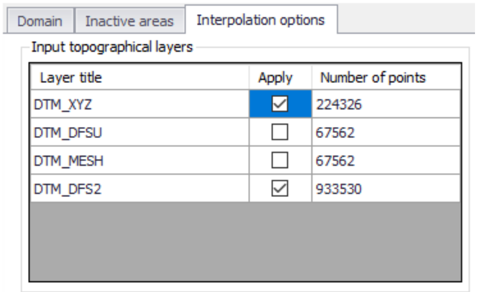
3.1.5 Creating a Rectangular Grid Workflow Example

1. Load desired input topographical layers for 2D grid interpolation into the MIKE URBAN+ project.
2. On the 2D Domain editor, choose a 'Rectangular grid' domain type and 'Domain file created from MIKE URBAN+ definition' as source. Specify the file name and path for the domain file to be created.
3. **Configure grid domain parameters**. Specify grid cell size along the x and y directions, and define the domain grid extent.

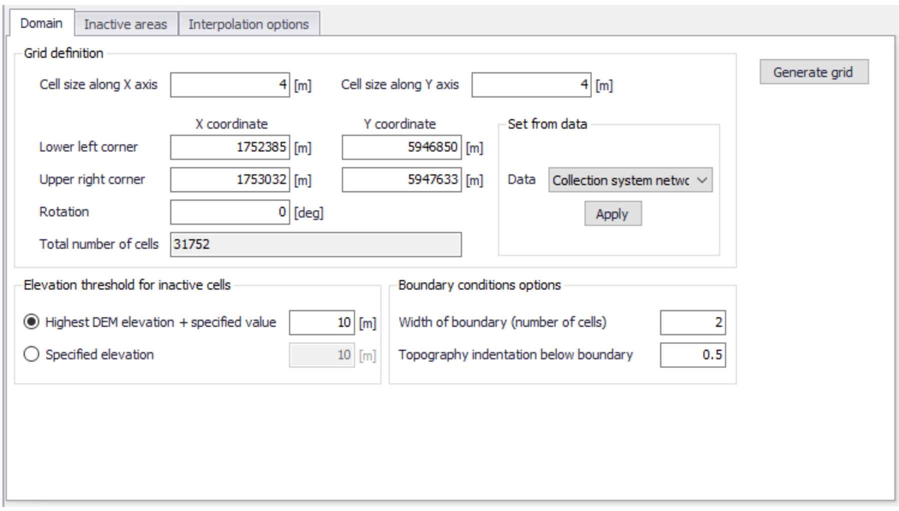
   You may choose to set the extent from Topographical data. If so, ensure that the appropriate topographical data layer is activated in the 'Interpolation Options' tab.

   You may also chose to define the grid extent by drawing it on the map. In the 2D overland ribbon, select the 'Grid definition' layer to draw a rectangle defining this extent. While editing this rectangle on the map, it is possible to resize and rotate it.

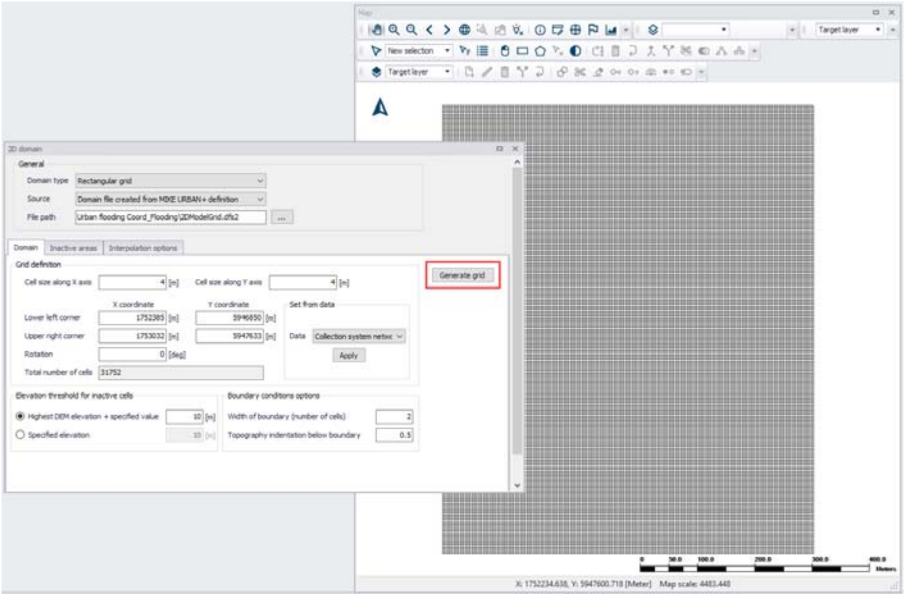




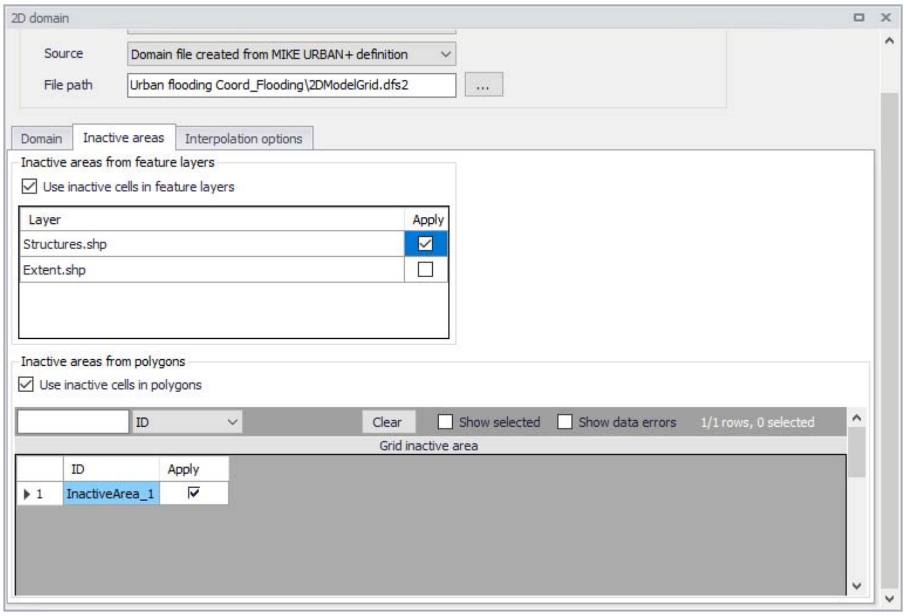
4. Customize other parameter values on the Domain tab, such as those under the 'Elevation threshold for inactive cells' and 'Boundary condition options' sections, if so desired. More details on these options are described in Chapter 3.1.2.



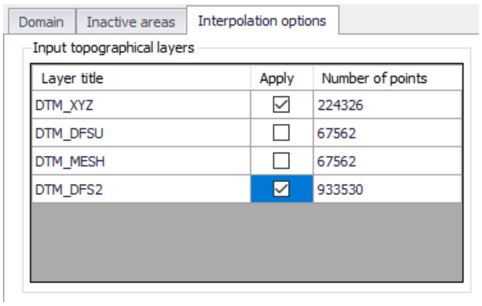
5. Click on the '**Generate grid'** button on the Domain tab page.



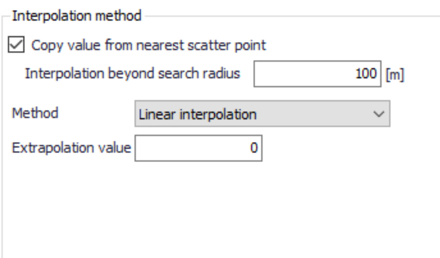
6. **Examine the generated grid.** Adjust grid sizes and extent, as needed. Re-generate the grid after changes to grid parameters.
7. **Define inactive areas, if any.** If inactive areas shall be defined for the 2D domain, specify them as polygons on the Map, or load polygon feature layers that represent them into the project (see Chapter 3.1.3 for more details). Make sure to activate the appropriate tick boxes to apply the desired feature layers as inactive areas in the subsequent interpolation.



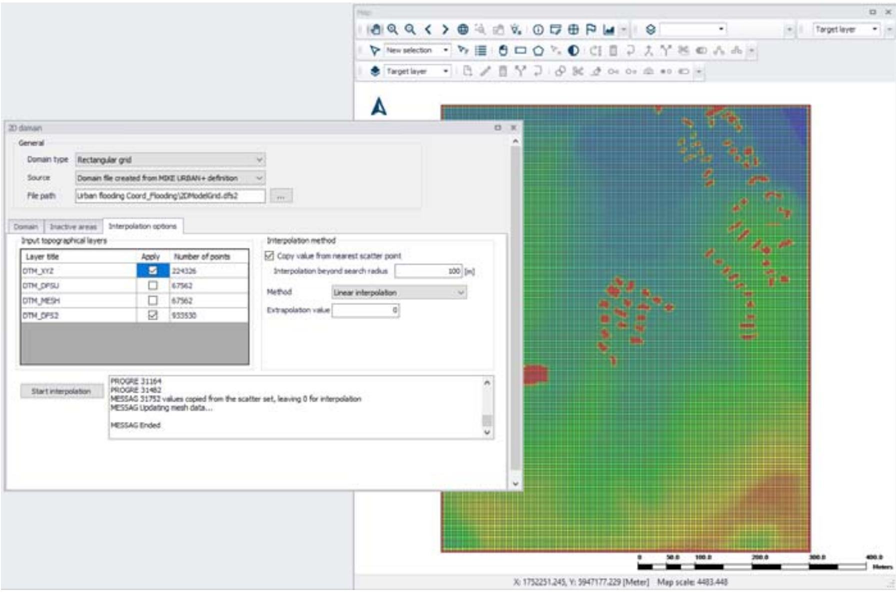
8. **Configure interpolation parameters.** Access the Interpolation Options tab page. Select the input topographical layers to use in the interpolation.



9. Choose the preferred interpolation method. More details on the methods are described in Chapter 3.1.4.

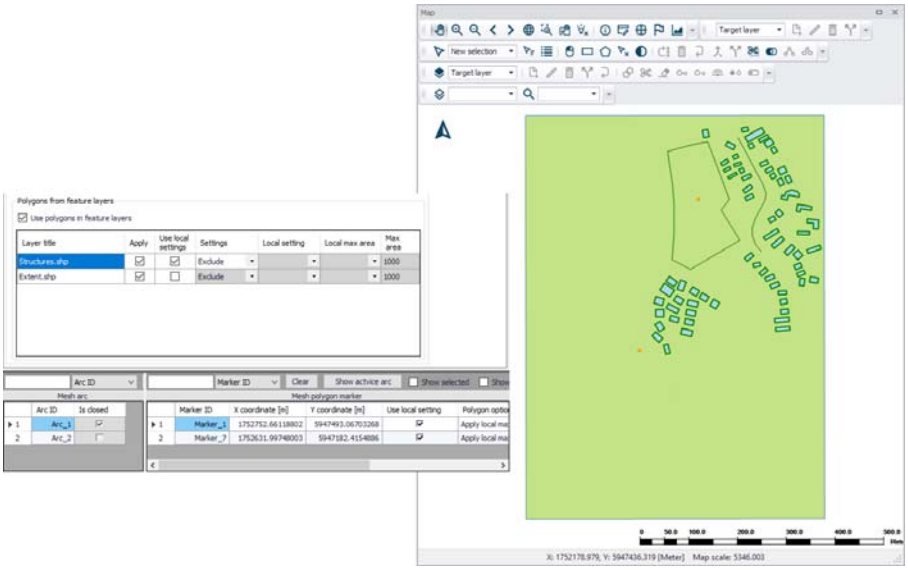


10. Click on the '**Start interpolation**' button. A progress bar appears, and the status of the process is shown on the log window.

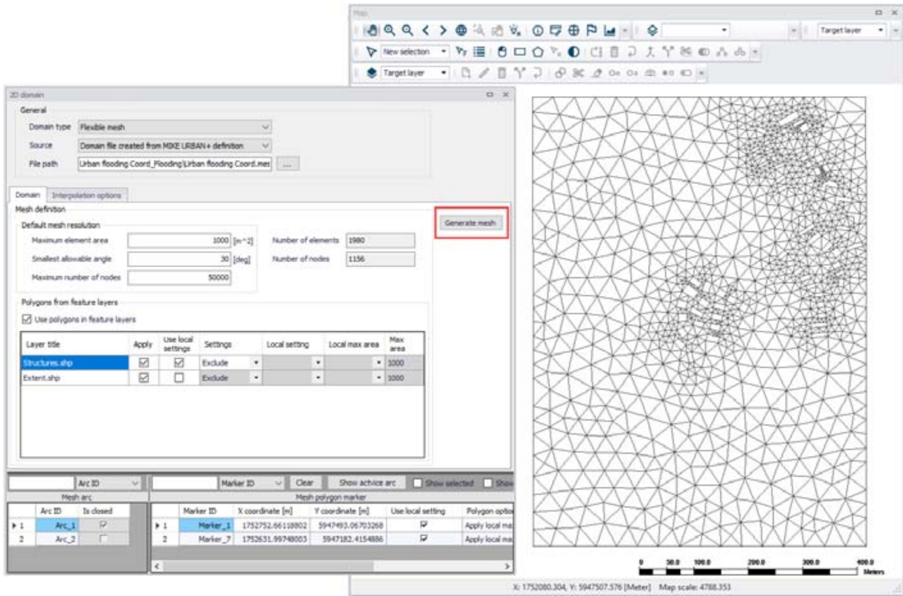


3.1.6 Creating a Flexible Mesh Workflow Example

1. Load desired input topographical layers to be used for 2D mesh value interpolation into the MIKE URBAN+ project.
2. On the 2D Domain editor, choose a ‘Flexible mesh’ domain type and ‘Domain file created from MIKE URBAN+ definition’ as source. Specify the file name and path for the domain *.MESH file to be created.
3. **Configure mesh domain parameters.** Define the mesh extent, and any local mesh areas via loaded polygon feature layers, or mesh arcs and polygons drawn directly on the Map. Pre-loaded feature layers are listed under the ‘Polygons from feature layers’ secondary table in the middle of the page, while arcs and polygons drawn on the Map are listed on the table at the bottom of the editor. More details on defining mesh areas are found in Chapter 3.1.2 - Flexible Mesh *(p. 18)*.

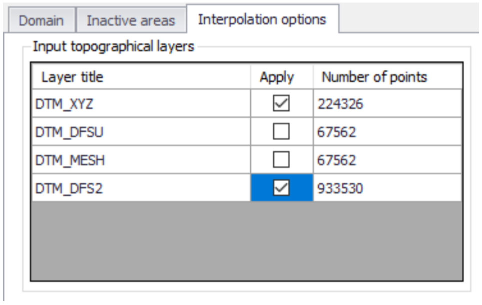


4. Customize other meshing parameters on the Domain tab, such as 'Maximum element area' and 'Smallest allowable angle', if so desired. More details on these options are described in Chapter 3.1.2 - Flexible Mesh *(p. 18)*.
5. Create a mesh by clicking on the '**Generate mesh**' button on the Domain tab page or in the 2D overland tab in the ribbon.

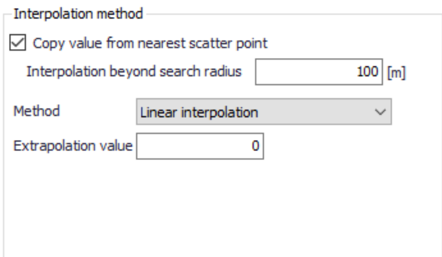


6. **Examine the generated computational mesh.** Adjust element sizes, the extent, or mesh areas using arcs and polygons, as needed. Re-generate the mesh after changes to meshing boundaries and parameters.

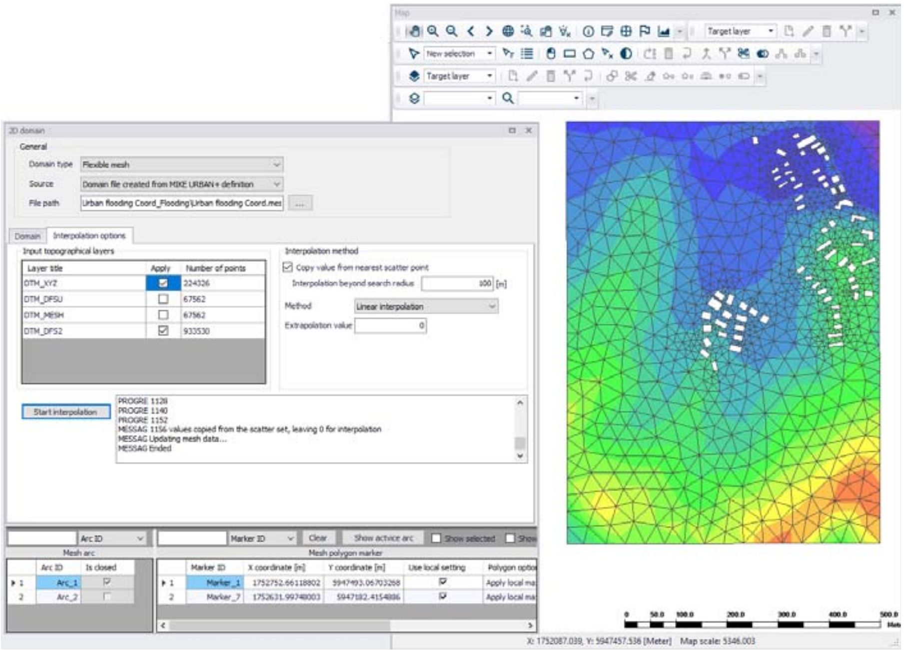
7. **Configure interpolation parameters.** When ready to finalize the 2D mesh, access the Interpolation Options tab page to begin interpolating topographical values onto the mesh. Select the input topographical layers to use in the interpolation.



8. Choose the preferred interpolation method. More details on the methods are described in Chapter 3.1.4 - Interpolation Method *(p. 25)*.



9. Click on the '**Start interpolation**' button. A progress bar appears, and the status of the process is shown on the log window.



3.2 2D Numerical Settings

Configure 2D model numerical solution settings in the 2D Numerical Settings editor.

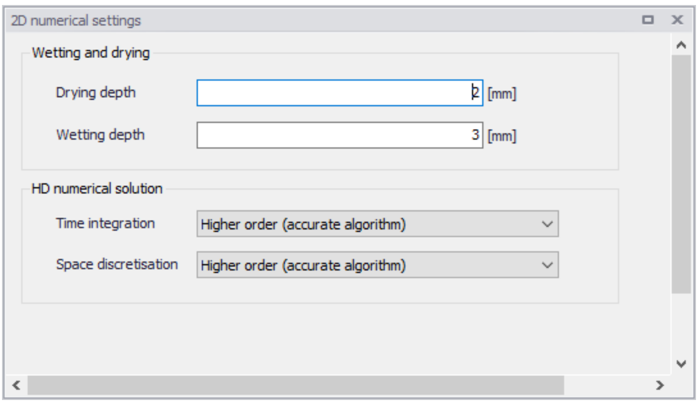


Figure 3.23 The 2D Numerical Settings editor

3.2.1 HD Numerical Solution

The 2D model simulation time and accuracy can be controlled by specifying the numerical scheme used in the calculations. Define the numerical solution technique to use in the 2D overland computations under the HD Numerical

Solution section. Both the scheme for time integration and for space discretization can be specified:

- **Lower order.** First-order explicit Euler scheme. Faster but less accurate.
- **Higher order.** Two-stage explicit second-order Runge-Kutta scheme.

More details on the numerical solution techniques are presented in the MIKE 21 Flow Model FM Hydrodynamic and Transport Module Scientific Documentation.

If the important processes are dominated by convection (flow), then higher order space discretization should be chosen. If they are dominated by diffusion, the lower order space discretization can be sufficiently accurate. In general, the time integration and space discretization methods should be the same.

Choosing the higher order scheme for time integration will increase the computing time by a factor of 2 compared to the lower order scheme. Choosing the higher order scheme for space discretization will increase the computing time by a factor of 1½ to 2. Choosing both as higher order will increase the computing time by a factor of 3-4. However, the higher order scheme will in general produce results that are significantly more accurate than the lower order scheme.

3.2.2 Wetting and Drying

Moving boundaries (i.e. wetting and drying fronts) in 2D overland numerical computations are dealt with by reformulating the computational problem for small depths and removing elements/cells from the calculation when the depths are very small. This approach is based on work by Zhao et al. (1994) and Sleigh et al. (1998). The reformulation is made by setting the momentum fluxes to zero and only taking the mass fluxes into consideration.

The depth in each element/cell in the 2D domain is monitored against the following threshold values:

- **Drying depth (h_{dry})**
- **Wetting depth (h_{wet})**

During overland computations, the elements are classed as dry, partially dry or wet in comparison to these depth thresholds. Also, the element faces are monitored to identify flooded boundaries:

- An element face is defined as flooded when the water depth at one side of the face is less than h_{dry} and the water depth at the other side of the face is larger than h_{wet}.

- An element is dry if the water depth is less than $\boldsymbol{h_{dry}}$ and none of the element faces are flooded boundaries. The element is removed from the calculation.
- An element is partially dry if the water depth is larger than $\boldsymbol{h_{dry}}$ and less than $\boldsymbol{h_{wet}}$, or when the depth is less than $\boldsymbol{h_{dry}}$ and one of the element faces is a flooded boundary. The momentum fluxes are set to zero and only the mass fluxes are calculated.
- An element is wet if the water depth is greater than $\boldsymbol{h_{wet}}$. Both the mass fluxes and the momentum fluxes are calculated.

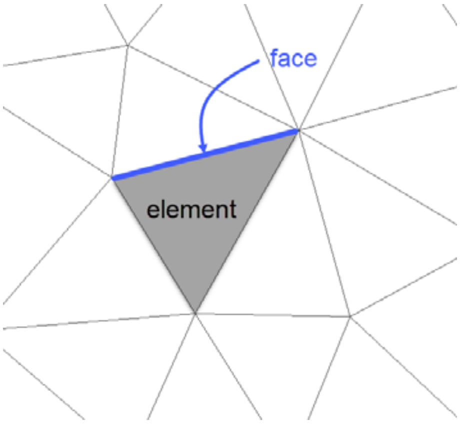


Figure 3.24 Illustration of elements and element faces

A non-physical flow across the face will be introduced for a flooded face when the surface elevation in the wet element on one side of the face is lower than the bed level in the partially wet element on the other side. To overcome this problem the face will be treated as a closed face.

In case the water depth becomes negative, the water depth is set to zero and the water is subtracted from the adjacent elements to maintain mass balance. In addition, the conserved variables h_u and h_v in the adjacent element where mass is subtracted, is corrected so that the velocities u and v remain the same. When mass is subtracted from the adjacent elements the water depth at these elements may become negative. Therefore an iterative correction of the water depth is applied (max 100 iterations).

Note: When an element is removed from the calculation, water is removed from the computational domain. However, the water depths at the elements, which are dried out, are saved and then reused when the element becomes flooded again.

Remarks and Hints

The default values are: drying depth h_{dry} = 5 mm and wetting depth h_{wet} = 10 mm.

The wetting depth, h_{wet}, must be larger than the drying depth, h_{dry}.

For floodplain simulations, the values for wetting and drying depths should be relatively low to minimize water balance errors. A reduction factor of up to 5 is viable.

3.3 2D Initial Conditions

The initial values for the hydrodynamic variables can be specified in three ways:

- Dry domain
- Uniform water level
- Varying in domain

To avoid the generation of shock waves and potential model instabilities, it is recommended that your initial surface elevation matches the boundary conditions at the start of the simulation.

3.3.1 Dry domain

In this case, the simulation will start with zero water depth and velocity across the entire domain.

3.3.2 Uniform water level

A single water level value can be specified to be applied across the domain. Elements where the topography is higher than this level will therefore be initially dry.

3.3.3 Varying in domain

To define spatially varying 2D initial conditions, the following sources of information can be used:

- MIKE URBAN+ 2D initial conditions layer of water level
- Background layer of water level
- 2D file with water level
- 2D file with water depth and velocities

When using an external 2D file (water level or water depth and velocity), the area in the data file must cover the model area. If a dfsu-file is used, piecewise constant interpolation is used to map the data. If a dfs2-file is used, bilinear interpolation is used to map the data. If the input data file contains a single time step, this field is used. In the case where the file contains several time steps, e.g. from the results of a previous simulation, the actual starting time of the simulation is used to interpolate the field in time. Therefore, the starting time must be between the start and end time of the file. Delete values (used in empty elements of the file) will be ignored and the value will be interpolated only from existing data.

MIKE URBAN+ 2D initial conditions layer of water level

This option enables an initial conditions layer with specified water levels to be graphically defined in the "Map" view using the tools available in the "2D overland" ribbon.

Steps to define initial conditions layer:

- Select the Target layer as "2D initial conditions"
- Click on "Create" and define polygons. Utilise the various editing tools to refine the polygons if necessary.

  Tip: specify a default water level in the 2D initial conditions table to automatically populate the water level value of a polygon as soon as it is digitised.

- In the 2D initial conditions table, the digitised polygons will be listed. Specify the initial water level values for each of these polygons, and prioritise them using the up and down buttons (important if there are overlaps)
- Where the model domain is not covered by the initial condition polygons the default value will be used in this area.

Clicking on "Review initial conditions file on map" will build a 2D initial conditions .dfs2 or dfsu file and add the layer to the map. This layer will also be saved in your project directory for use in the simulation. This file will also be created when starting the simulation, if it doesn't exist in the project directory of if input data have been modified.

Background layer of water level

Utilize predefined information to define the 2D initial water level, e.g. from shape files or text files.

Steps to define the Initial conditions details:

- Layer: Select the appropriate layer that has already been loaded into the MIKE URBAN+ interface as a background layer (available in the drop-down list) or browse to a saved file.
- Water level Item: Select the field in the layer that contains water level values
- Unit: specify the unit of the water level item

- Default water level: Value to be used if no water level value exists in the layer

Clicking on "Review initial conditions file on map" will build a 2D initial conditions .dfs2 or .dfsu file and add the layer to the map. This layer will also be saved in your project directory for use in the simulation. This file will also be created when starting the simulation, if it doesn't exist in the project directory of if input data have been modified.

2D file with water level

Use a previously defined .dfs2 or .dfsu file with a "Water level", "Elevation" or "Surface Elevation" item type as the 2D initial condition.

2D file with water depth and velocities

Use a previously defined .dfs2 or .dfsu file. The file must contain the total water depth and the velocity components in the x- and y-directions.

The initial conditions can be the result from a previous simulation in which case the initial conditions effectively act as a hot start of the flow field.

3.4 2D Surface Roughness

The way in which surface roughness is applied to the model can be specified in three different ways:

- Uniform
- Varying in domain
- Varying in domain and time

3.4.1 Uniform

When a Uniform roughness type is selected, the roughness formula must be selected (Manning M, Manning n or Chezy C) and one roughness value specified. This value will be applied to the entire 2D domain.

3.4.2 Varying in domain

To define spatially varying 2D roughness, the following sources of information can be used:

- MIKE URBAN+ 2D surface roughness layer
- Background layer
- 2D file

MIKE URBAN+ 2D surface roughness layer

This option enables a roughness layer to be graphically defined in the "Map" view using the tools available in the "2D overland" ribbon.

Steps to define roughness layers:

- Select the Target layer as "2D surface roughness"
- Click on "Create" and define polygons. Utilise the various editing tools to refine the polygons if necessary.

  Tip: specify a default roughness formula and value in the 2D surface roughness table to automatically populate the values of a polygon as soon as it is digitised.

- In the 2D roughness table, the digitised polygons will be listed. Specify the roughness values for each of these polygons, and prioritise them using the up and down buttons (important if there are overlaps)
  - To access predefined roughness values specified in the Materials table (Setup | Tables | Materials), specify the Value type as "Material" and select the Material type from the drop-down list.
  - To define specific roughness values, select the value type "Local" and input the value in the "Roughness" column
- Where the model domain is not covered by the roughness polygons the default value will be used in this area.

Clicking on "Review roughness file on map" will build a 2D Surface Roughness .dfs2 or dfsu file and add the layer to the map. This layer will also be saved in your project directory for use in the simulation. This file will also be created when starting the simulation, if it doesn't exist in the project directory of if input data have been modified.

Background layer

Utilize predefined information to define the 2D surface roughness. E.g. from Shape files, Tab files or Raster text files.

Steps to define the surface roughness details:

- Layer: Select the appropriate layer that has already been loaded into the MIKE URBAN+ interface as a background layer (available in the drop-down list) or browse to a saved file.
- Item: Select the field in the layer that contains roughness values
- Roughness formula: Select the formula to be used - Manning (M), Manning (n), Chezy (C)
- Unit: Depending on the formula, select the relevant SI or US unit

- Default roughness value: Value to be used if no roughness value exists in the layer

Clicking on "Review roughness file on map" will build a 2D Surface Roughness .dfs2 file and add the layer to the map. This layer will also be saved in your project directory for use in the simulation. This file will also be created when starting the simulation, if it doesn't exist in the project directory of if input data have been modified.

2D file

Use a previously defined .dfs2 or .dfsu file with "Mannings M", "Mannings n" or "Chezy No" item type as the 2D surface roughness.

If a dfsu file is used, piecewise constant interpolation is used to map the data. If a dfs2 file is used, bilinear interpolation is used to map the data. If the input data file contains a single time step, this field is used. In the case where the file contains several time steps, e.g. from the results of a previous simulation, the actual starting time of the simulation is used to interpolate the field in time. Therefore, the starting time must be between the start and end time of the file.

3.4.3 Varying in domain and time

The 2D surface roughness can be specified to vary both spatially and in time. In this case the user must select a previously defined .dfs2 or .dfsu, that varies in time. Browse to the file or if the file has been loaded as a background layer in MIKE URBAN+ it will be available from the drop-down list. Open the file via the "View" button to interrogate and edit the file. The item type for the values in this file must be "Mannings M", "Mannings n" or "Chezy No".

If a dfsu file is used, piecewise constant interpolation is used to map the data. If a dfs2 file is used, bilinear interpolation is used to map the data. If the data is varying in time the data must cover the complete simulation period. The time step of the input data file does not however need to match the time step of the hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

When using an external file (water level or water depth and velocity), the area in the data file must cover the model area. If a dfsu-file is used, piecewise constant interpolation is used to map the data. If a dfs2-file is used, bilinear interpolation is used to map the data. Delete values (used in empty elements of the file) will be ignored and the value will be interpolated only from existing data.

3.4.4 2D roughness general description

The 2D flow model uses either a Chezy number or a Manning number (M) assigned to each grid cell/element cell to describe the roughness (bed resistance). Note, that the Manning number used here is the reciprocal value of the Manning´s n described in some textbooks.

The bed resistance, τ, is determined by a quadratic friction law

$$\frac{\tau}{\rho_0} = c_f\, u|u| \tag{3.1}$$

where c_f is the drag coefficient, u is the velocity and ρ_0 is the density of the water.

The drag coefficient can be determined form the Chezy number, *C*, or Manning number, *M*.

$$c_f = \frac{g}{C^2} \tag{3.2}$$

$$c_f = \frac{g}{(Mh^{1/6})^2} \tag{3.3}$$

where *h* is the total water depth and *g* is the gravitational acceleration. The units of Chezy numbers and Manning numbers are $m^{1/2}/s$ and $m^{1/3}/s$ respectively.

Please note that the relation between the Manning number and the bed roughness length, k_s, can be estimated using the following

$$M = \frac{25.4}{k_s^{1/6}} \tag{3.4}$$

3.4.5 Recommended values

If the relative variation of the water depth is considerable then using variation in Manning's numbers are recommended. Values for Manning's M in the range 10-40 $m^{1/3}/s$ are normally used.

If Chezy numbers are applied, a model calibration can normally be achieved with values in the range 30 - 50 $m^{1/2}/s$.

Because of the definition of the resistance numbers the following applies when using Manning's M:

- Using a smaller resistance number increases the bed resistance

- Using a greater resistance number decreases the bed resistance

3.4.6 Remarks and hints

If the boundary conditions at one of your boundaries are inaccurate and you therefore have stability problems (blow-ups) at this boundary, you can specify a small band (2 to 4 grid lines) with a very high resistance. Manning numbers in the range 5 -10 $m^{1/3}/s$ have been applied successfully. However, this method should only be used if it is impossible to improve the boundary conditions. Furthermore, the simulation results in the area around the small resistance numbers should be used with caution.

Because $h^{1/6}$ is calculated for each grid point and at each time step when the Manning formulation is selected rather than the Chezy formulation, the computational time is increased.

3.5 2D Eddy Viscosity

The eddy viscosity can be specified in one of four different ways:

- None
- Uniform
- Varying in Domain
- Smagorinsky formulation

3.5.1 None

The eddy viscosity terms are omitted from the calculations.

3.5.2 Uniform

A constant eddy viscosity value in m^2/s is specified that applies to the entire 2D domain.

3.5.3 Varying in Domain

To define spatially varying 2D eddy viscosity, the following two sources of information can be used:

- MIKE URBAN+ 2D eddy viscosity layer
- 2D file

MIKE URBAN+ 2D eddy viscosity layer

This option enables an eddy viscosity layer to be graphically defined in the "Map" view using the tools available in the "2D overland" ribbon.

Steps to define eddy viscosity layers:

- Select the Target layer as "2D eddy viscosity"
- Click on "Create" and define polygons. Utilise the various editing tools to refine the polygons if necessary.

  Tip: specify a default eddy viscosity value in the 2D eddy viscosity table to automatically populate the values of a polygon as soon as it is digitised.

- In the 2D eddy viscosity table, the digitised polygons will be listed. Specify the eddy viscosity values for each of these polygons, and prioritise them using the up and down buttons (important if there are overlaps)
- Where the model domain is not covered by the roughness polygons the default value will be used in this area.

Clicking on "Review eddy viscosity file on map" will build a 2D eddy viscosity .dfs2 file and add the layer to the map. This layer will also be saved in your project directory for use in the simulation. This file will also be created when starting the simulation, if it doesn't exist in the project directory of if input data have been modified.

2D file

Use a previously defined .dfs2 or .dfsu file with "Viscosity" item type as the 2D eddy viscosity.

3.5.4 Smagorinsky formulation

Dynamically calculate eddy viscosity by means of the Smagorinsky formula.

A minimum and maximum value (m^2/s) for the eddy viscosity needs to be specified, along with either a uniform or spatially varying Smagorinsky coefficient. In the case of a spatially varying coefficient, a previously defined .dfs2 or .dfsu file with "Dimensionless factor" item type can be selected.

3.5.5 2D Eddy viscosity - general description

The effective shear stresses in the momentum equations contain momentum fluxes due to turbulence, vertical integration and sub-grid scale fluctuations. The terms are included using an eddy viscosity formulation to provide damping of short-wave length oscillations and to represent sub-grid scale effects (see e.g. Madsen et al., 1988; Wang, 1990).

The eddy coefficient, E, must fulfil the criterion:

$$\frac{E \cdot \Delta t}{\Delta l^2} \leq \frac{1}{2} \tag{3.5}$$

where l is a characteristic grid/element length (e.g. dx) and Δt the time step.

Smagorinsky formulation

Smagorinsky (1963) proposed to express sub-grid scale transports by an effective eddy viscosity related to a characteristic length scale. The sub-grid scale eddy viscosity in the horizontal direction is given by

$$\nu_t = c_s^2 l^2 \sqrt{2 S_{ij} S_{ij}} \tag{3.6}$$

where c_s is a constant, l is a characteristic length and the deformation rate is given by

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \quad (i, j = 1{,}2) \tag{3.7}$$

For more details on this formulation, the reader is referred to Lilly (1967), Leonard (1974), Aupoix (1984), and Horiuti (1987).

3.5.6 Recommended values

The Smagorinsky coefficient, c_S, should be chosen in the interval of 0.25 to 1.0.

When using the model for inland flooding the typical eddy viscosity value should be

$$E < 0{,}02 \cdot \frac{l^2}{\Delta t} \tag{3.8}$$

where l is a characteristic element length and Δt the time step.

3.5.7 Remarks and hints

In the same way as for the bed resistance you can use the eddy coefficients to damp out numerical instability. This method should only be used as a last resort to your stability problem: The schematisation of the bathymetry and the boundary conditions are usually the primary causes for a model blow-up.

When you use the Smagorinsky formulation the CPU time for a simulation is increased.

3.6 2D Structures

A number of structures can be implemented in the 2D domain:

- 2D Dikes
- 2D Culverts
- 2D Weirs

3.7 2D Dikes

In MIKE URBAN+, 2D dikes represent a natural or artificially constructed elongated ridge (levee, embankment, stopbank, etc) that can be used to regulate water levels. These often run parallel to the course of a channel/river in its floodplain or along low-lying coastlines.

3.7.1 Insert

This option enables the user to manually digitise along the path of the dike in the 2D domain.

Steps to digitise:

- Click on the "Insert" button at the top of the 2D dikes table
- Left click along the path of the dike in the 2D domain. Tip: right click to undo the previous graphical point
- Double click to complete the digitisation.

3.7.2 Insert from file

Import the path of the dike from an external source such as a Shape, XYZ or text file (.tab) file.

The file format is three space separated floats (real numbers) for the x- and y-coordinate and the crest level on separate lines for each of the points.

When importing a path from a file, ensure that the map projection (Longitude/Latitude, UTM, etc.) correlates with the MIKE URBAN+ project projection.

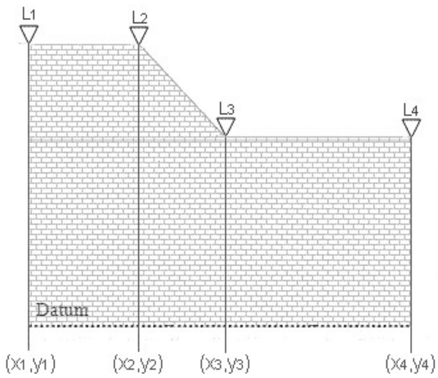
3.7.3 Location and Levels

Every dike digitised or imported will have georeferenced points (x and y coordinates) which together make up a polyline along the path of the dike. A minimum of two points is required. The polyline defines the width of the dike perpendicular to the flow direction. The polyline is composed of a sequence of line segments. The line segments are straight lines between two successive points. The polyline (cross section) in the numerical calculations is defined as a section of element faces. The face is included in the section when the line between the two element centres of the faces crosses one of the line segments (see Figure 3.38).

Note: The faces defining the line section for the dike will be listed in the log-file.

Insert, delete, manually edit values or change the order of the digitised points as needed.

The geometry of a dike is defined as shown below where the Z values refer to the elevation / crest level of the dike.



Easting	Northing	Crest level
X1	Y1	L1
X2	Y2	L2
X3	Y3	L3
X4	Y4	L4

Figure 3.25 Definition sketch of spatial varying dike geometry

3.7.4 Attributes

Control which dikes are activated during the simulation using the "Apply" switch.

A number of parameters define the dike characteristics:

Dampening delta depth

When the water level gradient across a structure is small, the corresponding gradient of the discharge with respect to the water levels is large. This in turn may result in a very rapid flow response to minor changes in the water level upstream and downstream. As a way of controlling this effect, a dampening delta depth has been introduced. The critical water level difference defines the water level difference below which the discharge gradients are suppressed. The default setting is 0.01 meter. If a structure shows oscillatory behaviour it is recommended to increase this value slightly.

Weir coefficient

A dike is defined as a cross section and in the numerical calculations the cross section is defined as a section of element faces which is treated as an internal discharge boundary. (However, the flux contribution to the continuity equation is corrected to secure mass conservation). The discharge across each face in the section is calculated using an empirical formula. The discharge over an element face with associated width, is calculated based on the water level in the elements to the left and right of the face. The upstream water level is then the highest of the two water levels and the downstream the smallest.

The flow, Q, over a section of the dike corresponding to an element face with the length (width), *w*, is based on a standard weir expression, reduced according to the Villemonte formula :

$$\begin{aligned} Q &= wC(H_{us}-H_w)^{3/2}\left(1-\left(\frac{H_{ds}-H_w}{H_{us}-H_w}\right)^{3/2}\right)^{0,385} && \text{for } H_{us}>H_{ds}>H_w \\ Q &= wC(H_{us}-H_w)^{3/2} && \text{for } H_{us}>H_w\geq H_{ds} \\ Q &= 0 && \text{for } H_w\geq H_{us}>H_{ds} \end{aligned} \tag{3.9}$$

where Q is discharge through the structure, w is the local width (cell face width), C is discharge coefficient, H_{us} is upstream water level, H_{ds} is downstream water level and H_w is the crest level taken with respect to the global datum.

The default value of the weir coefficient is 1.838.

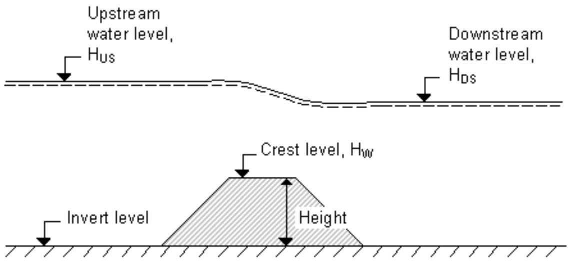


Figure 3.26 Definition sketch for Dike Flow

3.7.5 Relative change to crest level

The crest level relative change for a dike can be specified as:

- None

- Uniform
- Varying along dike
- Varying along dike and in time

None
The crest level does not change and the original z levels defined for the dike are used.

Uniform
The crest level is defined by a specific value specified as an input (in m).

Varying along dike
The crest level is predefined for the dike in an external .dfs1 file. Browse to a file already created, edit a file or create a new file (the MIKE Zero interface is opened).

Varying along dike and in time
The crest level varying in time is predefined for the dike in an external .dfs1 file. Browse to a file already created, edit a file or create a new file (the MIKE Zero interface is opened).

For the external .dfs1 file, the number of grid points needs to correspond to the number of points, which is used to define the location of the dike. The data must cover the complete simulation period. The time step of the input data file does not however have to be the same as the time step of the hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

3.8 2D Culverts

A typically embedded structure that allows water to flow from an open channel through an obstruction such as a road, railroad, wall, etc. can be represented by a 2D culvert in MIKE URBAN+. The result is that a head loss is applied between two 2D cells, a line of cells or two cross-sectional lines of 2D cells.

3.8.1 Culvert types

Two types of culverts are available:

- Short culvert
- Long culvert

Short culvert

A short culvert is defined as a cross section where the total discharge across the cross section is calculated using empirical formulas and distributed along the cross section. In the numerical calculations the cross section is defined as a section of element faces which are treated as an internal discharge boundary.

The culvert points specify the cross section of the channel. To identify the exact cells that are linked, the M21 log file can be interrogated where each upstream and downstream point is listed. During the model initialisation period, the model will look at the level of these cells and check to see if they are greater than or equal to the channel invert. If the MIKE 21 levels are higher than the culvert invert level, then an error message will be thrown.

Tip: It is important to make sure that the MIKE 21 cell levels across the culvert are less than or equal to the respective culvert upstream and downstream invert levels.

The short culvert module was designed for applications where the length of the culvert is similar or smaller than the grid size. Thus when you try to use a culvert in a 2D grid with a fine grid size it is important to take care with schematization. Culverts that are much longer than the grid size can instead be modelled as a long culvert or in 1D coupled to 2D.

Example of a short culvert setup:

There are two images in Figure 3.27 below. The left image illustrates a case with a stream running south to north and a road bisecting the stream, and the right image illustrates the grid modified in order to represent a culvert under the road, within the MIKE 21 model. The grid spacing is 10m x 10m. The black line represents the culvert cross section, defined in the model setup. The length of the culvert will be the waterway length; in this case will be 1 grid cell or 10 meters. The head loss from the culvert occurs across the two cell faces. The water level upstream of the culvert is calculated for cells 1 and 2 and the downstream levels calculated for cells 3 and 4.

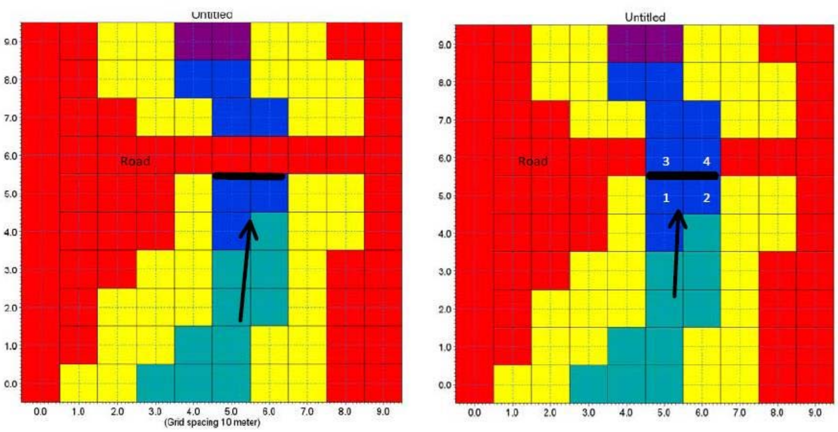


Figure 3.27 Setup of a short culvert

Long culvert

A long culvert is defined by a longitudinal line where inlet and outlet locations are defined as two extent lines at the ends of the transversal line, see Figure 3.28 below. The polyline (line section) in the numerical calculations for each of the two extent lines is defined as a section of element faces. A long culvert is treated as two connected area sources where the total discharge is calculated using empirical formulas. For each of the two extent lines, the area is determined at the area of the elements to the right of the section of element faces.

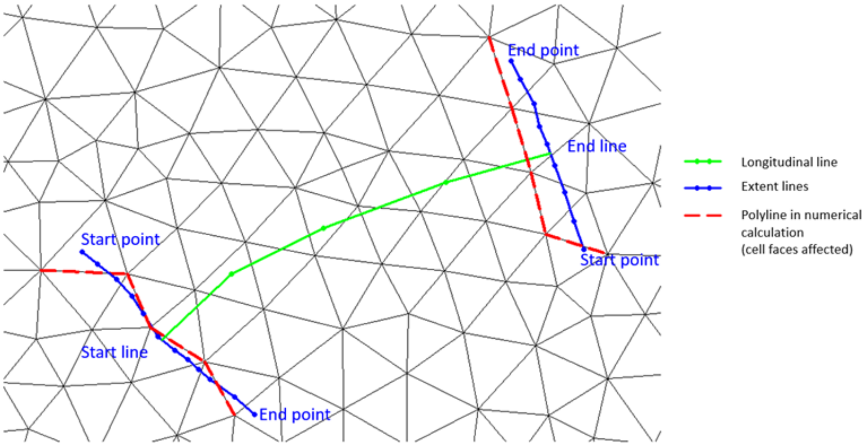


Figure 3.28 Definition sketch for a long culvert

3.8.2 Insert

From the MIKE URBAN+ interface, a short or long 2D culvert can be manually digitised in the 2D domain.

Steps to digitise:

- Click on the "Insert" button at the top of the 2D culvert table
- Select whether to define a short culvert (digitise along width of culvert) or long culvert (specify width and digitize longitudinal line of culvert)
- Left click along the path of the culvert in the 2D domain. Tip: right click to undo the previous graphical point.
- Double click to complete the digitisation.

In the case of a long culvert, click on "Edit end sections coordinates" to manually edit the points for the extent lines if required (as per Figure 3.28).

3.8.3 Insert from file

The path of the culvert can be imported from an external source such as a Shape, XYZ or text file (.tab) file.

Steps to utilise an external file:

- Click on "Insert from file" button at the top of the 2D culvert table
- Select whether to define a short culvert (along width of culvert) or long culvert (specify width and import longitudinal line of culvert)
- Browse to the external file to be imported.

In the case of a long culvert, click on "Edit end sections coordinates" to manually edit the points for the extent lines if required (as per Figure 3.28).

When importing a path from a file, ensure that the map projection (Longitude/Latitude, UTM, etc.) correlates with the MIKE URBAN+ project projection.

3.8.4 Geometry

The culvert geometry defines the geometrical shape of the active flow area of the culvert.

The location and geometrical layout of the culverts must be specified.

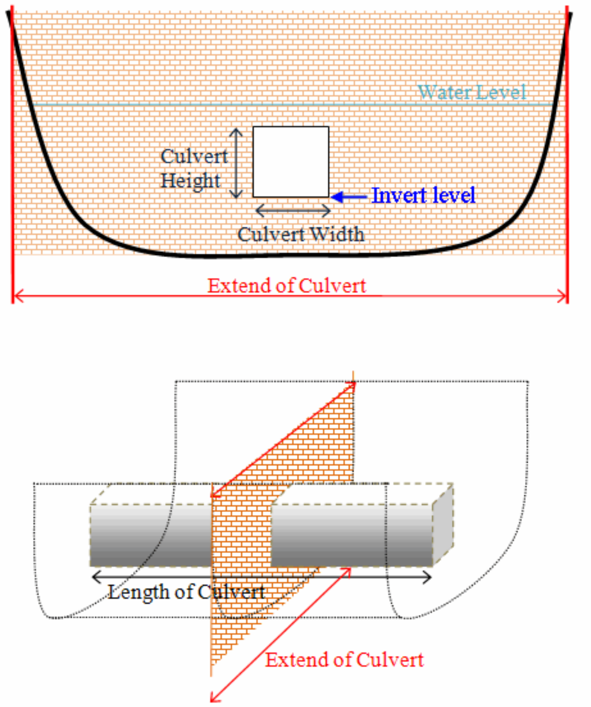


Figure 3.29 Setup definition of culvert

The cross-sectional geometry of a culvert can be specified as:

Rectangular

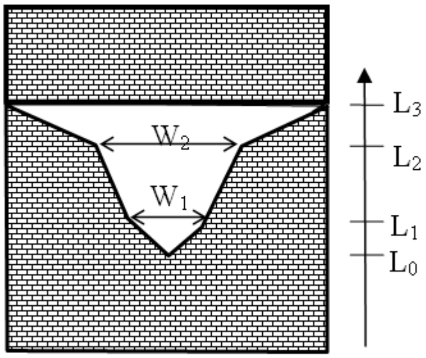
The geometry is specified by the width and height.

Circular

The geometry is specified by the diameter.

Irregular Level-Width Table

The geometry is specified using a level/width table. The Level/Width table defines the Culvert shape as a set of corresponding levels and flow widths. Values in the level column must be increasing.



Level	Width
L_0	0
L_1	W_1
L_2	W_2
L_3	W_3

Figure 3.30 Definition sketch for irregular culvert

No. of Culverts

'No. of Culverts' is a number identifying how many culverts exist at the specific culvert location with identical geometrical definition.

An example

Five identical shaped draining pipes are placed next to each other in an earth dam, and in order not to make 5 individual culvert definitions - one for each pipe - the 'No. of Culverts' in this case can be defined as 5 and the simulation engine will recognize that 5 culverts of identical shape and size are located here and flow calculations will take this into account accordingly.

Upstream Invert

Invert level to the left of the cross section for a short culvert and at the start line for a long culvert.

Downstream Invert

Invert level to the right of the cross section for a short culvert and at the end line for a long culvert.

Length

Length of the culvert.

Manning's n

Manning's bed resistance number along the culvert (for friction loss contribution). n = 1/M (Manning number)

3.8.5 Attributes

Control which culverts are activated during the simulation using the "Apply" switch.

A number of parameters define the culvert characteristics as described in the following.

Dampening delta depth

When the water level gradient across a structure is small the corresponding gradient of the discharge with respect to the water levels is large. This in turn may result in a very rapid flow response to minor changes in the water level upstream and downstream. Alpha zero is the water level difference at which the discharge calculation is described by a linear variation. If the water level difference is below this value the discharge gradients are suppressed. The default setting is 0.01 meter. If a structure shows oscillatory behaviour it is recommended to increase this value slightly.

Non return flap

None
No valve regulation applies (flow is not regulated).

Only left to right flow
Only flow in the positive flow direction is allowed. Valve regulation does not allow flow in the negative flow direction in which case the flow through the structure will be zero.

The flow direction is positive when the flow occurs from the right of the line structure to the left, positioned at the first point and looking forward along the line section.

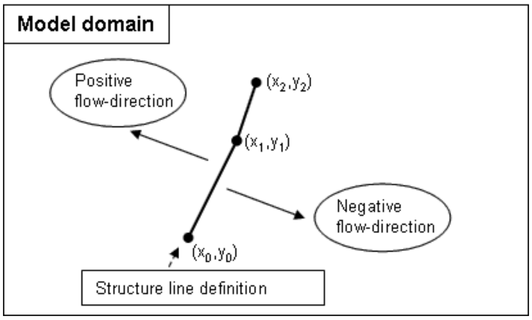


Figure 3.31 Positive and Negative flow direction definition for weirs and short culverts

Only Negative Flow

Only flow in the negative flow direction is allowed. Valve regulation does not allow flow in the positive flow direction in which case the flow through the structure will be zero.

Flow distribution

For a short culvert the total discharge is calculated based on the mean water level in the real wet elements to the left and right of the section of faces. For a long culvert the total discharge calculated based on the mean water level in the real wet elements to the right of the section of faces for the two extent lines. The mean level is calculated using the length of the element faces as the weighting factor. Real wet elements are elements where the water depth is larger than the wetting depth. The upstream water level is then the highest of the two water levels and the downstream water level the smallest.

The distribution of the calculated total discharge along the section faces can be specified in two ways

- Uniform
- Non-uniform

When the discharge for a short culvert is distributed to the 2D faces, if these elements are not wet, the discharge is distributed to the faces where the upstream elements are wet elements. When non-uniform distribution is applied the discharge will be distributed as it would have been in a uniform flow field with the Manning resistance law applied, i.e. relative to $h^{5/3}$, where h is the total water depth.

When the discharge for a long culvert is distributed to the 2D faces, if no elements are wet, the discharge is distributed uniformly to all faces in the section. When non-uniform distribution is applied the same approach as for short culverts is used with the Manning resistance law applied. The non-uniform distribution is, in most cases, a good approximation. This does not apply if there are very large variations over the bathymetry or the geometry.

For Composite structures the distribution for the first structure is applied.

Section type

A culvert structure can be defined with either a Closed or an Open section type.

If set to open, the culvert will never run full or partially full, therefore only those flow conditions which represent a free water surface are modelled. When the water level is higher than the soffit the hydraulic parameters are calculated based on a section extended vertically upwards with a width equal to that at the soffit. For example, in the case of a rectangular section the height value is essentially redundant as the cross-section will be modelled as an open section of constant width.

In the case of a circular section, this switch is invalid and will be set to closed.

Momentum

For a long culvert it is possible to include or exclude a contribution to the momentum equations at the outlet location. This contribution is estimated as the discharge multiplied by a velocity. Here the magnitude of the velocity is calculated as the discharge divided by the local total water depth. The direction used for the two extent lines is the direction of the first and last segment of the transversal polyline. If the transversal polyline only contains one segment (two points), the direction is determined as the direction perpendicular to the line given by the first and last point of the extent line.

Head Loss Factors

The factors determining the energy loss occurring for flow through hydraulic structures.

The following head loss factors can be defined (for positive and negative flow directions):

- Inflow (contraction loss)
- Outflow (expansion loss)

For definition of flow direction for a short culvert refer to Figure 3.31.

For a long culvert the flow direction is positive when flow is from the start line to the end line (see Figure 3.28).

Calibration factors can also be applied to the head losses.

The total head loss, ΔH_{loss} through a culvert is given by

$$\Delta H_{loss} = \frac{q^2}{2g}\left(\frac{\zeta_1}{A_{s_1}^2} + \frac{\zeta_f}{A_{s_A}^2} + \frac{\zeta_2}{A_{s_2}^2}\right) \tag{3.10}$$

where A_{S1}, A_{S1} and A_{S1} are the mean cross section areas along the length of the culvert and q is the discharge, ζ_1 is the entrance or contraction loss coefficient, ζ_2 is the outlet or expansion loss coefficient and ζ_f is the friction loss. ζ_1 and ζ_2 are calculated by

$$\zeta_1 = \zeta_{in}\left(1 - \frac{A_s}{A_1}\right) \tag{3.11}$$

$$\zeta_2 = \zeta_{out}\left(1 - \frac{A_s}{A_2}\right)^2 \tag{3.12}$$

The upstream and downstream cross section areas, A_1 and A_2 are not processed and extracted in the present implementation and hence, defined as an infinite value. Contraction and expansion losses are therefore assumed to be applied in full using the defined inflow and outflow loss coefficients, ζ_{in} and ζ_{out}. The friction loss coefficient is calculated using the Manning formula

$$\zeta_f = \frac{2gLn^2}{R^{4/3}} \tag{3.13}$$

where L is the culvert length, n is Manning's coefficient and R is the mean hydraulic radius along the culvert. The Manning's n-value depends on the interior surface of the culvert. Table values can be found in literature. For example, a concrete culvert n would typically range from 0.011 to 0.017.

3.9 2D Weirs

In MIKE URBAN+, the 2D weir is used to represent a structure that raises the water level upstream and regulates flow downstream. E.g. a low dam.

A weir is defined as a cross (line) section where the total discharge across the cross section is calculated using empirical formulas and distributed along the cross section. In the numerical calculations the weir cross section is defined as a section of element faces (from the 2D domain mesh/grid which is treated as an internal discharge boundary).

3.9.1 Insert

This option enables the user to manually digitise along the path of the weir in the 2D domain.

Steps to digitise:

- Click on the "Insert" button at the top of the 2D weir table
- Left click along the width of the weir in the 2D domain. Tip: right click to undo the previous graphical point
- Double click to complete the digitisation.

3.9.2 Insert from file

Import the path of the weir from an external source such as a Shape, XYZ or tab file.

The text file format is two space separated floats (real numbers) for the x- and y-coordinate on separate lines for each of the points.

3.9.3 Location and Levels

Weirs are defined in the domain as a cross (line) section specified as a list of points in the location and levels table (a minimum of two points required). The section is composed of a sequence of line segments. The line segments are straight lines between two successive points (see Figure 3.38).

Note: The faces defining the line section for the weir will be listed in the MIKE 21 logfile.

Insert, delete, manually edit values or change the order of the digitised points as needed.

3.9.4 Geometry

The geometrical layout of the weirs must be specified.

Weir Type

A range of formulas are available to be applied to each weir structure:

- Broad crested weir formula
- Villemonte formula
- Honma formula

Broad crested weir formula

For a broad crested weir the geometrical shape of the active flow area of the weir needs to be defined. The geometry is defined as a Level-width relationship, where the Level/Width table defines the weir shape as a set of corresponding set of levels and flow widths. Values in the level column must be continuous, increasing values.

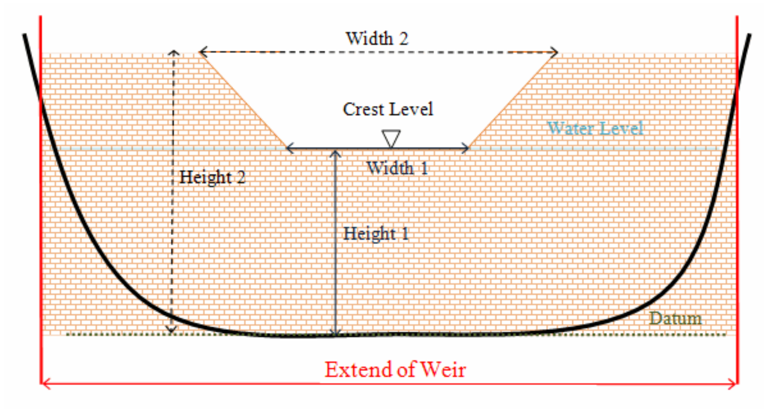
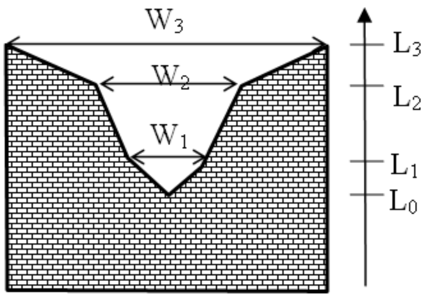


Figure 3.32 Setup definition of contracted weir



Level	Width
L_0	0
L_1	W_1
L_2	W_2
L_3	W_3

Figure 3.33 Definition sketch of broad crested weir geometry

Levels are defined relative to the datum (starting from the crest or sill level, upwards). A datum value for the weir may be used to shift the levels by a constant amount. This is typically used if the weir geometry has been surveyed with respect to a local benchmark.

The standard formulations for flow over a broad crested weir are established on the basis of the weir geometry and the specified head loss and calibration coefficients (see “Head Loss Factors” on page 64). These formulations assume a hydrostatic pressure distribution on the weir crests. Different algorithms are used for drowned flow and free overflow, with an automatic switching between the two.

The energy loss over a weir is given by:

$$q = \zeta_t \frac{V_s}{2g} \tag{3.14}$$

where ζ_t is the total head loss coefficient and V_s is the mean cross sectional velocity at the structure. The total head loss coefficient, ζ_t is composed of entrance, ζ_1, and exit, ζ_2, loss coefficients. The coefficients are generally related to the input parameters for inflow, ζ_{in}, and outflow, ζ_{out}, and the changes in velocity, V and area A.

$$\zeta_t = \zeta_1 + \zeta_2 = \zeta_{in}\left(\frac{V_1}{V_s}\right) + \zeta_{out}\left(\frac{A_s}{A_2}\right)^2 \tag{3.15}$$

where suffix '1' and '2' represents velocity and area on the inflow and outflow side of the structure respectively, and 's' represents the velocity and area in the structure itself. However, in the present implementation, upstream and downstream cross sections are not extracted and accordingly, tabulated relations on cross section areas as a function of water levels are not known. Instead, upstream and downstream areas are set to a large number resulting in a full loss contribution from the head loss factors defined

$$\zeta_t = \zeta_1 + \zeta_2 = \zeta_{in} + \zeta_{out} \tag{3.16}$$

Care must be taken when selecting loss coefficients, particularly in situations where both subcritical and supercritical flow conditions occur. When flow conditions change from subcritical to supercritical (or the Froude number, FR, becomes greater than 1), the loss coefficients ζ_{in} and ζ_{out} are modified:

- If FR > 1 for upstream conditions, then $\zeta_{in} = \zeta_{in}/2$
- If FR > 1 for downstream conditions, then $\zeta_{out} = \zeta_{out}/2$

The critical flows are multiplied by the critical flow correction factor, α_c, specified as the free overflow head loss factor. Typically, a value of 1.0 is used.

Villemonte formula

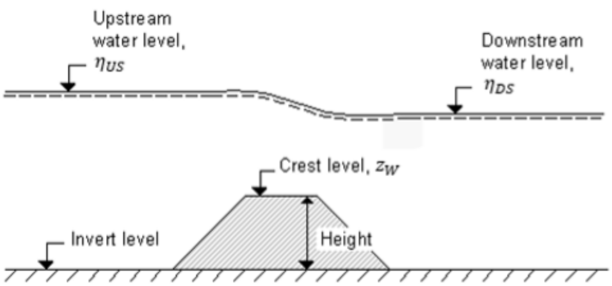


Figure 3.34 Definition sketch for Weir Flow

For this type of weir the width, height and invert level for the weirs must be specified (Figure 3.34). The width is perpendicular to the flow direction. Typically, the invert level coincides with the overall datum. In addition, a weir coefficient and weir exponent must be specified.

This formula is based on a standard weir expression, reduced according to the Villemonte formula.

$$q = C(\eta_{US} - z_w)^k \left(1 - \frac{\eta_{DS} - z_w}{\eta_{US} - z_w}\right)^{-0,385} \cdot W \tag{3.17}$$

where q is the discharge through the structure, w is the width, C is the weir coefficient, k is the weir exponential coefficient, η_{US} is the upstream water level, η_{DS} is the downstream water level and z_w is the weir level. The invert level is the lowest point in the inlet or outlet section respectively.

Honma formula

For this type of weir the width and crest level for the weirs must be specified (see Figure 3.34). The crest level is taken with respect to the global datum. The width is perpendicular to the flow direction. In addition, a weir coefficient must be specified.

The Honma formula is expressed as:

$$q = \begin{cases} C_1(\eta_{US} - z_w)\sqrt{\eta_{US} - z_w}W & \dfrac{(\eta_{DS} - z_w)}{\eta_{US}} < \dfrac{2}{3} \\ C_1(\eta_{DS} - z_w)\sqrt{\eta_{US} - \eta_{DS}}W & \dfrac{(\eta_{DS} - z_w)}{\eta_{US}} \geq \dfrac{2}{3} \end{cases} \tag{3.18}$$

where q is the discharge through the structure, W is the width, C_1 and $C_2 = 1{,}5\sqrt{3}C_1$ are the two weir coefficients, η_{US} is the upstream water level, η_{DS} is the downstream water level and z_w is the weir level.

3.9.5 Attributes

The user can control which culverts are activated during the simulation using the "Apply" switch.

A number of parameters define the weir characteristics.

Dampening delta depth

When the water level gradient across a structure is small the corresponding gradient of the discharge with respect to the water levels is large. This in turn may result in a very rapid flow response to minor changes in the water level upstream and downstream.

The dampening delta depth is the water level difference at which the discharge calculation is described by a linear variation. If the water level difference is below this value the discharge gradients are suppressed.

The default setting is 0.01 meter. If a structure shows oscillatory behaviour it is recommended to increase this value slightly.

Non return flap

None

No valve regulation applies (flow is not regulated).

Only left to right flow

Only flow in the positive flow direction is allowed. Valve regulation does not allow flow in the negative flow direction in which case the flow through the structure will be zero.

The flow direction is positive when the flow occurs from the right of the line structure to the left, positioned at the first point and looking forward along the line section.

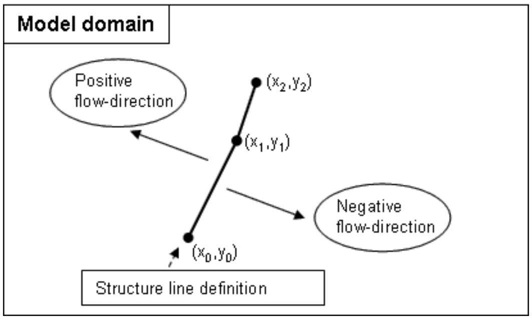


Figure 3.35 Positive and Negative flow direction definition for weirs and culverts

Only Negative Flow

Only flow in the negative flow direction is allowed. Valve regulation does not allow flow in the positive flow direction in which case the flow through the structure will be zero.

Flow distribution

The total discharge across the weir is calculated based on the mean water level in the real wet elements that directly neighbour the section of 2D mesh/grid faces that define the weir. For an irregular mesh, the mean level is calculated using the length of the element faces as the weighting factor. Real wet elements are elements where the water depth is larger than the wetting depth. The upstream water level is then the highest of the two water levels and the downstream water level the smallest.

The distribution of the calculated total discharge along the section faces can be specified in two ways

- Uniform

- Non-uniform

When non-uniform distribution is selected the discharge will be distributed as it would have been in a uniform flow field with the Manning resistance law applied, i.e. is relative to $h^{5/3}$, where h is the depth. This distribution is, in most cases, a good approximation. This does not apply if there are very large variations over the bathymetry or the geometry. The distribution of the discharge only includes the faces for with the element to the left and the right of the face is a real wet element. In no elements on the downstream side of the structure are real wet elements the distribution is determined based on the upstream information.

For Composite structures (when multiple short culverts and/or weirs are located on the same faces) the distribution for the first structure is applied.

Head Loss Factors

The factors determining the energy loss occurring for flow through hydraulic structures. Head Loss Factors are only applied for a broad crested weir.

The following head loss factors can be defined (for positive and negative flow directions):

- Inflow (contraction loss)
- Outflow (expansion loss)

Calibration factors can also be applied to the head losses.

3.10 Composite structures

A composite structure is a combination of weirs and short culverts. It is defined as a cross section in the 2D model where total discharge across the cross section is calculated using empirical formulas for weirs and culverts and distributed along the cross section.

In the numerical calculations the cross section is defined as a section of element faces which is treated as an internal discharge boundary but the flux contribution to the continuity equation is corrected to secure mass conservation. The approach for calculating the mean water level and for distribution of the total discharge are the same as the approach used for weirs and culverts.

An example of a composite structure could be a bridge with multiple waterways. Such a structure can be described by a number of culverts, each defining an individual waterway. Additionally, for a potential bridge deck overtopping a weir can be included to describe such an overflow.

A set of structures forming a composite structure are recognised by the program from the location definitions. Locations must be completely identical for all the structures forming the composite structure. That is, the table of coordinates defining the structure locations must be exactly identical (number of coordinates and coordinate values) for all structures defined.

For more complex schematisations of structures, consider modelling the structures in 1D, coupled to 2D.

Examples of composite structures are given below.

Example 1 - A wide weir with a small opening

Consider a weir as illustrated in Figure 3.36.

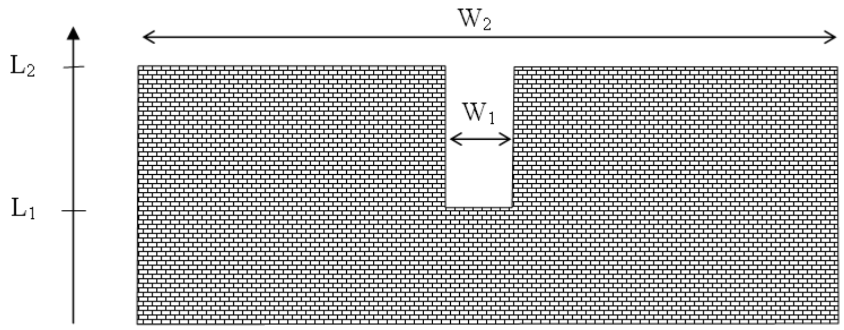


Figure 3.36 Wide weir with a small opening

There are a number of possibilities when modeling this:

1. One weir
2. Two weirs with width W_1 and width (W_2-W_1), respectively
3. Three weirs with widths W_1 and two with width (W_2-W_1)/2, respectively

Using the first approach is only appropriate if the weir can be contained within a single grid cell. The second approach may be used if the weir spans multiple cells, keeping in mind that the flow over the highest crest (L_2) is uniformly distributed over all the affected cells. The third approach will give the best representation of the flow. Note that the location needs to be defined for each of the segments for case 2 and 3.

Example 2 - A wide weir with multiple culverts

Consider a structure as illustrated in Figure 3.37.

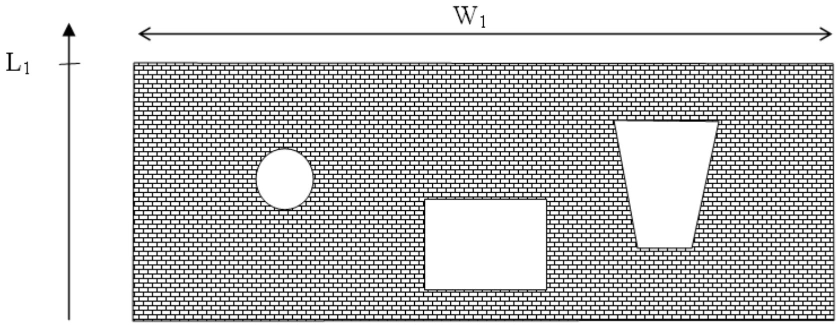


Figure 3.37 Wide weir with multiple culvert openings

The composite structure should be implemented as four separate structures:

1. A weir with a constant crest level L1 and a location defined by the full extent of the weir
2. A circular culvert and a location defined by the full extent of the weir
3. A rectangular culvert and a location defined by the full extent of the weir
4. An irregular culvert described by a level/width table and a location defined by the full extent of the weir

Note that the location needs to be defined for each of the four structure components separately. The location line should correspond to the maximum width of the structure component while still obeying the minimum requirement with respect to intersecting the line segments connecting cell centres.

3.11 Structures - General description

Weirs, culverts, and dikes are defined as line sections. The location in the domain of a line section is given by a number of geo-referenced points which together make up a polyline. This is illustrated in Figure 3.38. The polyline defines the width of the cross section perpendicular to the flow direction. A minimum of two points is required. The polyline is composed of a sequence of line segments. The line segments are straight lines between two successive points. The polyline (line section) in the numerical calculations is defined as a section of element faces. The face is included in the section when the line between the two element centers of the faces crosses one of the line segments. If two faces in a triangular element are part of the same face section, the face section is corrected so that these two faces are excluded from the face section and instead the third face in the triangle is applied. The left and right side of the of the line section is defined by positioning at the start point and looking forward along the cross-section.

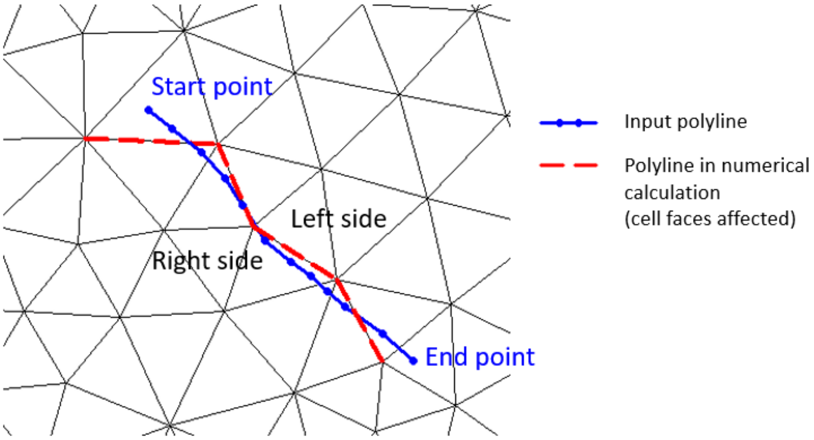


Figure 3.38 The location of a line section

To ensure a proper mapping simply extend the polyline defining the structure location so that it intersects a line segment connecting cell mid-points, see

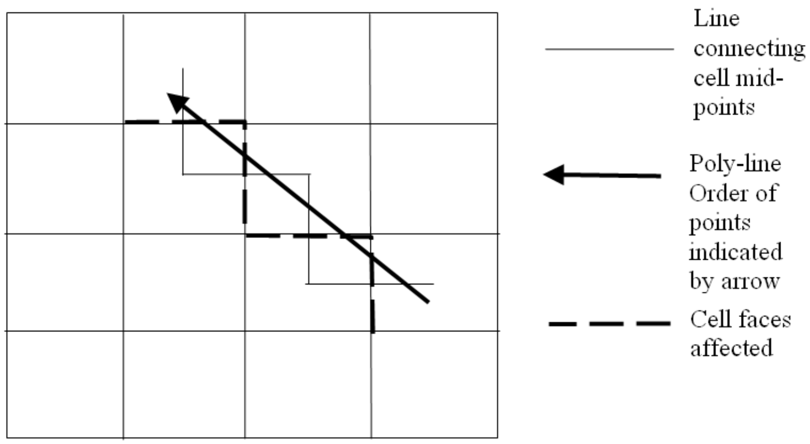


Figure 3.39 Definition of cell mid-points

3.12 List of References

Aupoix, B. Eddy Viscosity Subgrid Scale Models for Homogeneous Turbulence, in Macroscopic Modelling of Turbulent Flow, Lecture Notes in Physics, Proc. Sophie-Antipolis, France, 1984.

Horiuti, K. Comparison of Conservative and Rotational Forms in Large Eddy Simulation of Turbulent Channel Flow, Journal of Computational Physics, 71, pp 343-370, 1987.

Leonard, A. Energy Cascades in Large-Eddy Simulations of Turbulent Fluid Flows, Advances in Geophysics, 18, pp 237-247, 1974.

Lilly, D.K. On the Application of the Eddy Viscosity Concept in the Inertial Subrange of Turbulence, NCAR Manuscript No. 123, National Center for Atmospheric Research, Boulder, Colorado, 1966.

Madsen, P.A., Rugbjerg, M. and Warren, I.R. Subgrid Modelling in Depth Integrated Flows, Coastal Engineering Conference, 1, pp 505-511, Malaga, Spain, 1988.

Sleigh, P.A., Gaskell, P.H., Bersins, M. and Wright, N.G. (1998), An unstructured finite-volume algorithm for predicting flow in rivers and estuaries, Computers & Fluids, Vol. 27, No. 4, 479-508.

Smagorinsky, J. General Circulation Experiment with the Primitive Equations, Monthly Weather Review, 91, No. 3, pp 99-164, 1963.

Wang, J.D. Numerical Modelling of Bay Circulation, The Sea, Ocean Engineering Science, 9, Part B, Chapter 32, pp 1033-1067, 1990.

Zhao, D.H., Shen, H.W., Tabios, G.Q., Tan, W.Y. and Lai, J.S. (1994), Finite-volume 2-dimensional unsteady-flow model for river basins, Journal of Hydraulic Engineering, ASCE, 1994, 120, No. 7, 863-833.

4 Boundary Conditions

Hydrodynamic, water quality, and source boundary conditions may be specified for 2D overland models in MIKE URBAN+.

4.1 2D Boundary Conditions

Configure boundary conditions in the 2D Boundary Conditions Editor (Boundary|2D Boundary Conditions).

If the 2D domain source is 'Domain file created from MIKE URBAN+ definition', the user must pre-define open 2D boundaries in the computation grid/mesh on the Map before configuring in the 2D Boundary Conditions editor (Figure 4.1). If the 2D domain source is 'Existing domain file', then the open 2D boundaries are identified from the selected flexible mesh file, and their locations cannot be modified in MIKE URBAN+.

By default, the 2D model is initially assigned a closed boundary, wherein water will not enter or leave the domain across its perimeter. However, it is possible to locally define other types of 2D hydrodynamic boundaries in MIKE URBAN+:

- **Discharge**: To define known discharge (constant or time varying) conditions along the boundary, such as where the 2D model perimeter crosses a river or similar.
- **Water Level**: Typically applied where the 2D model perimeter crosses a larger water body, such as a lake or the sea, where the water level is known (constant or time varying).
- **Q/h Relation**: A rating curve may be used to describe varying water level at the boundary as a function of the incoming discharge. May be used when the relationship between the discharge and the water level is known for the model.
- **Free Outflow**: Boundary condition type where water can freely leave the 2D domain across its perimeter.
- **Source**: To define local sources (or sinks) within the 2D domain.
- **Closed**: Boundary condition type where water will not enter or leave the 2D domain across its perimeter.

Closed boundary conditions are maintained along domain perimeter sections where no other boundary condition types are specified.

Each boundary section must span at least 2 grid/mesh nodes, and open boundary conditions require wet boundary sections.

The available options and tabs in the 2D Boundary Conditions editor vary depending on the type of boundary condition being set-up. For every type, a

Description tab is available, wherein a free text description or notes on the boundary condition may be added.

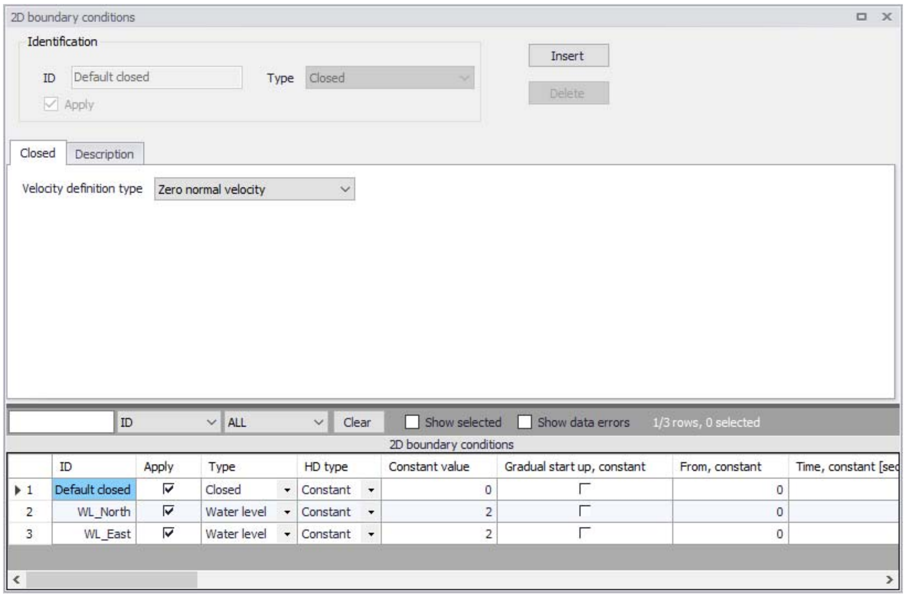


Figure 4.1 The 2D Boundary Conditions editor

4.1.1 Defining 2D boundary conditions

Example workflows for setting-up 2D boundary conditions in MIKE URBAN+ are described in the following sections.

For newly-created domain grids/meshes

This workflow is relevant for when new domain grids/meshes are being created in MIKE URBAN+ (i.e. Source = Domain file from MIKE URBAN+ definition).

1. Ensure the new domain grid/mesh has been generated e.g. following instructions in Chapter 3.1 2D Domain *(p. 14)*.
2. Define 2D model boundary sections on the Map via the Edit Features toolbox on the 2D Overland menu ribbon (Figure 4.2) or the Flooding layer editing tools toolbar on the Map (Figure 4.3.)

   Use the 'Create' tool to draw features on the Map. Click on mesh/grid nodes to locate the boundary condition. Right-click to remove the last vertex added, and double-click to finish drawing.

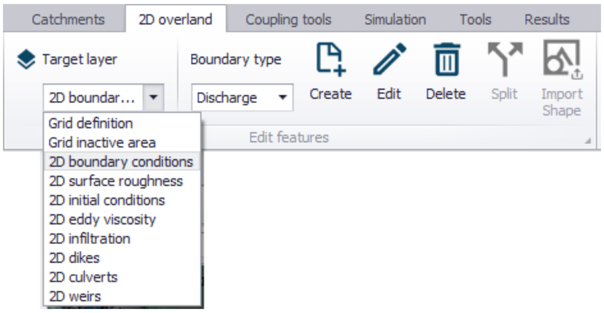


Figure 4.2 The Edit features toolbox on the 2D Overland menu ribbon

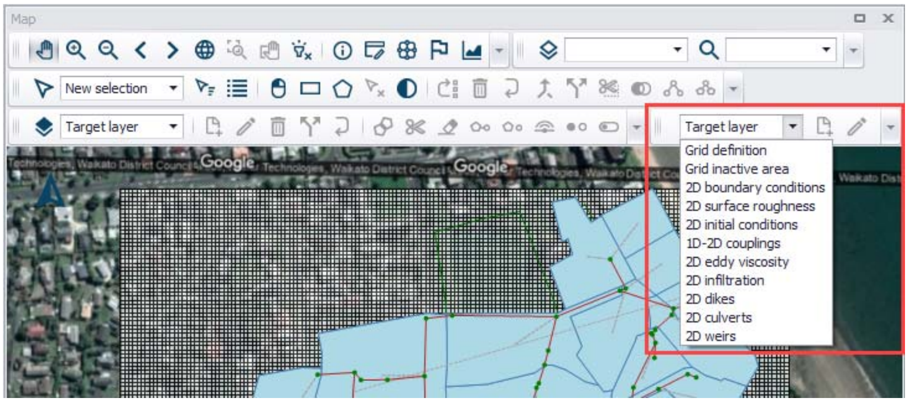


Figure 4.3 The Flooding Layer Editing Tools toolbar on the Map

3. Configure boundary conditions in the 2D Boundary Conditions Editor (Boundary|2D Boundary Conditions) (Figure 4.1). Details on the various configuration parameters for each type of boundary condition are described in succeeding sections.

For loaded existing domain grids/meshes

1. Specify the existing domain file for the 2D model in the 2D Overland editor.
2. When existing domain grids/meshes are loaded into MIKE URBAN+ (i.e. Source = Existing domain file), the program automatically detects any existing boundary definitions in the grid or mesh, and adds corresponding records to the 2D Boundary Conditions table.

   Grid files have intrinsic information on boundary sections, wherein grid perimeter cells with (Bathymetry) values less than the 'Land value' are considered open boundaries (see Figure 4.4). These information may be checked by viewing the *.DFS2 grid file in the Grid Editor.

   Viewing rectangular grids in the Grid Editor from MIKE URBAN+ is offered via the ellipsis button beside the 'File path' parameter in the 2D

Domain editor, or using the ‘Grid editor’ tool in the ‘2D Domain Editors’ toolbox on the ‘2D Overland’ menu ribbon.

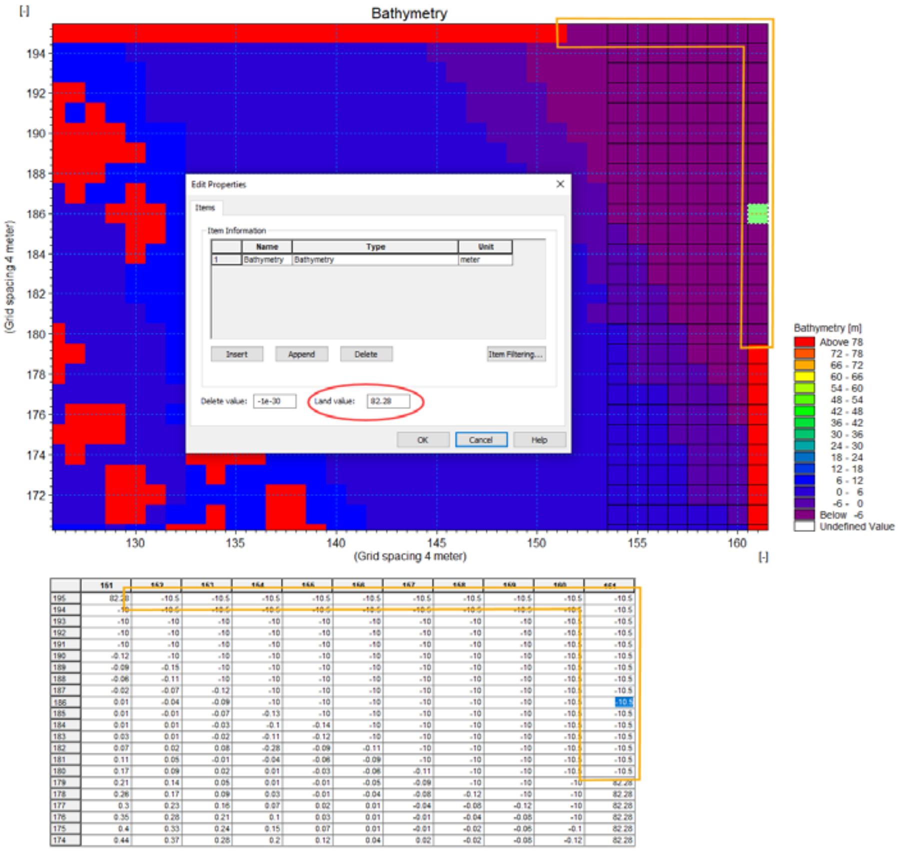


Figure 4.4 Rectangular grid file viewed in the Grid Editor showing the ‘Land value’ property (encircled in red) in the Edit Properties dialog (Edit|Items). Grid values are shown on the map as well as on the overview table, and the open boundary sections are outlined in yellow.

Mesh files also have built-in information about boundary sections; in this case via ‘Code value’ properties at element nodes (see Figure 4.5). These information may be checked by viewing the *.MESH file in the Data Viewer.

The ‘Code value’ is used to distinguish between the different boundary sections in the mesh. A value of ‘1’ is used for all closed boundaries, whereas all other open boundaries must be defined with a number >1.

Viewing meshes in the Data Viewer from MIKE URBAN+ is offered via the ellipsis button beside the ‘File path’ parameter in the 2D Domain editor, or using the ‘Data Viewer’ tool in the ‘2D Domain Editors’ toolbox on the ‘2D Overland’ menu ribbon.

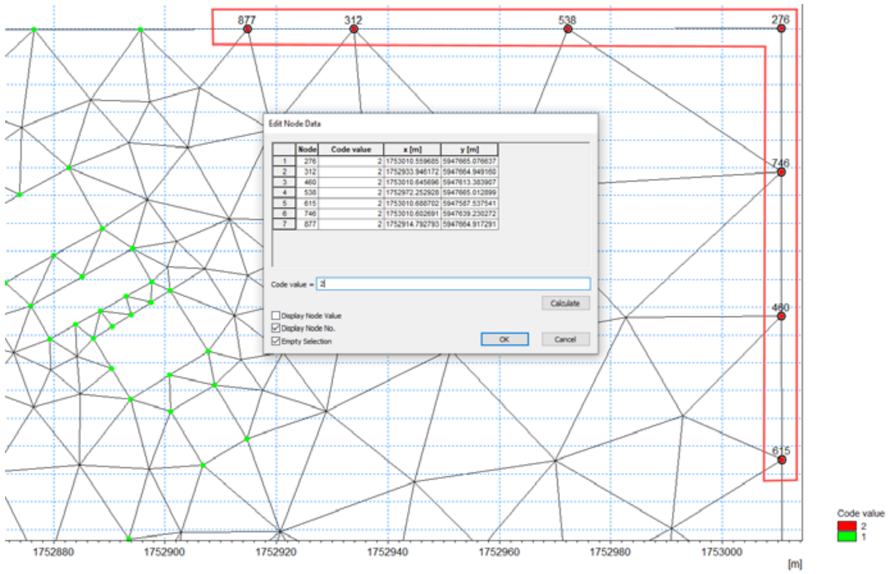


Figure 4.5 Mesh file viewed in the Data Viewer showing the 'Code value' properties for mesh perimeter nodes (outlined in red). Code values may be shown on the map by activating the corresponding data layer from the toolbar, or by selecting node points from the map.

3. Configure all the defined boundary conditions in the 2D Boundary Conditions Editor (Boundary|2D Boundary Conditions) (Figure 4.1). Details on the various configuration parameters for each type of boundary condition are described in succeeding sections.

4.1.2 Closed

A closed boundary is first assigned the 2D domain perimeter by default. A closed boundary indicates water will not enter or leave the model domain across its perimeter. Closed boundaries can be of two types:

- **Zero normal velocity**. The default closed boundary type for 2D models in MIKE URBAN+. For this type, the free-slip boundary condition is assumed, wherein the normal velocity component is zero.
- **Zero velocity**. For this type, the no-slip boundary condition is assumed where both the normal and tangential velocity components are zero.

4.1.3 Discharge

Discharge boundary conditions can be constant in time, or vary as specified in a *.DFS0 timeseries file:

- **Constant**

- **Varying in time.** Requires a *.DFS0 timeseries file of discharge that covers the entire simulation period. The time step of the input data file does not have to be the same as the time step of the hydrodynamic simulation.

An option for controlling the application of the boundary condition at the start of the simulation is offered via the '**Gradual start up**' option. One may specify a soft start interval during which boundary values are increased from a specified reference value to the specified boundary value in order to avoid shock waves being generated in the model.

Discharge values shall either be positive or negative depending on the desired direction of the boundary (i.e. inflow or outflow).

Interpolation

For time-varying discharge boundaries, the interpolation method for determining values between available time step values in the specified file may be specified as Linear or Piecewise cubic in the 'Interpolation' tab. Also see Chapter 4.1.8 Interpolation Types *(p. 82)* for more details.

The distribution of the total flow in the individual grid points along the boundary is calculated relative to the depth. The discharge is distributed as in a uniform flow field with the Manning resistance law applied, i.e. is relative to $h^{5/3}$, where h is the depth.

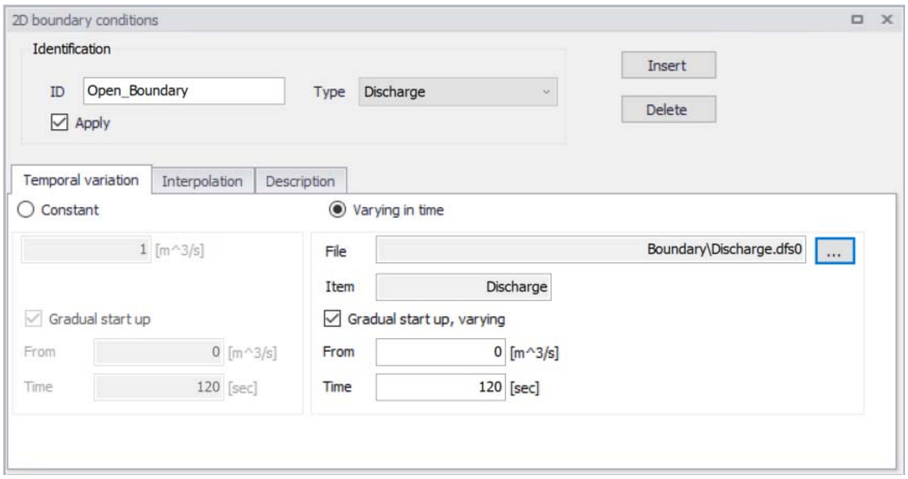


Figure 4.6 Set-up parameters for Discharge 2D Boundary Conditions

4.1.4 Water Level

Water level boundaries may be constant, time-varying, or temporally- and spatially-varying along the boundary line:

- **Constant**

- **Varying in time**. Requires a *.DFS0 timeseries file of water level that covers the entire simulation period. The time step of the input data file does not have to be the same as the time step of the hydrodynamic simulation.
- **Varying in domain and time**. Requires a *.DFS1 profile timeseries file.

Interpolation

For time-varying and space-varying water level boundaries, the interpolation method for determining values between available time step values in the specified file may be specified as Linear or Piecewise cubic in the 'Interpolation' tab.

For space-varying water level boundaries, mapping input data to the boundary section may also be specified as Clockwise or Counter-clockwise in the 'Interpolation' tab.

More details on options for interpolating and mapping the input data file to the boundary section are found under Chapter 4.1.8 Interpolation Types *(p. 82)*.

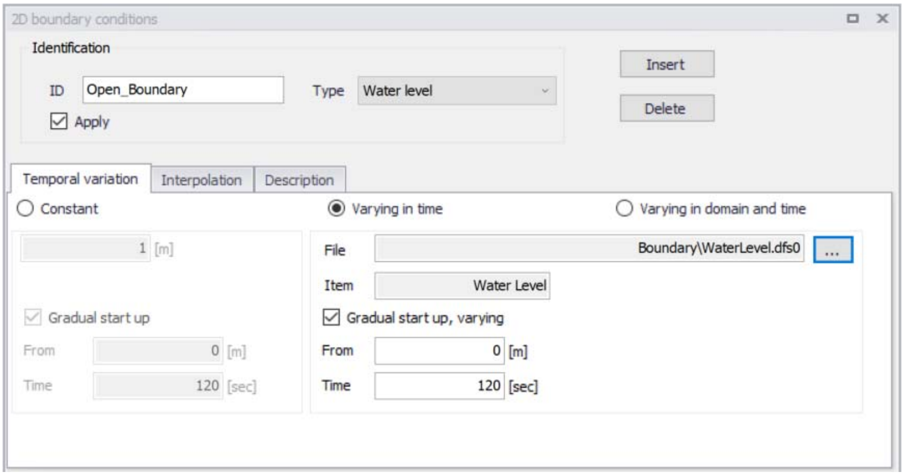


Figure 4.7 Parameters for Water Level 2D Boundary Conditions

4.1.5 Q/h Relation

This type of boundary is only recommended for downstream boundaries where water flows out of the 2D model.

The rating curve comprises of water level and discharge value pairs. The water level value is determined from the rating curve table using the appropriate discharge in the adjacent cells.

Define a table containing the relation between discharge and water level in the secondary table on the Q/h Relation tab page on the editor (Figure 4.8).

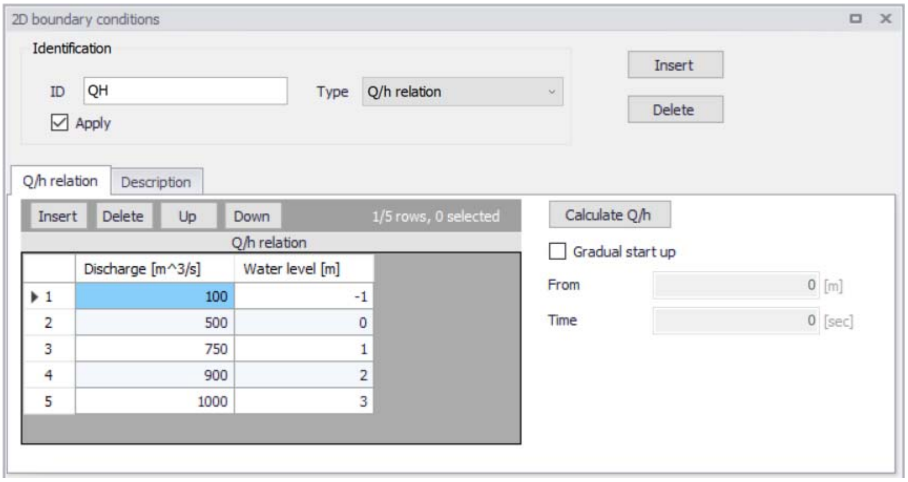


Figure 4.8 The rating curve secondary table on the Q/h Tab page on the 2D boundary conditions editor

Calculate Q/h

The Q/h relation rating curve may be automatically generated via the ‘Calculate Q/h’ button.

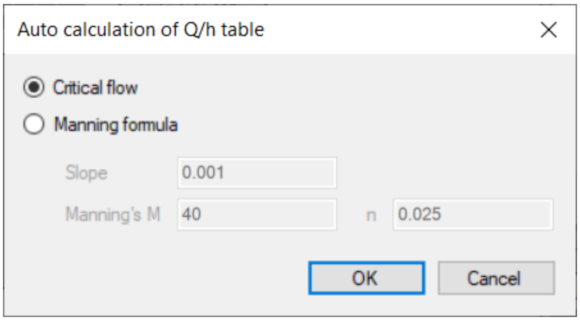


Figure 4.9 The dialog for configuring the auto-calculation of Q/h table values

The Q/h relation is computed using the topography in the mesh/grid along this boundary. Thus, topography values must have been interpolated for the mesh/grid prior the use of this tool. (Figure 4.10).

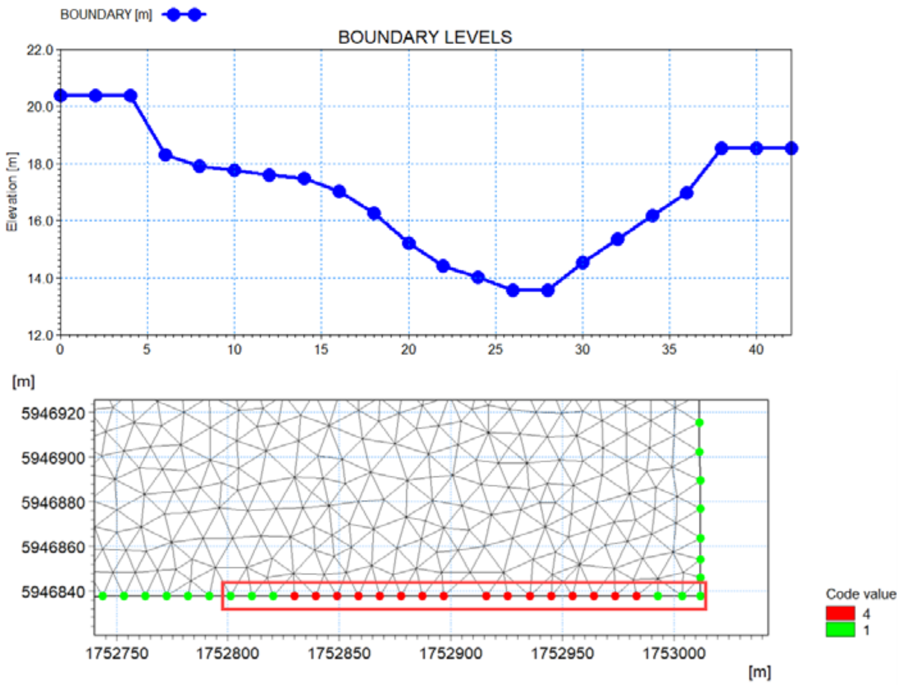


Figure 4.10 Illustration of topographical cross section (top) along an e.g. mesh boundary (bottom).

Discharge values are computed for various water levels, either using the ‘Critical Flow’ or the ‘Manning Formula’ as specified in the ‘Auto calculation of Q/h table’ dialog (Figure 4.9).

If the latter is chosen the bed slope and Manning’s "n" or "M" must be speci-fied.

In case the critical flow formula is used, Q is calculated from:

$$Q(h) = A(h)\sqrt{g\frac{A(h)}{W(h)}}$$

In case of uniform flow by Manning’s Formula, Q is calculated from:

$$Q(h) = Conv(h)\sqrt{I}$$

Where:

$Q(h)$ = Level dependent discharge

$A(h)$ = Level dependent area (from the cross section processed data)

$W(h)$ = Level dependent width (from the cross section processed data)

I = Bed slope

$Conv(h)$ = Level dependent conveyance calculated as a function of the resistance type defined for the cross section

$$Conv(h) = MA(h)R^{2/3}$$

Where:

$M = \frac{1}{n}$ = Manning number defined in the 'Auto calculation of Q/h table' dialog

Gradual Start Up

A soft start option for applying the boundary condition is offered via the '**Gradual start up**' option. One may specify a soft start interval during which boundary values are increased from a specified reference value to the specified boundary value to avoid shock waves in the model.

4.1.6 Free Outflow

The free outflow condition is typically used at downstream boundaries where the water is flowing out of the model. This type of boundary does not require additional input in the editor.

4.1.7 Source

Source boundaries are used to define local sources (or sinks) of water inland within the 2D domain (as opposed to along the domain perimeter).

Sources may be specified via the 2D Boundary Conditions editor by clicking on the 'Insert' button, or on the Map using the 'Edit features' toolbox on the 2D Overland menu ribbon or the Flooding layer editing tools toolbar. Point sources may be of type:

- **Single point source**
- **Single distributed source**
- **Connected point source**

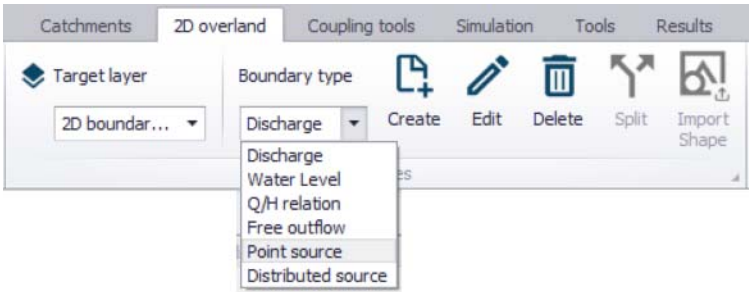


Figure 4.11 Adding Source boundaries on the Map using the Edit features toolbox on the 2D Overland menu ribbon

On the 2D Boundary Conditions editor, the Location and Temporal Variation for sources should be defined. The editor also has a Description tab page for any notes about each source boundary item.

Location

Single Point Source

Define the x- and y-coordinates of the source on the Location tab page. You may also use the arrow button to graphically select a location on the Map. The x- and y-coordinates in the editor are updated accordingly.

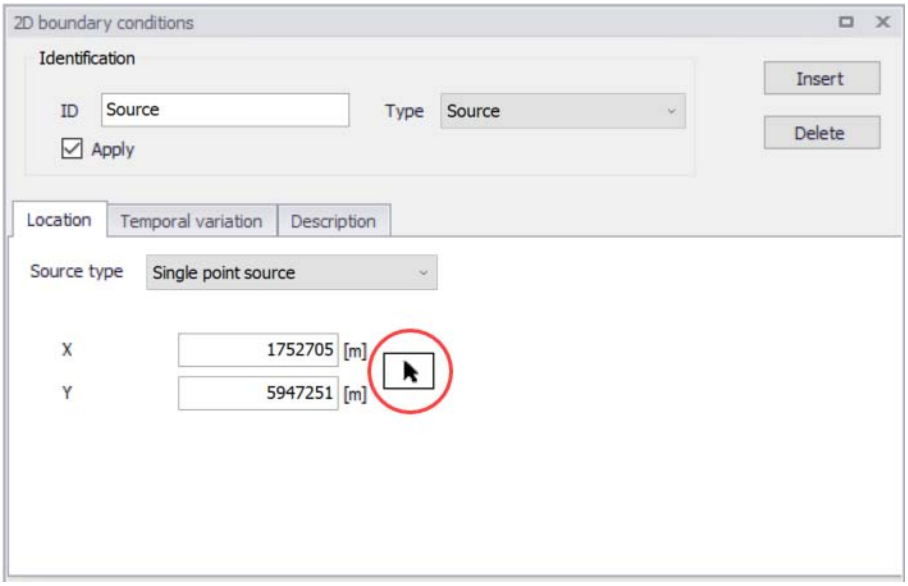


Figure 4.12 Locate the 2D point source graphically on the Map using the arrow button

Single Distributed Source

A distributed source is characterized by a polygon area in MIKE URBAN+. For this type of source, the specified discharge value is distributed (i.e. weighed distribution based on element area) in all mesh/grid elements within the polygon.

Define the x-and y-coordinates for the polygon in the secondary table on the Location tab page (Figure 4.13). Alternatively, draw the distributed source polygon on the Map via the arrow button beside the secondary table.

The Edit Features toolbox on the 2D Overland menu ribbon, or the Flooding Layer Editing Tools map toolbar may also be used to create or edit distributed source polygons on the Map.

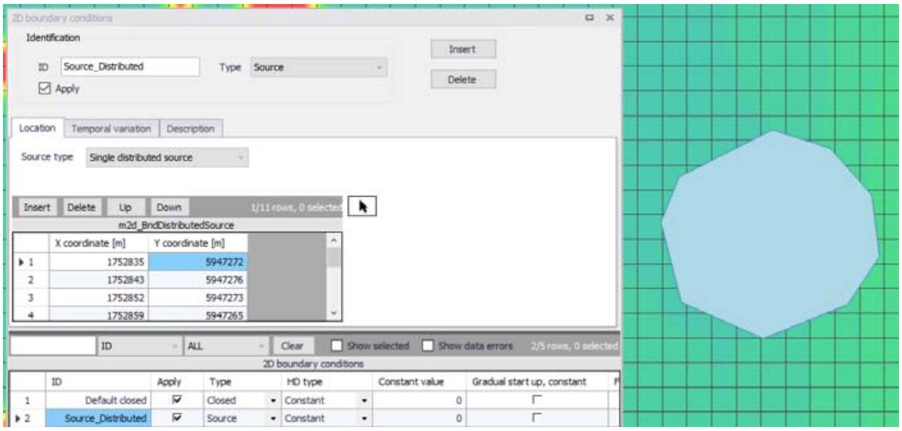


Figure 4.13 Distributed Source location on the 2D Boundary Conditions editor and the Map

Connected Point Source

Connected sources are defined with two source locations (i.e. First Source and Second Source). For this option the source contribution to both the continuity equation and the momentum equations is taken into account.

Define the coordinates for the First Source and Second Source points on the Location tab page (Figure 4.14). Alternatively, locate the points on the Map via the arrow buttons beside the input boxes.

The Edit Features toolbox on the 2D Overland menu ribbon, or the Flooding Layer Editing Tools map toolbar may also be used to create or edit connected point source segments on the Map.

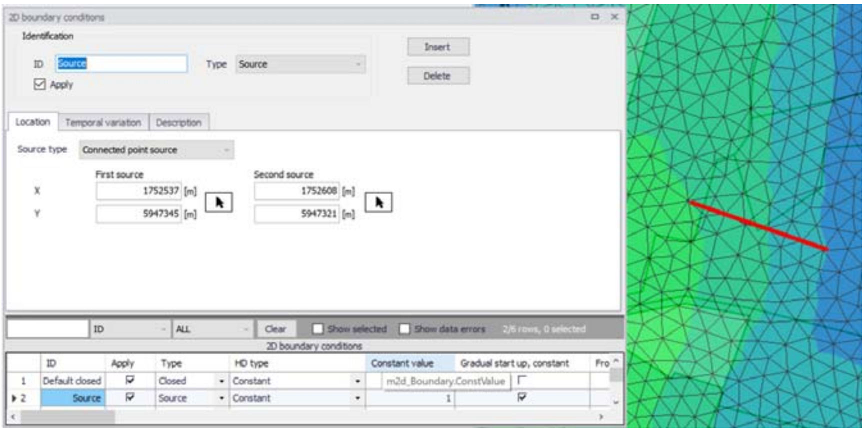


Figure 4.14 Define locations for the First Source and Second Source points for Connected Point Sources on the 2D Boundary Conditions editor and the Map

The magnitude of the First Source is the specified magnitude in the ‘Temporal Variation’ tab and the magnitude of the Second Source is the same value but with opposite sign. Therefore, if the objective is to use a connected source from the First Source point to the Second Source point, the magnitude specified in the ‘Temporal Variation’ tab must have a negative sign.

The velocity by which the water is discharged into the domain must be specified. Note that the contribution to the momentum equation is only taken into account when the water is discharged into the ambient water (i.e. the magnitude at the Second Source point is positive).

Temporal Variation

Define whether the point source is:

- **Constant**
- **Varying in time**

For time-varying sources, specify a timeseries file (*.DFS0) containing source information (i.e. discharge and velocity components, if needed). The data must cover the complete simulation period, but the time step may differ from the time step of the hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

Activate the ‘**Apply source velocity**’ if desired, for which the source contributions to the continuity equation as well as the momentum equation are both taken into account. In this case, the u- and v-velocity components need to be specified together with the source magnitude (i.e. Discharge). Otherwise, the source has no velocity component, for which only the source contribution to the continuity equation is considered. Note that connected point sources shall always have associated velocity components defined, and thus the ‘Apply source velocity’ option for these types of boundaries is disabled.

See the section on Source Types *(p. 82)* for more details.

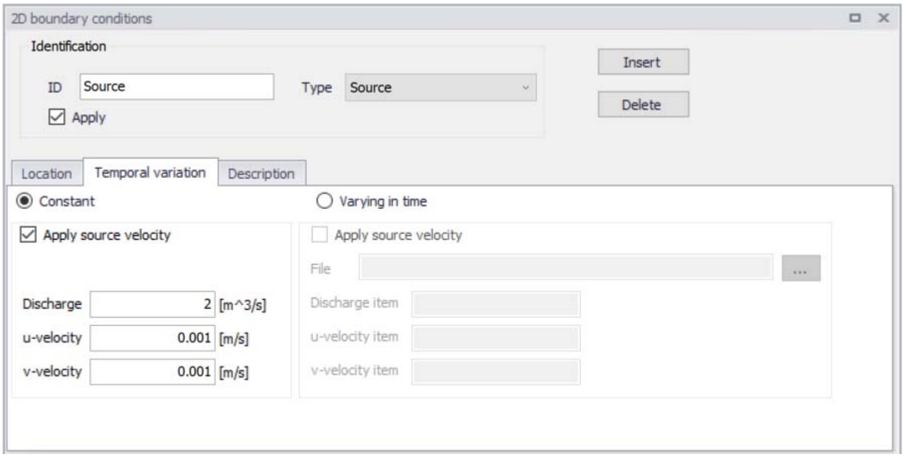


Figure 4.15 Temporal Variation tab for Source 2D boundary conditions

Source Types

For sources with no velocity component defined, only the source contribution to the continuity equation is taken into account. For this option, specify only the magnitude of the source. If the magnitude of the source is positive, water is discharged into the ambient water. If the magnitude is negative, water is discharged out of the ambient water.

For sources with velocity components defined, both the source contribution to the continuity equation and the momentum equations is taken into account. For this option, specify both the magnitude of the source and the velocity by which the water is discharged into the ambient water. Note that the contribution to the momentum equation is only taken into account when the magnitude of the source is positive (water is discharged into the ambient water).

4.1.8 Interpolation Types

For time-varying discharge and water level boundary types, the time step of the related timeseries data file does not need to match the hydrodynamic calculation time step. Thus, value interpolation may be required during computations. The interpolation method may be set as:

- **Linear**. Values obtained from a linear function (straight line) between 2 known value points.
- **Piecewise cubic**. Values obtained from a cubic polynomial approximation over sub-intervals.

In the cases with values varying along the boundary (i.e. Water level varying in domain and time), two methods of mapping from the input data file to the boundary section are available:

- **Counter-clockwise definition**. The first and last points of the line (i.e. *.DFS1) are mapped to the first and the last nodes along the boundary section, respectively, and the intermediate boundary values are found by linear interpolation.
- **Clockwise definition**. The last and first points of the line are mapped to the first and the last nodes along the boundary section, respectively, and the intermediate boundary values are found by linear interpolation.

Define these time- and space-wise interpolation settings in the Interpolation tab page of the 2D Boundary Conditions editor. These options are available for time-varying Discharge and Water Level boundaries.

These settings may also be modified for a 2D boundary condition under the 'Interpolation type in time' and 'Spatial order along boundary' columns in the overview table at the bottom of the editor (Figure 4.16).

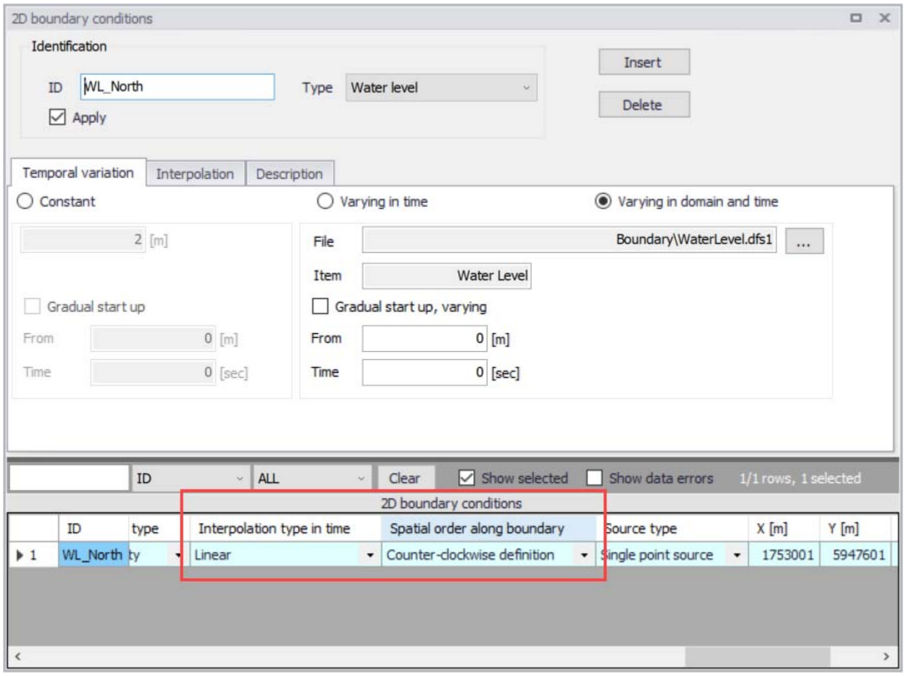


Figure 4.16 The 'Interpolation type in time' and 'Spatial order along boundary' attributes in the 2D boundary conditions overview table

4.2 2D WQ Boundary Conditions

MIKE URBAN+ 2D Overland offers options for simulating transport of materials (e.g. pollutants) via the 'Water Quality (AD)' module.

Options for setting-up 2D WQ boundary conditions (Figure 4.17) are made available when both the 'Hydrodynamic (HD)' and 'Water Quality (AD)' 2D Overland modules are activated.

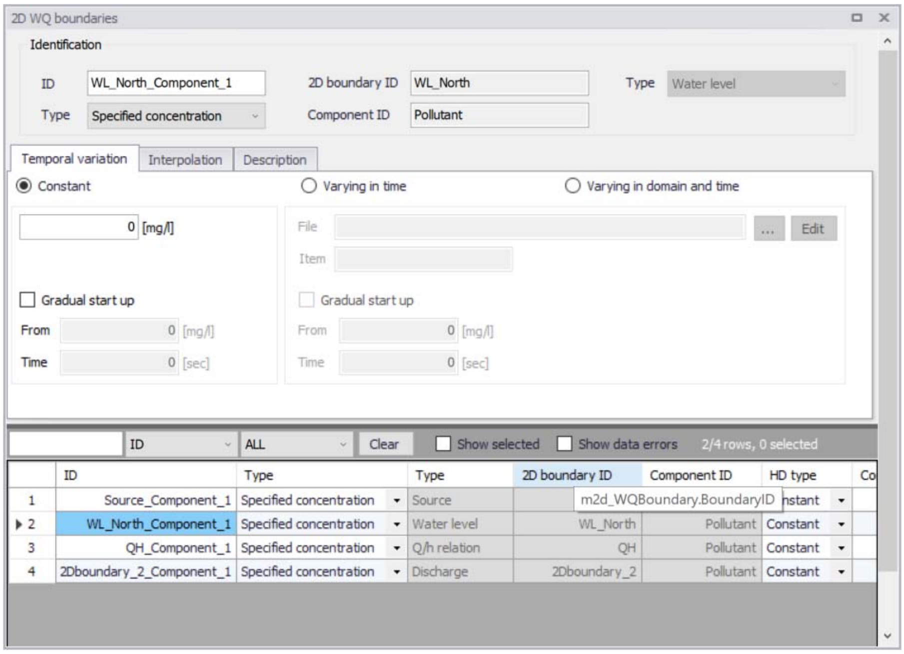


Figure 4.17 The 2D WQ Boundaries editor is automatically filled when water quality components have been defined in the model with open 2D boundary conditions that have been set-up.

Water quality AD (Advection-Dispersion) computations are based on flow conditions from hydrodynamic calculations. As such, water quality boundary conditions are specified in association with (i.e. attached to) **open** 2D hydrodynamic boundary conditions.

That is, hydrodynamic boundary conditions must first be set-up, after which water quality conditions are attached to them. Also, water quality components to be simulated must be pre-defined in the model through the WQ Components editor.

The main steps involved in defining 2D WQ Boundary Conditions are:

1. Define water quality components to be simulated in the WQ Components editor.
2. Set-up hydrodynamic boundary conditions in the 2D Boundary Conditions editor. See Chapter 4.1 2D Boundary Conditions *(p. 69)* for details on setting-up hydrodynamic boundary conditions.
3. Access the 2D WQ Boundaries editor, and configure the automatically-detected open HD boundaries for associated water quality characteristics. Details on the various parameters for configuring 2D WQ boundary conditions are described in succeeding sections.

4.2.1 Identification

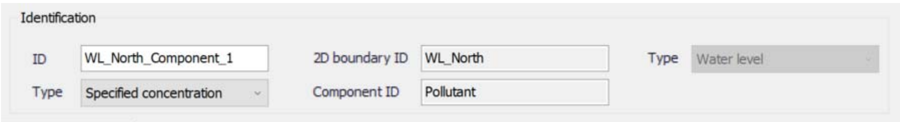


Figure 4.18 The Identification group box on the 2D WQ Boundaries editor

This group box contains general information on each 2D WQ boundary condition shown in the overview table at the bottom of the editor:

- **ID**. Unique ID for the 2D WQ boundary condition. The default ID is '2DBoundaryID_ComponentID', where 2DBoundaryID and ComponentID are the IDs for the 2D boundary condition and the WQ component related to the active row from the overview table, respectively.
- **Type** (WQ boundary)**.** 2D WQ boundary types vary depending on the associated HD boundary. More details on 2D WQ boundary types are found in the succeeding section (See 2D WQ Boundary Types, page 85).
- **2D boundary ID.** The ID for the related 2D boundary condition. It is non-editable.
- **Component ID**. The unique ID for the related WQ component. It is non-editable.
- **Type** (HD boundary)**.** The type of related 2D HD boundary condition (e.g. Discharge, Water level, etc.). It is program- detected and non-editable.

2D WQ Boundary Types

Possible 2D WQ boundary types vary depending on the associated HD boundary.

For most HD boundary types (i.e. Discharge, Water Level, Q/h Relation, and Free Outflow), WQ boundaries may be:

- **Specified concentration**. With this definition, the boundary concentration is the specified concentration for inflows into the domain (i.e. water is discharged into the domain).

  Otherwise, the boundary concentration is the concentration at the boundary if water is discharging out of the domain.

- **Zero normal gradient**. For this option, the concentration at the boundary is assumed to be identical to the concentration at the adjacent interior cell/element.

Source WQ Boundaries

Point sources of dissolved components are important in many applications such as release of nutrients from rivers, and intakes and outlets from cooling water or desalination plants. Associated water quality properties for Source boundaries may be defined as:

- **Specified concentration**. With this definition, the source concentration is the specified concentration if the magnitude of the source is positive (i.e. water is discharged into the ambient water).

  Otherwise, the source concentration is the concentration at the source point if the magnitude of the source is negative (i.e. water is discharged out of the ambient water). This option is pertinent to e.g. river outlets or other sources where the concentration is independent of the surrounding water.

- **Excess concentration**. For this option, the source concentration is the sum of the specified excess concentration and concentration at a point in the model if the magnitude of the source is positive (i.e. water is discharged into the ambient water).

  The source concentration is the concentration at the source point if the magnitude of the source is negative (i.e. water is discharged out of the ambient water). This type can be used to describe e.g. a heat exchange or other processes where the temperature (heat) or salinity is added to the water by a diffusion process.

The source flux is calculated as the product $Q_{source} \times C_{source}$, where Q_{source} is the magnitude of the source and C_{source} is the component concentration of the source.

4.2.2 Temporal Variation

Temporal variation options are offered via the Temporal Variation tab on the 2D WQ Boundaries editor (Figure 4.19). Temporal variation is relevant only for concentration type WQ boundaries (i.e. 'Specified concentration' type).

The parameters offered on the tab page vary according to the 2D (HD) boundary type associated with the WQ boundary, as indicated in the 'Identification' group box.

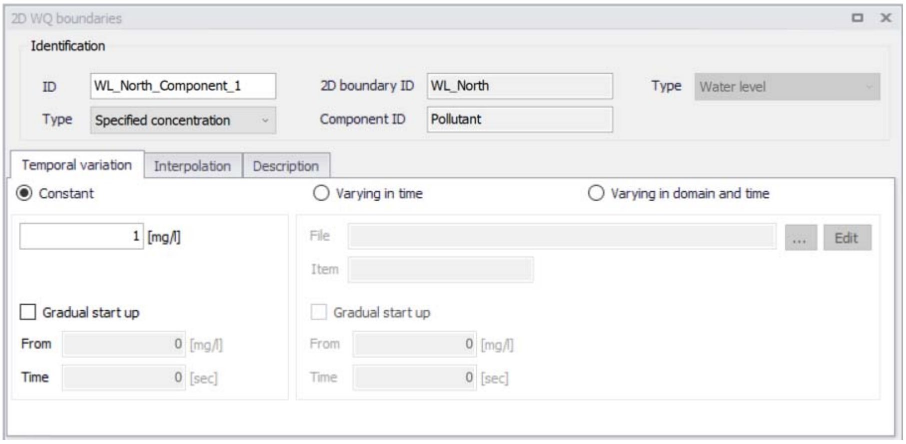


Figure 4.19 The Temporal Variation tab page on the 2D WQ Boundaries editor

For most types of related HD boundaries (i.e. Discharge, Water Level, Q/h Relation, and Free Outflow), the temporal variation of 'Specified concentration' type WQ boundaries may be:

- **Constant** (in time and along boundary)
- **Varying in time** (and uniform along boundary). Requires a *.DFS0 data file containing the component concentration (in component unit) in the setup.

  The data must cover the whole simulation period, although the data time step does not have to be the same as the simulation time step. This may require value interpolation during computations, for which an interpolation method may be specified on the Interpolation tab (See Interpolation, page 88).

- **Varying in domain and time**. This option requires a *.DFS1 data file containing information on component concentration (in component unit). and spatial variation.

  The data must cover the complete simulation period, but the time step of the input data file does not have to match he simulation time step. This may require value interpolation during computations, for which an interpolation method may be specified on the Interpolation tab (See Interpolation, page 88).

  In addition, spatial mapping of the *.DFS1 data file values to the boundary section must be specified on the Interpolation tab page.

Gradual start up

You can specify a soft start interval during which boundary values are increased from a specified reference value to the specified boundary value in order to avoid shock waves being generated in the model. The increase follows a linear variation.

To use this option, tick on the ‘**Gradual start up**' check box on the tab page.

- **From.** The component reference value for the soft start. The unit of the component is shown on the right.
- **Time**: The desired soft start duration.

Source WQ Boundaries

Temporal variation options for Source water quality boundaries differ slightly from those for other types (Figure 4.20).

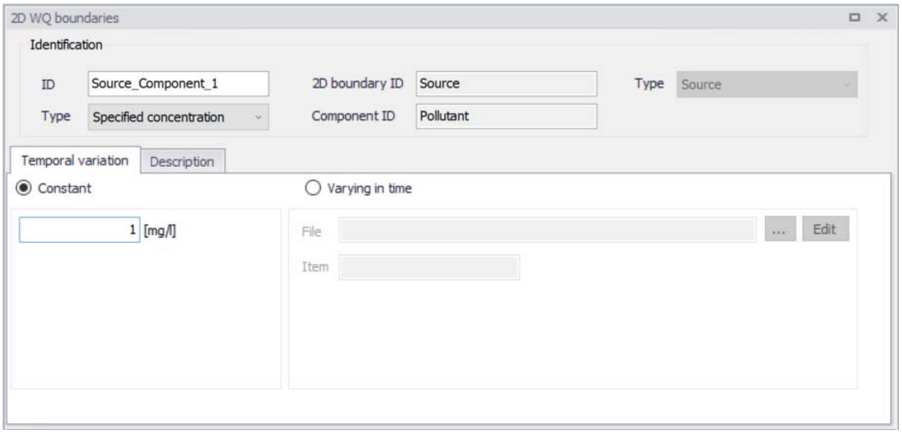


Figure 4.20 Temporal variation options for 2D Source water quality boundary conditions

Source WQ boundaries may be:

- **Constant**
- **Varying in time**. This option requires a timeseries (*.DFS0) data file containing concentration values (in concentration units) for the source.

  The data must cover the complete simulation period, but the time step of the input data file does not have to be the same as the time step of the computation. A linear interpolation of timeseries concentration values is applied if the time steps differ.

Note: Point sources are entered into elements in a way that the inflowing mass of the component is initially distributed over the element where the source is located. Therefore, concentration simulation results are usually lower than the specified source concentration.

4.2.3 Interpolation

Interpolation options are offered for 2D WQ boundaries expressed as concentrations (i.e. ‘Specified concentration’ type) except for 'Source' 2D boundaries.

Time- and space-wise interpolation settings may be defined in the Interpolation tab page of the 2D WQ Boundaries editor (Figure 4.21).

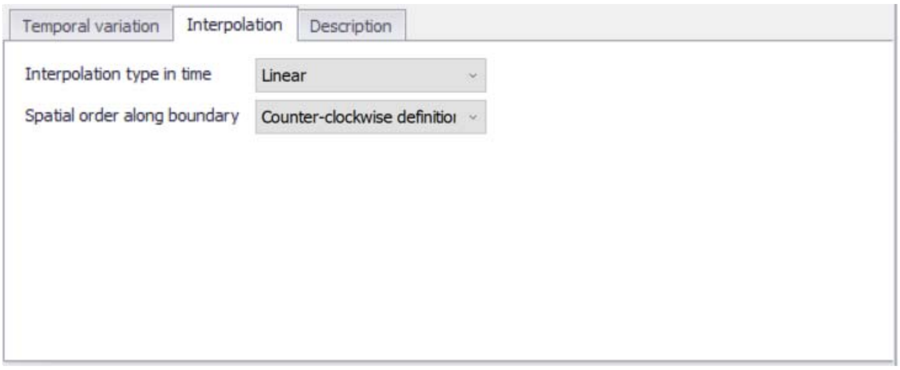


Figure 4.21 The Interpolation tab page on the 2D WQ Boundaries editor

The 'Interpolation' tab page offers options for the following:

- **Interpolation type in time**. For time-varying boundary types, the time step of the related timeseries (*.DFS0) file does not need to match the hydrodynamic calculation time step. Thus, value interpolation may be required during computations. Time-wise interpolation options are:
  - **Linear**. Values obtained from a linear function (straight line) between 2 known value points.
  - **Piecewise cubic**. Values obtained from a cubic polynomial approximation over sub-intervals.

- **Spatial order along boundary**. In cases where values vary along the boundary (e.g. Varying in domain and time), two methods of mapping the input data file (i.e. *.DFS1) to the boundary section are available:
  - **Counter-clockwise definition**. The first and last points of the line (i.e. *.DFS1) are mapped to the first and the last nodes along the boundary section, respectively, and the intermediate boundary values are found by linear interpolation.
  - **Clockwise definition**. The last and first points of the line are mapped to the first and the last nodes along the boundary section, respectively, and the intermediate boundary values are found by linear interpolation.

4.2.4 Description

Annotations for each 2D WQ boundary item may be added in the Description tab page.

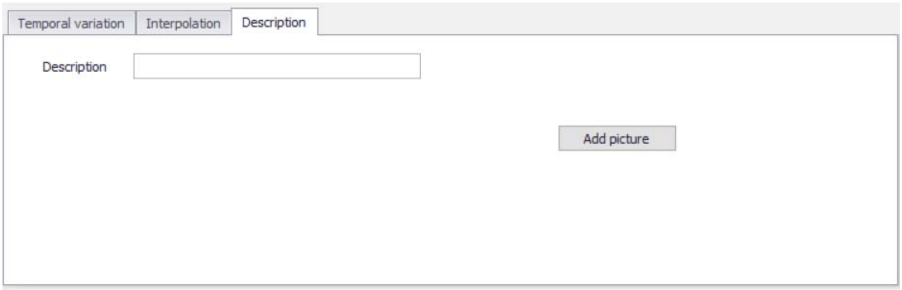


Figure 4.22 The Description tab page in the 2D WQ Boundaries editor

4.3 2D Precipitation and Evaporation

MIKE URBAN+ 2D Overland offers options for specifying ambient conditions for precipitation as well as evaporation for the 2D model domain. This is relevant for applications where the direct introduction of rainfall on the 2D model is important.

These options are available on the 2D Precipitation and Evaporation editor when the 2D Overland Hydrodynamic (HD) module is active (Figure 4.23).

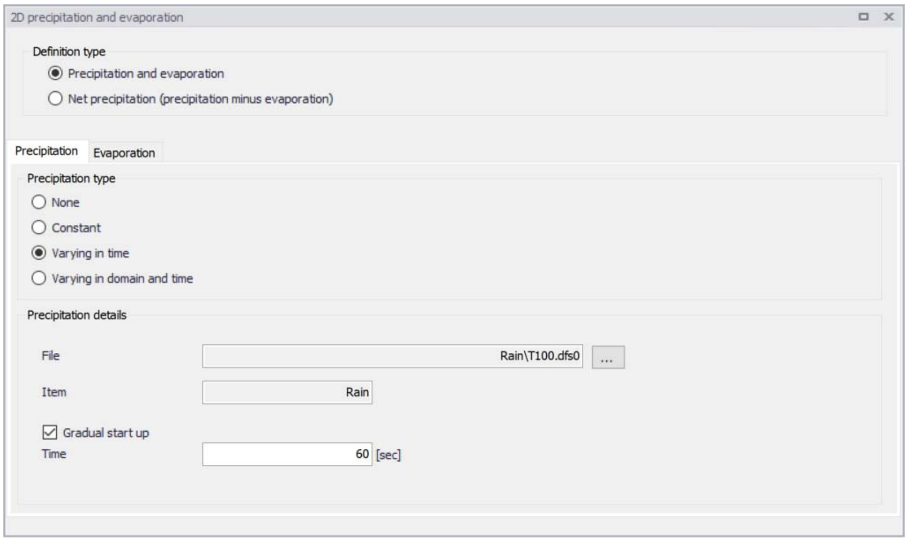


Figure 4.23 The 2D Precipitation and Evaporation editor

Precipitation and evaporation can be introduced to the 2D model as:

- **Precipitation and evaporation.** Requires definition of both precipitation and evaporation conditions in the setup. Different combinations of precipitation and evaporation conditions may be specified. Defined precipitation and evaporation rates must be positive.

- **Net precipitation**. Net precipitation is the precipitation minus evaporation. Thus, for this option, evaporation will occur if negative rates are specified.

4.3.1 Precipitation

The Precipitation tab offers various options for defining precipitation or net precipitation input types for the 2D overland model.

Precipitation Type

Available precipitation type options are:

- **None**
- **Constant**. Precipitation with a constant intensity is applied uniformly over the 2D domain.
- **Varying in time.** Time-varying precipitation applied uniformly over the 2D domain.
- **Varying in domain and time**. Time- and space-varying precipitation rates over the 2D domain.

Depending on the selected Precipitation Type, various parameters need to be defined under the Precipitation Details group box. These parameters are described in the succeeding sections below.

Precipitation Details

For Constant precipitation, the Precipitation Intensity should be specified.

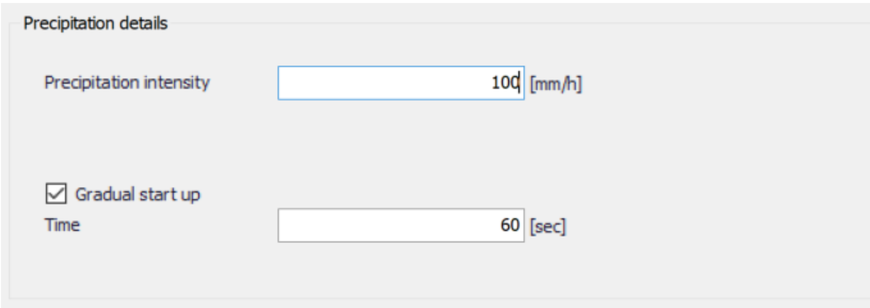


Figure 4.24 Define Precipitation Intensity for Constant Precipitation type.

Varying in time

For time-varying precipitation input, a *.DFS0 time series file containing Precipitation Rate data is required. Browse to the file location via the ellipsis button beside the 'File' input box (Figure 4.25).

An option for creating a time series file is also available using the ‘Create new timeseries file’ option. This will launch the Time Series editor, with which a new *.DFS0 file may be created. Also, the program requires a ‘Precipitation Rate’ item type for the time series data (Figure 4.26). This property may be set using the Time Series editor, which may be accessed from via the ellipsis button with the ‘Edit timeseries in TS editor’ option.

The data must cover the whole simulation period, but its time step need not be the same as the hydrodynamic simulation time step. A linear interpolation of values will be applied if the time steps differ.

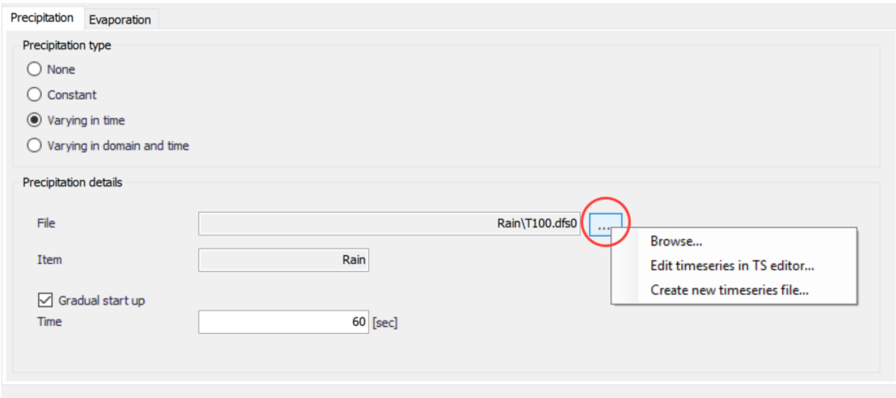


Figure 4.25 Options offered via the ellipsis button in relation to specification of precipitation time series data file

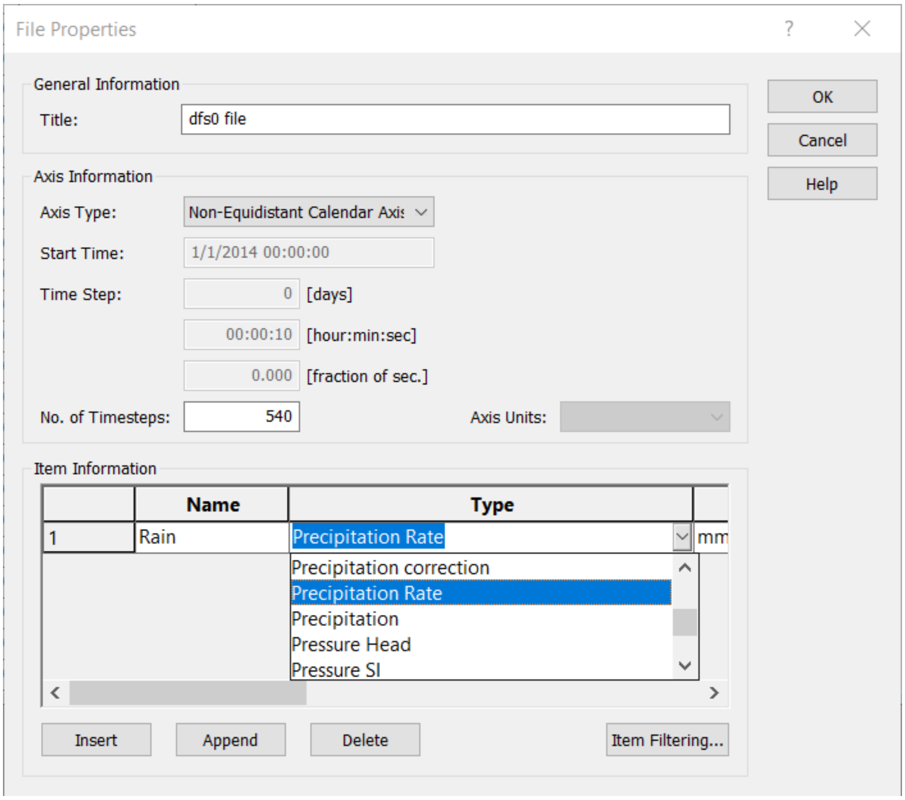


Figure 4.26 The item type for precipitation input data files must be 'Precipitation Rate'. Edit the item type in the Time Series editor via the File Properties dialog (Edit| Properties).

Varying in domain and time

This option requires a temporally- and spatially-varying data file containing Precipitation Rate information. The file may be a 2D *.DFSU unstructured data file or a 2D *.DFS2 grid data file. Browse to the file location via the ellipsis button beside the 'File' input box (Figure 4.27).

An option for creating a 2D data file is also available using the 'Create new grid file' option. This will launch the Grid Series editor, with which a new *.DFS2 file may be created.

Also, the program requires a 'Precipitation Rate' item type for the grid series data. This property may be set using the Grid Series editor accessed via the ellipsis button option 'Edit grid in Grid editor' (Figure 4.28). If the loaded data is a *.DFSU file, this option will open the loaded data file in the Data Viewer.

The data extent must cover the 2D overland model domain. If a *.DFSU file is used, piecewise constant interpolation is used to map the data to the domain. If a *.DFS2 file is used, bilinear interpolation is applied to map the data to the domain.

The data must cover the complete simulation period, although the input time step does not need to be the same as the hydrodynamic simulation time step. A linear interpolation will be applied if the time steps differ.

Figure 4.27 2D grid or mesh files containing time- and domain-varying precipitation rates data must be specified

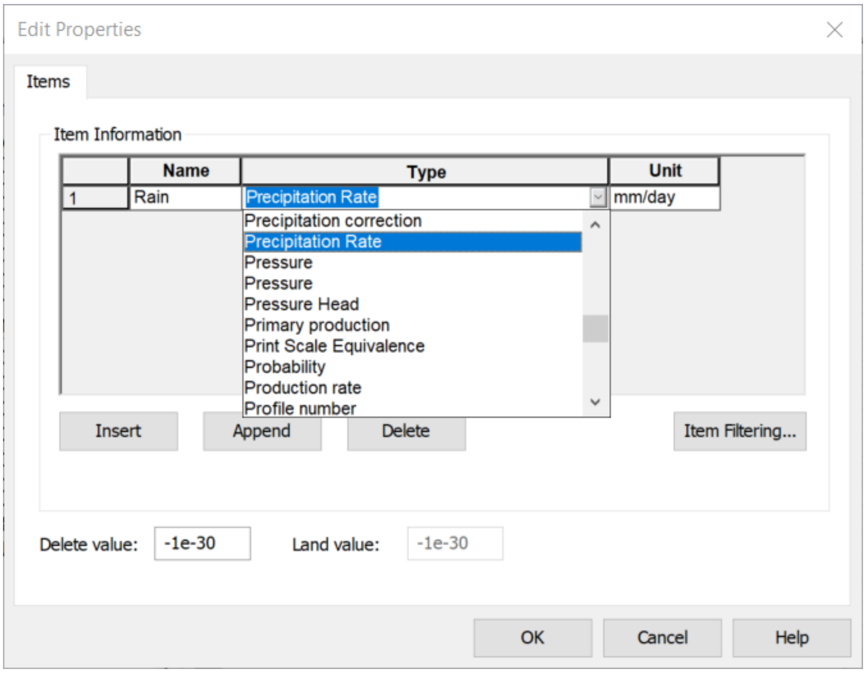


Figure 4.28 Edit the grid item type in the Grid Series editor via the Edit Properties dialog (Edit| Items).

Gradual start up

A soft start may be applied using the 'Gradual start up' check box on the tab page. With this option, the precipitation rate is increased linearly from 0 to the specified precipitation rate values. The soft start time interval must be specified.

4.3.2 Evaporation

Choosing to define both precipitation and evaporation for the 2D overland model, the Evaporation tab page is made available for specifying evaporation-related parameters (Figure 4.29).

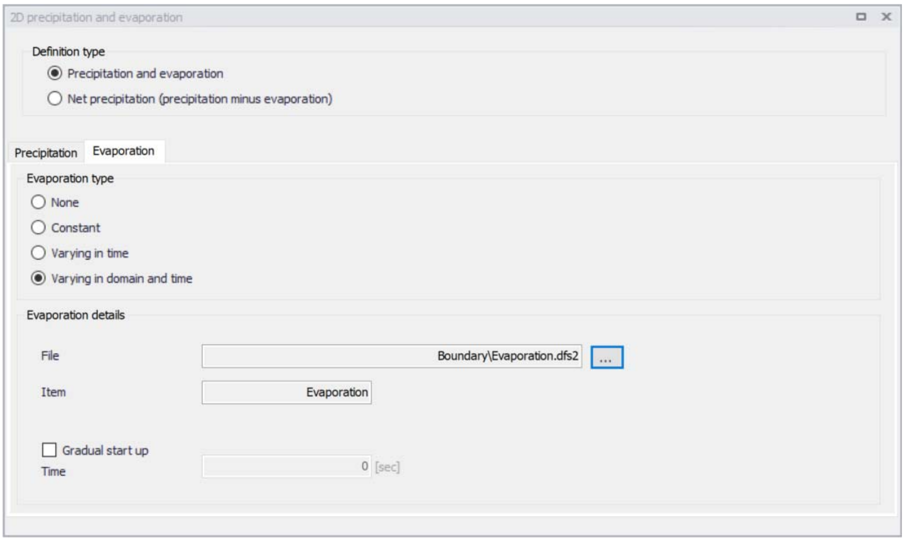


Figure 4.29 Contents of the Evaporation tab page on the 2D Precipitation and Evaporation editor

The Evaporation tab page is structured very similarly as the Precipitation tab. It offers various options for defining evaporation input types for the 2D overland model.

Evaporation Type

Available evaporation type options are:

- **None**
- **Constant**. Evaporation at a constant rate is applied uniformly over the 2D domain.
- **Varying in time**. Time-varying evaporation rates applied uniformly over the 2D domain.

- **Varying in domain and time**. Time- and space-varying evaporation rates over the 2D domain.

Depending on the selected Evaporation Type, various parameters need to be defined under the Evaporation Details section.

Evaporation Details

For Constant evaporation, the evaporation rate should be specified.

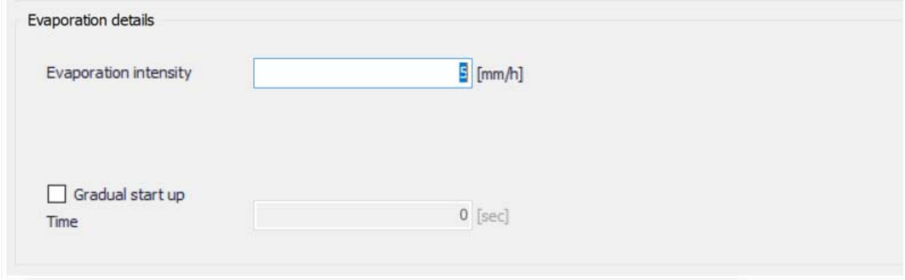


Figure 4.30 Define the constant evaporation rate to use in the 2D model

Varying in time

For time-varying evaporation, a *.DFS0 time series file containing Evaporation Rate data is required. Browse to the file location via the ellipsis button beside the 'File' input box (Figure 4.31).

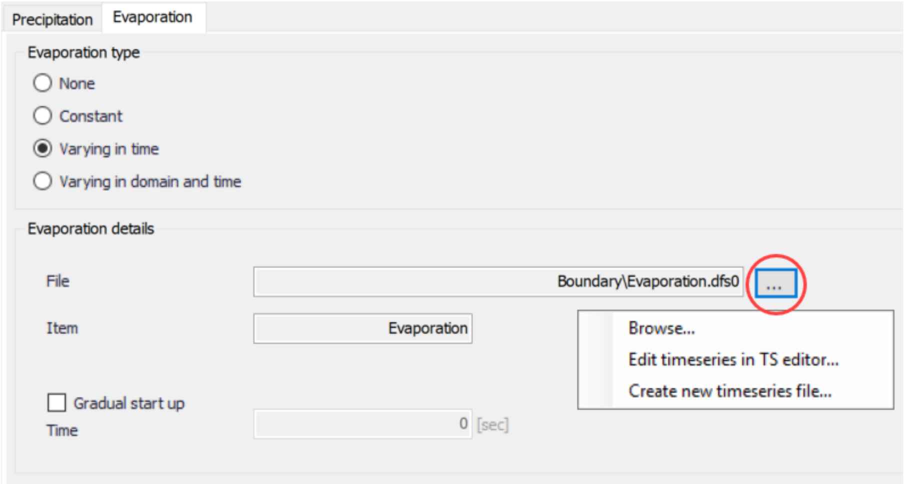


Figure 4.31 Options offered via the ellipsis button in relation to specification of evaporation time series data file

An option for creating a time series file is also available using the 'Create new timeseries file' option. This will launch the Time Series editor, with which a new *.DFS0 file may be created.

Also, the program requires an 'Evaporation Rate' item type for the time series data. This property may be set using the Time Series editor, which may be accessed from via the ellipsis button with the 'Edit timeseries in TS editor' option. Edit the item type in the Time Series editor via the File Properties dialog (Edit| Properties).

The data must cover the whole simulation period, but its time step need not be the same as the hydrodynamic simulation time step. A linear interpolation of values will be applied if the time steps differ.

Varying in domain and time

This option requires a temporally- and spatially-varying data file containing Evaporation Rate information. The file may be a 2D *.DFSU unstructured data file or a 2D *.DFS2 grid data file. Browse to the file location via the ellipsis button beside the 'File' input box (Figure 4.32).

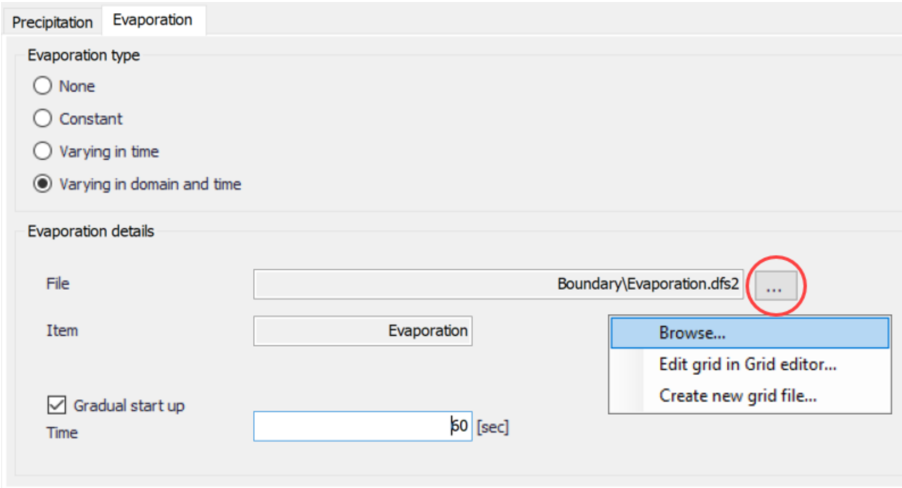


Figure 4.32 2D grid or mesh files containing time- and domain-varying evaporation rates data must be specified

An option for creating a 2D data file is also available using the 'Create new grid file' option. This will launch the Grid Series editor, with which a new *.DFS2 file may be created.

Also, the program requires a 'Evaporation Rate' item type for the grid series data. This property may be set using the Grid Series editor accessed via the ellipsis button option 'Edit grid in Grid editor'. If the loaded data is a *.DFSU file, this option will open the loaded data file in the Data Viewer.

The spatial extent must cover the 2D overland model domain. If a *.DFSU file is used, piecewise constant interpolation is used to map the data to the domain. If a *.DFS2 file is used, bilinear interpolation is applied to map the data to the domain.

The data must cover the complete simulation period, although the input time step does not need to be the same as the hydrodynamic simulation time step. A linear interpolation will be applied if the time steps differ.

Gradual start up

A soft start may be applied using the ‘Gradual start up’ check box on the tab page. With this option, the evaporation rate is increased linearly from 0 to the specified evaporation rate values. The soft start time interval must be specified.

4.4 2D WQ Precipitation and Evaporation

Water quality associated with precipitation inputs and evaporation conditions may be modelled in MIKE URBAN+ 2D Overland.

If the model setup includes precipitation and/or evaporation boundaries, specify the concentration of each component in the precipitation and evaporation water mass.

These options are available in MIKE URBAN+ if the 2D Overland modules 'Hydrodynamic (HD)' and 'Water quality (AD)' are active AND 2D Precipitation/Evaporation boundaries are included in the model setup.

Precipitation and evaporation water quality conditions can be included as:

- **Ambient concentration**. The concentration of the precipitated/evaporated water mass is set equal to the concentration of the ambient water in the domain.
- The concentration of the precipitated/evaporated water mass may be specified explicitly as:
  - **Uniform**
  - **Varying in time**
  - **Varying in domain**
  - **Varying in domain and time**

Configure these WQ characteristics individually for each WQ component defined in the setup.

Use the 2D WQ Precipitation editor to configure water quality properties for Precipitation boundaries in the model.

Use the 2D WQ Evaporation editor to configure water quality properties for Evaporation boundaries in the model.

Note that if the 2D Precipitation boundary is modelled as ‘Net precipitation’ (see 4.3 2D Precipitation and Evaporation *(p. 90)*) the precipitation concentration will be used when the net precipitation is positive. When the net precipitation is negative, the evaporation concentration is used.

More details on data requirements for each of the above-mentioned 2D WQ Precipitation/Evaporation boundary types are presented in following sections.

4.4.1 2D WQ Precipitation

The following sections describe the data requirements for the different ways of specifying 2D water quality characteristics for 2D precipitation boundaries in MIKE URBAN+.

Ambient Concentration

For this type of 2D WQ precipitation input, the concentration of the precipitation water mass is set equal to the concentration of the ambient water in the domain.

Thus, this type of WQ boundary condition does not require additional details on precipitation concentrations.

Uniform

Use this option if precipitation boundary concentrations shall remain constant (in time and domain).

Under the Precipitation Concentration Details section (Figure 4.33) define:

- **Precipitation concentration value**. Specified concentration of the selected/active pollutant component in the precipitation boundary condition.
- **Start up interval.** Soft start interval during which WQ boundary values are increased from 0 to the specified value in order to avoid shock waves being generated in the computations.

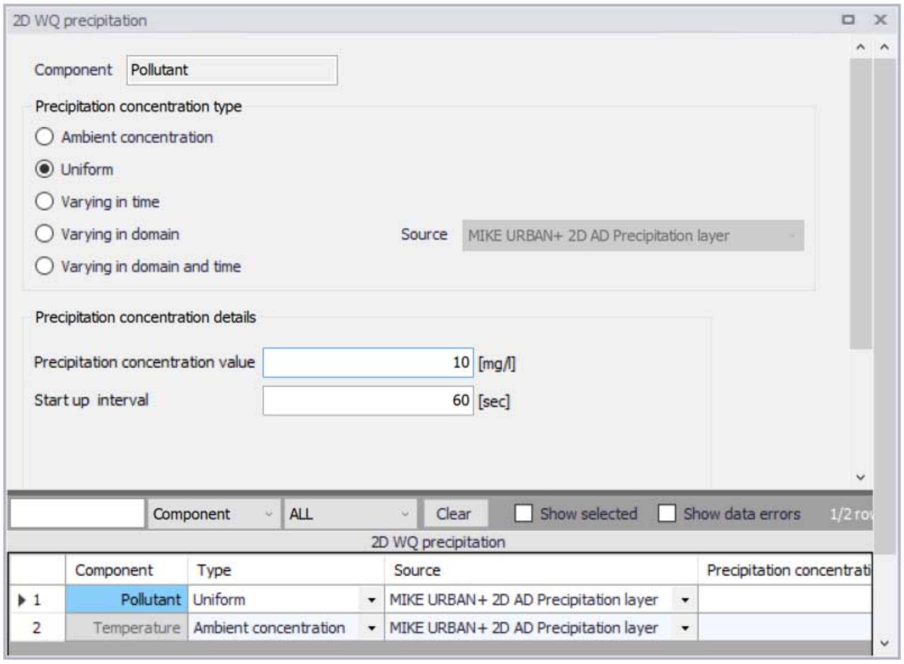


Figure 4.33 Specify parameters for Uniform 2D WQ Precipitation conditions under the Precipitation Concentration Details section on the editor. A list of WQ components in the model setup is shown in the table overview at the bottom of the editor.

Varying in Time

The case with precipitation concentrations varying in time but constant in domain requires a (*.DFS0) data file containing timeseries data on concentrations (in component unit) (Figure 4.34).

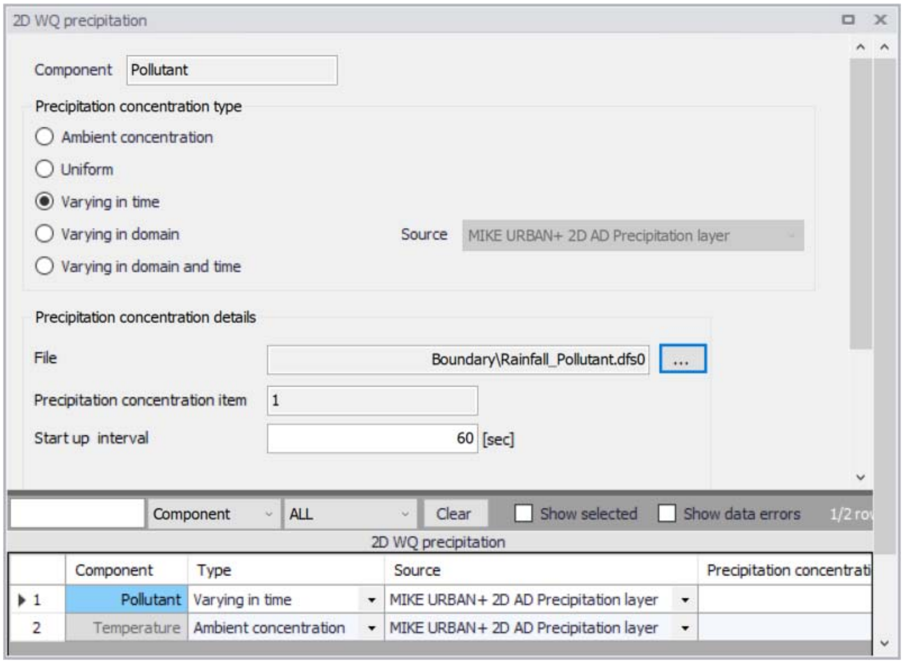


Figure 4.34 Data requirements for '2D WQ precipitation' boundaries that are 'Varying in time'

The data must cover the complete simulation period, but the time step of the input data file does not need to be the same as the time step of the hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

Varying in Domain

2D WQ precipitation boundary characteristics for various WQ components may also be modelled as varying in domain (constant in time). Input for this type of 2D WQ boundary may be defined using:

- **MIKE URBAN+ 2D AD Precipitation layer**
- **Background layer**

The data requirements for each are described in the following sections.

MIKE URBAN+ 2D AD Precipitation layer

This option requires definition of '2D AD precipitation' polygon features on the Map.

One may use the Edit Features toolbox from the 2D Overland menu ribbon, or the Flooding Layer Editing Tools toolbar on the Map (Figure 4.35). Use the 'Create' tool to draw features on the Map. Right-click to remove the last vertex added, and double-click to finish polygon drawing.

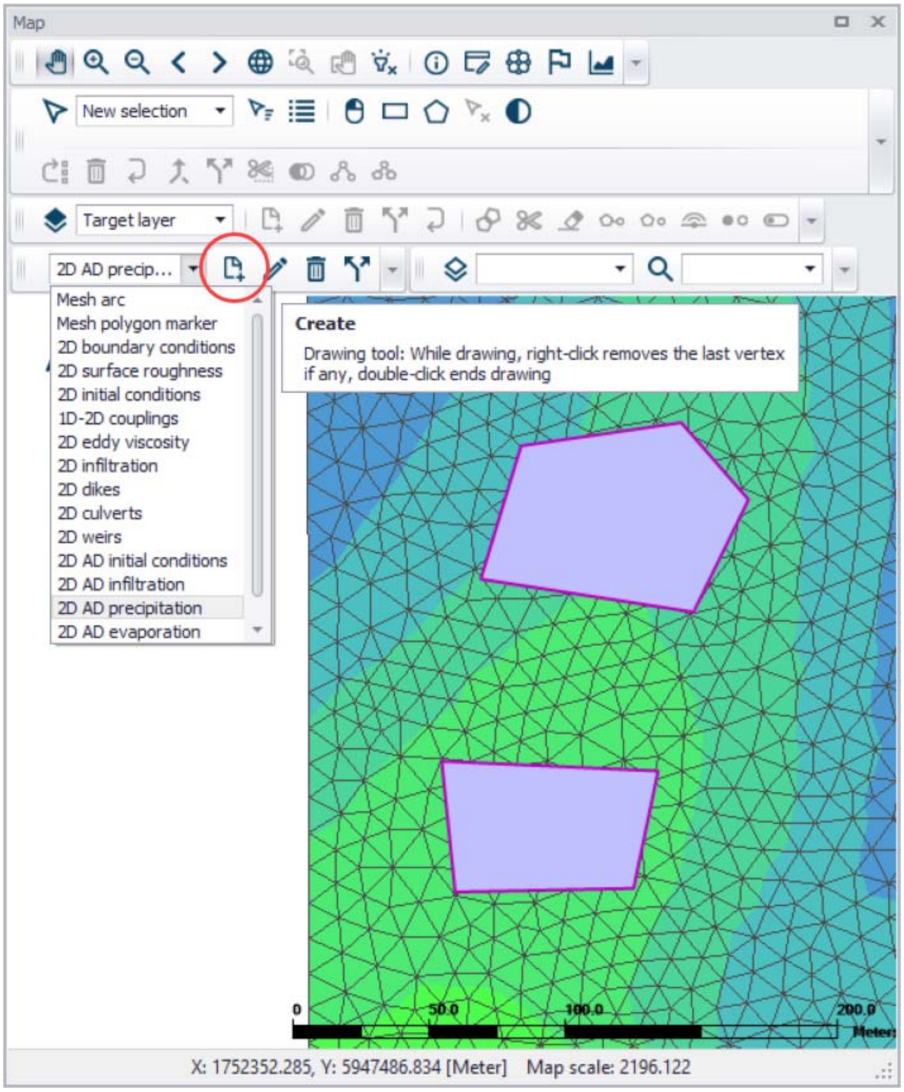


Figure 4.35 Draw 2D AD Precipitation layer polygons on the Map using the Create tool

Under the ‘Precipitation Concentration Details’ section on the editor, define the '**Default precipitation concentration**'. This value will be used over areas in the domain for which 2D AD precipitation parameters are not defined via the 2D AD Precipitation layer polygons on the Map.

Records for 2D AD precipitation polygons drawn on the Map are listed in the table on the 2D WQ precipitation editor (Figure 4.36). New records cannot be added from the tabular view. They are only added when drawing a new polygon on the map.

Specify the concentration value (related to the active/selected WQ component) for each polygon feature on the table. All items from the table are WQ component-specific. The table has the following columns:

- **Priority**: This is equivalent to the row number. Indicates the order with which values for overlapping features will be prioritized for use in the model.
- **Apply**: Check box allowing activation/deactivation of individual polygon features without deleting the polygon and its properties from the Map.
- **Polygon ID**: This is a text string used to identify the polygon.
- **Precipitation concentration value**: This is a numerical field containing the Precipitation concentration value assigned to the polygon. The unit is shown in the header (same unit as for the numerical field above the table).
- **Description**: This is an optional text string used to save a free user description for a polygon feature.

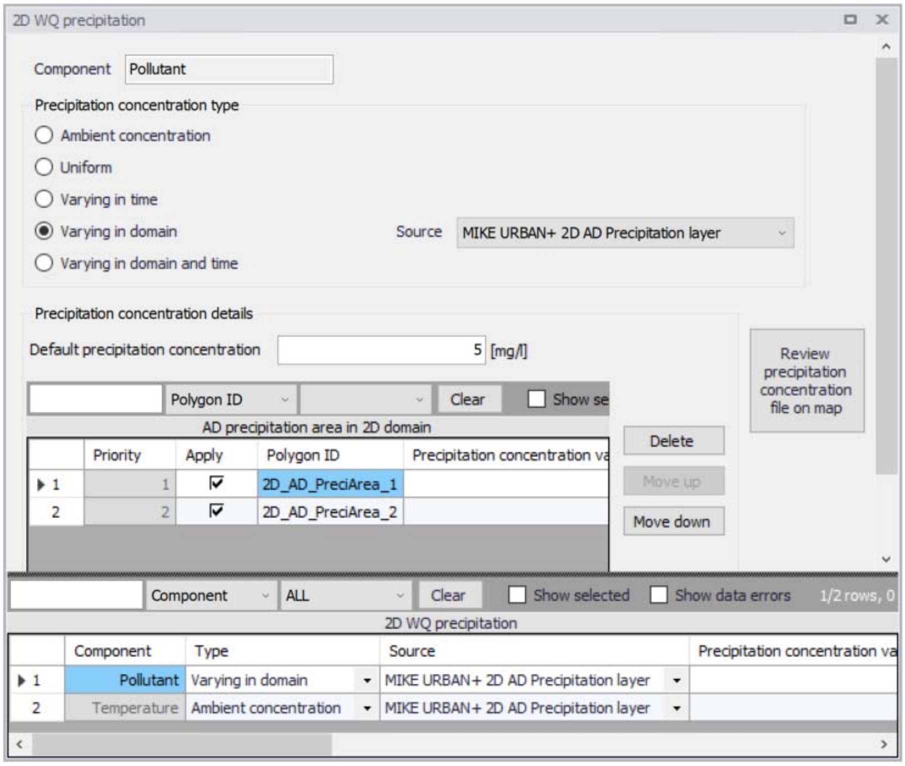


Figure 4.36 2D AD Precipitation layer polygons on the Map are listed in a table on the editor. Specify the related WQ component concentration for each polygon feature on the table.

Use the '**Review precipitation concentration file on map**' button to generate a 2D file from the precipitation concentration configuration and view the generated file on the Map. The generated 2D file type depends on the defined 2D model type: *.DFS2 grid files for rectangular grid models, and *.DFSU unstructured files for flexible mesh models.

When the file is created, it is then added as a layer on the Map (i.e. it is listed in the tree view and visible on the Map). When the file contains multiple items/components, the component which is drawn on the map is the active component in the overview table. If the file was already loaded as a layer, it is only refreshed (with new data, and with the last active component number).

Background layer

This option requires specification of a background polygon layer to represent domain-varying 2D precipitation concentrations.

The ‘Precipitation concentration details’ section on the 2D WQ Precipitation editor displays:

- **Default Precipitation concentration**: This value will be used over areas in the domain for which parameters are not defined with the background layer polygons.
- **Layer**: A drop-down list populated with valid layers loaded as background layers on the Map. A layer is valid if it is a *.TAB, or *.SHP polygon layer.
- **Precipitation concentration item**: Drop-down list populated with the full list of attributes from the selected layer.
- **Unit**: Drop-down list showing unit options depending on the 'Type' of WQ component as specified in the 'WQ components' editor:
  - For Type = 'Pollutant', it shows the list of all supported units for concentration (e.g. mg/l, g/m3, kg/m3, g/l, mu-g/l, pound/(feet US)^3, pound/feet^3, pound/(yard US)^3, pound/yard^3, etc.)
  - For Type = 'Microorganism', it shows the list of all supported units for bacteria / micro-organism concentrations (e.g. million/100 ml, per 100 ml, per liter, etc.)
  - For Type = 'Temperature', it shows the list of all supported units for temperature (e.g. degree Celsius, degree Fahrenheit, degree kelvin, etc.)
  - For Type = 'pH', it shows a single undefined unit [].
  - For Type = 'Salinity', it shows the list of all supported units for salinity (e.g. PSU, per thousand, etc.)
  - For types 'Water age' and 'Water blend', it shows a single undefined unit [].
- **Start up interval:** Soft start interval during which WQ boundary values are increased from 0 to the specified value in order to avoid shock waves being generated in the computations.

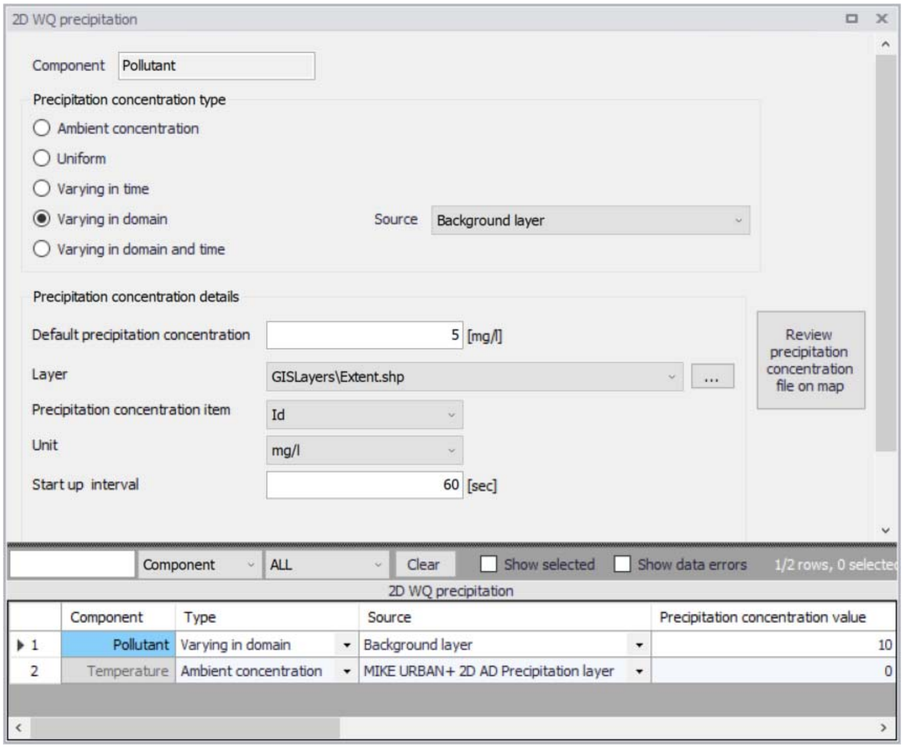


Figure 4.37 Define the Background Layer to use as 2D Precipitation Concentration input in the editor

The '**Review precipitation concentration file on map**' button generates a 2D file from the precipitation concentration configuration and loads the generated file on the Map. The generated 2D file type depends on the defined 2D model type: *.DFS2 grid files for rectangular grid models, and *.DFSU unstructured files for flexible mesh models.

When the file is created, it is then added as a layer on the Map (i.e. it is listed in the tree view and visible on the Map). When the file contains multiple items/components, the component which is drawn on the map is the active component in the overview table. If the file was already loaded as a layer, it is only refreshed (with new data, and with the last active component number).

Varying in Domain and Time

The case with concentration varying both in time and domain requires a 2D file with concentration values. The file must be a 2D unstructured data file (*.DFSU) or a 2D grid data file (*.DFS2) with 'Dimensionless Factor ()' Item type.

The area in the 2D data file must cover the model domain. If a *.DFSU file is defined, piecewise constant interpolation is used to map the data to the domain mesh. If a *.DFS2 file is defined, bilinear interpolation is used to map the data to the domain grid.

The data must cover the complete simulation period, although the time step of the input data file does not have to be the same as the time step of the hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

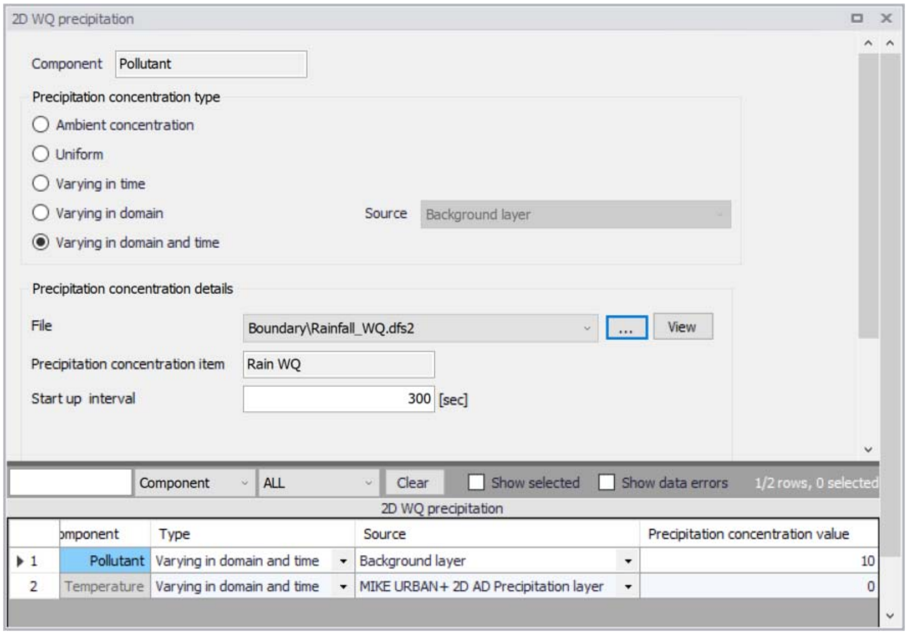


Figure 4.38 Data requirements for 2D WQ Precipitation boundaries that vary in domain and time

On the editor, define the 2D data file under the 'Precipitation Concentration Details' section:

- **File**: A drop-down list of valid 2D layer files on the Map. A file is valid if it is a time-varying *.DFS2 or *.DFSU file with Item type = 'Dimensionless Factor'.

  The ellipsis button launches the 'Open a dfs file' dialog that is used to locate a valid time-varying 2D file. The dialog filters on types '2D Grid file (*.DFS2)' or 'Unstructured data file (*.DFSU)', and has a drop-down list for Items that may be loaded.

  The 'View' button opens the selected time-varying 2D file for viewing/editing. It launches the Grid Editor if a *.DFS2 file has been selected in the editor. If a *.DFSU file has been selected, it opens the file using the Data Manager.

- **Precipitation concentration item**: Non-editable text box displaying the selected Item from the loaded file.

  The database stores one file name and item for each WQ component.

- **Start up interval:** Soft start interval during which WQ boundary values are increased from 0 to the specified value in order to avoid shock waves in the computations.

4.4.2 2D WQ Evaporation

The following sections describe the data requirements for the different ways of specifying 2D water quality characteristics for 2D evaporation boundaries in MIKE URBAN+.

The structure and data requirements for 2D evaporation boundaries are very similar to those for precipitation boundaries described in the previous section (see Chapter 4.4.1 2D WQ Precipitation *(p. 99)*).

Also, note that it is usually most sensible to set the concentration of the evaporated water mass to zero, in which case the component is conserved in the water.

Ambient Concentration

For this type of 2D WQ evaporation input, the concentration of the evaporated water mass is set equal to the concentration of the ambient water in the domain.

Thus, this type of WQ boundary condition does not require additional details on evaporation concentrations.

Uniform

Use this option if evaporation boundary concentrations shall remain constant (in time and domain).

Under the Evaporation Concentration Details section define:

- **Evaporation concentration value**. Specified concentration of the selected/active pollutant component in the evaporation boundary condition.
- **Soft start interval.** Soft start interval during which WQ boundary values are increased from 0 to the specified value in order to avoid shock waves being generated in the computations.

Specify parameters for Uniform 2D WQ Evaporation conditions under the Evaporation Concentration Details section on the editor. A list of WQ components in the model setup is shown in the table overview at the bottom of the editor.

Varying in Time

The case with evaporation concentrations varying in time but constant in domain requires a (*.DFS0) data file containing timeseries data on concentrations (in component unit) (Figure 4.39).

The data must cover the complete simulation period, but the time step of the input data file does not need to be the same as the time step of the hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

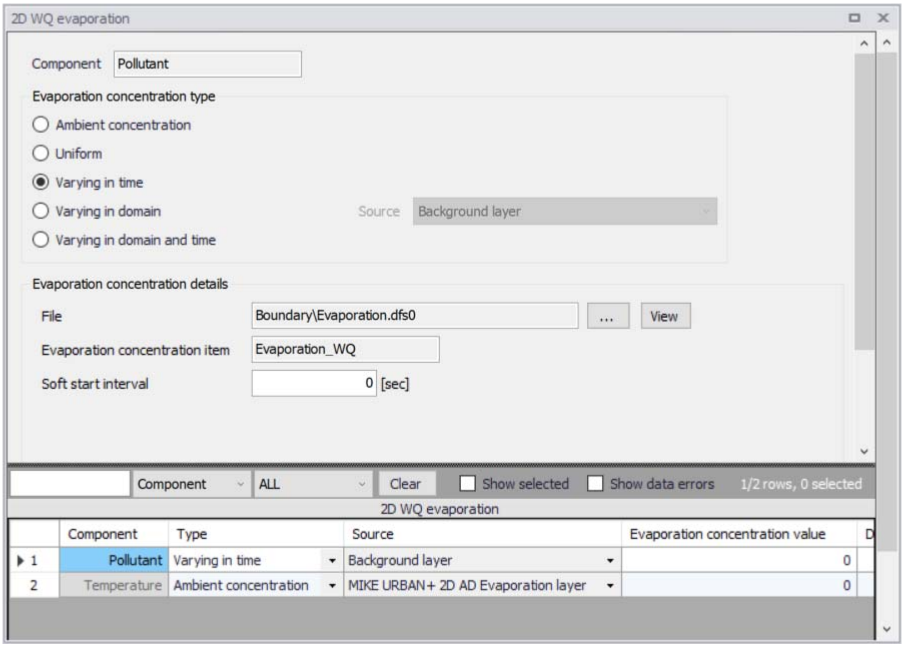


Figure 4.39 Data requirements for '2D WQ evaporation' boundaries that are 'Varying in time'

Varying in Domain

2D WQ evaporation boundary characteristics for various WQ components may also be modelled as varying in domain (constant in time). Input for this type of 2D WQ boundary may be defined using:

- **MIKE URBAN+ 2D AD Evaporation layer**
- **Background layer**

The data requirements for each are described in the following sections.

MIKE URBAN+ 2D AD Evaporation layer

This option requires definition of '2D AD evaporation' polygon features on the Map.

One may use the Edit Features toolbox from the 2D Overland menu ribbon, or the Flooding Layer Editing Tools toolbar on the Map. Use the 'Create' tool to draw features on the Map. Right-click to remove the last vertex added, and double-click to finish polygon drawing.

Under the 'Evaporation Concentration Details' section on the editor, define the '**Default evaporation concentration**'. This value will be used over areas in the domain for which 2D AD evaporation parameters are not defined via the '2D AD evaporation' polygons on the Map.

Records for 2D AD evaporation polygons drawn on the Map are listed in the table on the 2D WQ Evaporation editor (Figure 4.40). New records cannot be added from the tabular view. They are only added when drawing a new polygon on the map.

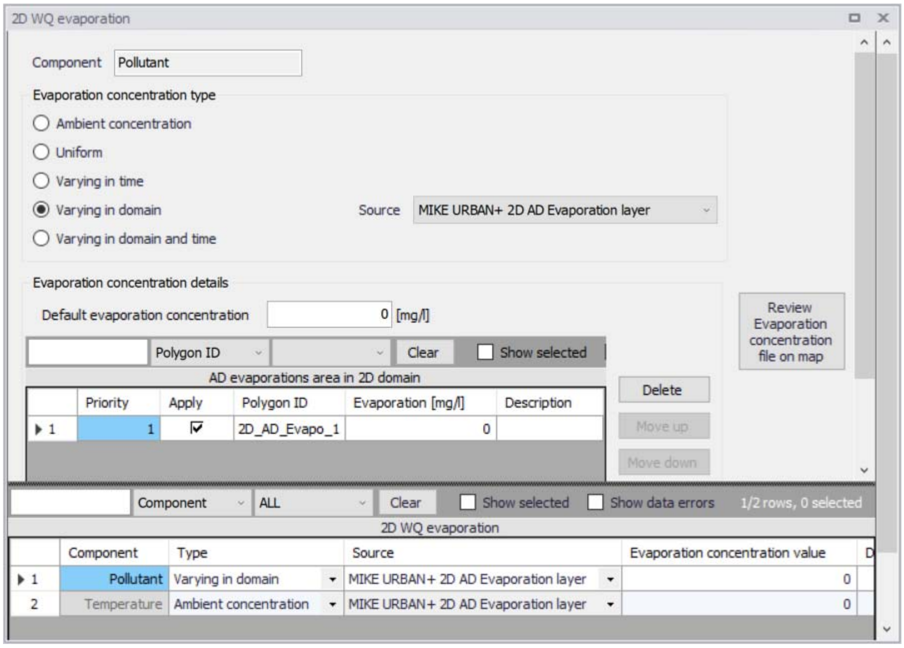


Figure 4.40 Edit the properties of the '2D AD evaporation' layer polygons in the table on the 2D WQ Evaporation editor.

Specify the concentration value (related to the active/selected WQ component) for each polygon feature on the table. All items from the table are WQ component-specific. The table has the following columns:

- **Priority**: This is equivalent to the row number. Indicates the order with which values for overlapping features will be prioritized for use in the model.
- **Apply**: Check box allowing activation/deactivation of individual polygon features without deleting the polygon and its properties from the Map.

- **Polygon ID**: This is a text string used to identify the polygon.
- **Evaporation**: This is a numerical field containing the evaporation concentration value assigned to the polygon. The unit is shown in the header (same unit as for the numerical field above the table).
- **Description**: This is an optional text string used to save a free user description for a polygon feature.

Use the '**Review evaporation concentration file on map**' button to generate a 2D file from the evaporation concentration configuration and view the generated file on the Map. The generated 2D file type depends on the defined 2D model type: *.DFS2 grid files for rectangular grid models, and *.DFSU unstructured files for flexible mesh models.

When the file is created, it is then added as a layer on the Map (i.e. it is listed in the tree view and visible on the Map). When the file contains multiple items/components, the component which is drawn on the map is the active component in the overview table. If the file was already loaded as a layer, it is only refreshed (with new data, and with the last active component number).

Background layer

This option requires specification of a background polygon layer to represent domain-varying 2D evaporation concentrations.

The 'Evaporation concentration details' section on the 2D WQ Evaporation editor displays:

- **Default evaporation concentration**: This value will be used over areas in the domain for which parameters are not defined with the background layer polygons.
- **Layer**: A drop-down list populated with valid layers loaded as background layers on the Map. A layer is valid if it is a *.TAB, or *.SHP polygon layer.
- **Evaporation concentration item**: Drop-down list populated with the full list of attributes from the selected layer.
- **Unit**: Drop-down list showing unit options depending on the 'Type' of WQ component as specified in the 'WQ components' editor:
  - For Type = 'Pollutant', it shows the list of all supported units for concentration (e.g. mg/l, g/m3, kg/m3, g/l, mu-g/l, pound/(feet US)^3, pound/feet^3, pound/(yard US)^3, pound/yard^3, etc.)
  - For Type = 'Microorganism', it shows the list of all supported units for bacteria / micro-organism concentrations (e.g. million/100 ml, per 100 ml, per liter, etc.)
  - For Type = 'Temperature', it shows the list of all supported units for temperature (e.g. degree Celsius, degree Fahrenheit, degree kelvin, etc.)
  - For Type = 'pH', it shows a single undefined unit [].

- For Type = 'Salinity', it shows the list of all supported units for salinity (e.g. PSU, per thousand, etc.)
- For types 'Water age' and 'Water blend', it shows a single undefined unit [].

- **Soft start interval**: Interval during which WQ boundary values are increased from 0 to the specified value in order to avoid shock waves in the computations.

The '**Review evaporation concentration file on map**' button generates a 2D file from the evaporation concentration configuration and loads the generated file on the Map. The generated 2D file type depends on the defined 2D model type: *.DFS2 grid files for rectangular grid models, and *.DFSU unstructured files for flexible mesh models.

When the file is created, it is then added as a layer on the Map (i.e. it is listed in the tree view and visible on the Map). When the file contains multiple items/components, the component which is drawn on the map is the active component in the overview table. If the file was already loaded as a layer, it is only refreshed (with new data, and with the last active component number).

Varying in Domain and Time

The case with evaporation concentrations varying both in time and domain requires a 2D file with concentration values. The file must be a 2D unstructured data file (*.DFSU) or a 2D grid data file (*.DFS2) with 'Dimensionless Factor ()' Item type.

The area in the 2D data file must cover the model domain. If a *.DFSU file is defined, piecewise constant interpolation is used to map the data to the domain mesh. If a *.DFS2 file is defined, bilinear interpolation is used to map the data to the domain grid.

The data must cover the complete simulation period, although the time step of the input data file does not have to be the same as the time step of the hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

On the editor, define the 2D data file under the 'Evaporation Concentration Details' section:

- **File**: A drop-down list of valid 2D layer files on the Map. A file is valid if it is a time-varying *.DFS2 or *.DFSU file with Item type = 'Dimensionless Factor'.

  The ellipsis button launches the 'Open a dfs file' dialog that is used to locate a valid time-varying 2D file. The dialog filters on types '2D Grid file (*.DFS2)' or 'Unstructured data file (*.DFSU)', and has a drop-down list for Items that may be loaded.

The 'View' button opens the selected time-varying 2D file for viewing/editing. It launches the Grid Editor if a *.DFS2 file has been selected in the editor. If a *.DFSU file has been selected, it opens the file using the Data Manager.

- **Evaporation concentration item**: Non-editable text box displaying the selected Item from the loaded file.

  The database stores one file name and item for each WQ component.

- **Soft start interval**: Interval during which WQ boundary values are increased from 0 to the specified value in order to avoid shock waves in the computations.

4.5 2D Infiltration

The effect of infiltration and leakage at the surface zone in 2D models may be modelled with the MIKE URBAN+ 2D Overland module.

In MIKE URBAN+, the following infiltration Types are available via the 2D Infiltration editor:

- **None**
- **Varying in Domain**. Infiltration rate varying spatially over the domain.
- **Varying in Domain and Time**. Infiltration rates that vary both spatially and temporally.
- **Varying in Domain Using Capacity**. Spatially-varying infiltration considering the capacity of the infiltration zone. Uses simplified infiltration model that calculates net infiltration using specified information on surface zone layer properties.

More details on the various options and data requirements for the above-mentioned Infiltration Types are presented in succeeding sections.

4.5.1 Net Infiltration Rate

The most direct way of including infiltration in the 2D model is to specify net infiltration rate. When using a net infiltration rate, an unsaturated zone is not defined and thus has no capacity limits. The specified infiltration rates are in full effect as long as there is enough water in the grid cell/element.

4.5.2 Constant Infiltration With Capacity

It is however also possible to calculate the net infiltration rate by a simplified infiltration model.

The simplified model describes the infiltration from the free surface zone to the unsaturated zone, and the leakage from the unsaturated zone to the saturated zone.

The following parameters are specified for the simplified infiltration model:

1. **Infiltration rate**. Flow between the free surface and infiltration zones.
2. **Leakage rate**. Flow between saturated and unsaturated zones.
3. **Depth to/level of bottom of infiltration zone.** The depth to or level of the bottom of the infiltration zone.
4. **Porosity** of infiltration zone. Porosity (void fraction) of the infiltration zone.
5. **Initial water volume** in Percentage of capacity (interval: 0-100 [%]) OR as fraction of the infiltration zone volume (interval: 0-porosity[()])

The model can account for a decreased storage capacity due to previous rainfall events. The model assumes the following:

- The unsaturated zone is modelled as an infiltration zone with constant porosity over the full depth of the zone.
- The infiltrated volume from the free surface zone and to the unsaturated zone is based on a constant flow rate $V_{infiltration} = Q_i \Delta t$ where Q_i is the prescribed infiltration rate.
- The leaked volume from the unsaturated zone to the saturated zone is based on a constant flow rate $V_{leakage} = Q_l \Delta t$, where Q_l is the prescribed leakage rate.

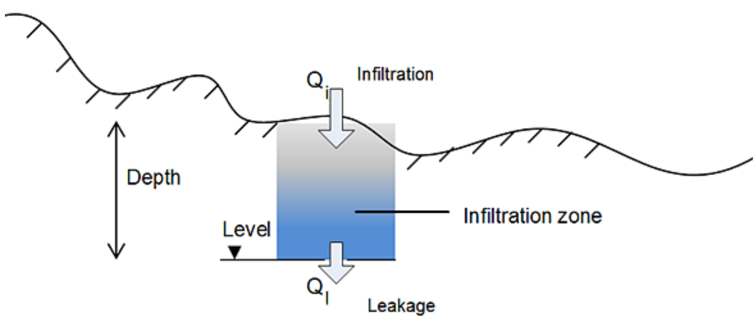


Figure 4.41 Definition of infiltration with storage and leakage: In the case of net infiltration, $Q_{infiltration} = Q_{leakage}$, the model assumes an infinite storage volume in the infiltration zone.

The infiltrated flow volume cannot exceed the amount of water available in the free surface water zone nor the difference between the water capacity of the infiltration zone and the actual amount of water stored there. It is possible that the infiltration flow completely drains the free surface zone from water and thus creates a dried-out point in the 2-dimensional horizontal flow calculations. In case the infiltration is described by 'Constant infiltration with capacity' it is also possible that the infiltration zone becomes fully saturated so infiltration cannot take place (in which case Q_i is set to 0).

4.5.3 Varying in Domain

Data for infiltration that is 'Varying in domain' may be defined in the 2D overland model with:

- MIKE URBAN+ 2D infiltration layer
- Background layer
- 2D file

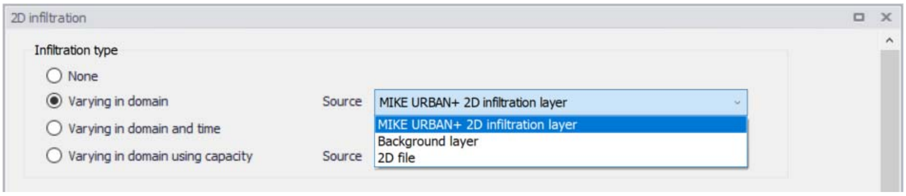


Figure 4.42 Options for specifying spatially-varying infiltration for the 2D overland model under the Information Type section

MIKE URBAN+ 2D infiltration layer

This option requires definition of 2D infiltration polygon features on the Map.

One may use the Edit Features toolbox from the 2D Overland menu ribbon (Figure 4.43), or the Flooding Layer Editing Tools toolbar on the Map (Figure 4.44).

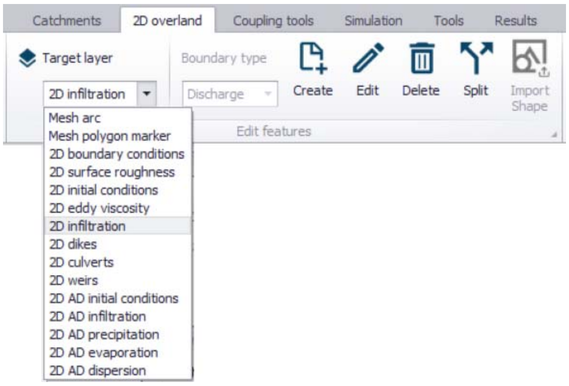


Figure 4.43 Edit 2D model layers on the Map using the Edit Features toolbar on the 2D Overland menu ribbon.

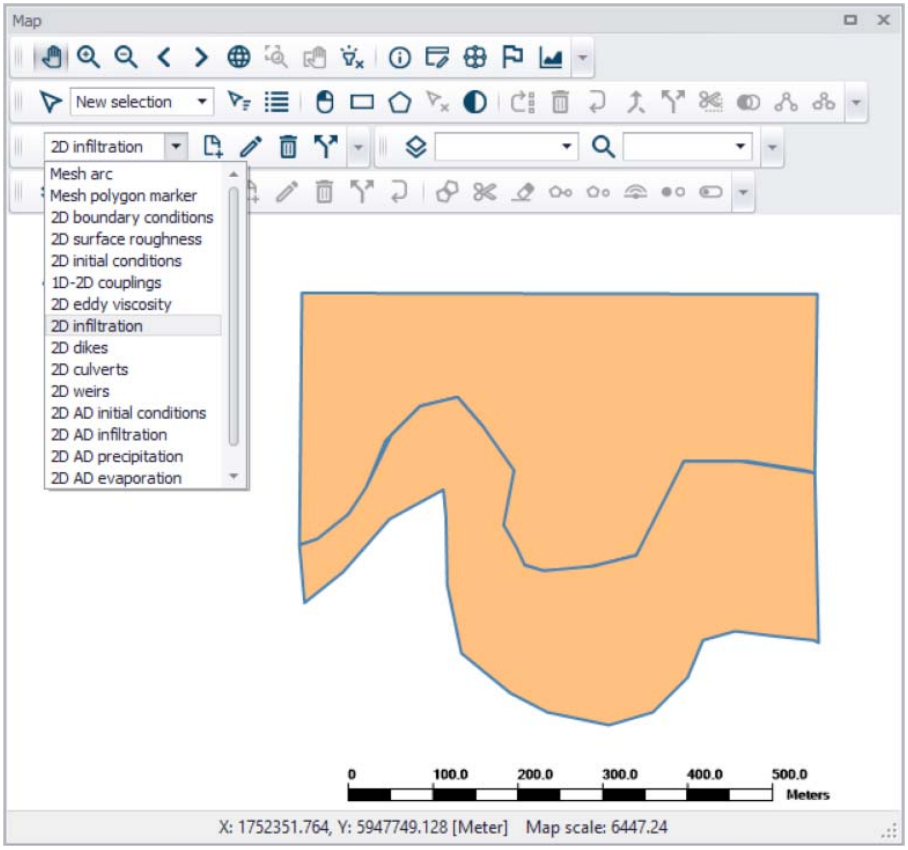


Figure 4.44 Create 2D infiltration polygon features on the Map using tools from the Flooding Layer Editing Tools toolbar

Define the '**Default infiltration rate**' under the Infiltration Details section on the editor. This value will be used over areas in the domain for which infiltration parameters are not defined via the 2D Infiltration layer polygons on the Map.

The table under the Infiltration Details section on the editor lists information on each 2D infiltration polygon feature on the Map (Figure 4.45). Assign relevant infiltration properties for each feature on the table:

- **Priority**: Indicates the order with which values for overlapping features will be prioritized for use in the model.
- **Apply**: Check box allowing activation/deactivation of individual polygon features without deleting the polygon and its properties from the Map.
- **Polygon ID**: Unique 2D Infiltration polygon identifier.
- **Infiltration rate**: Infiltration rate value assigned to a polygon. The unit is shown in the header.
- **Description**: Optional text string for a free user description.

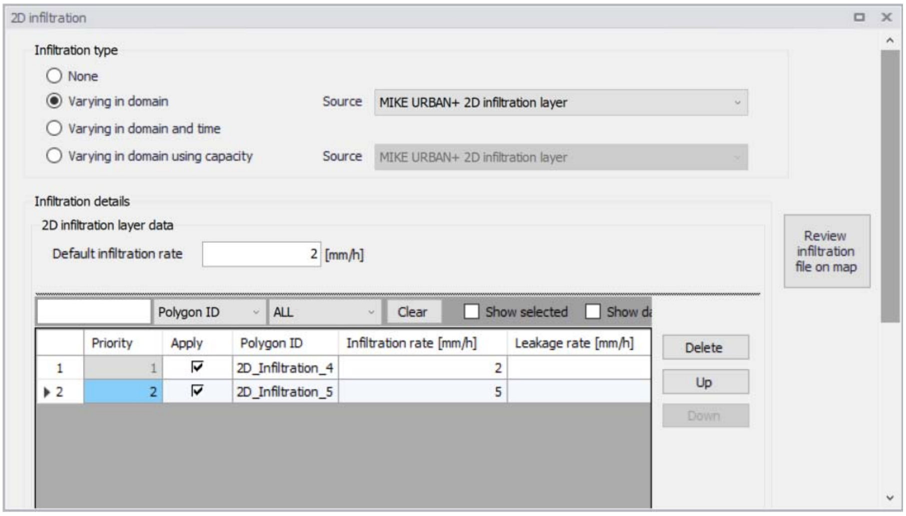


Figure 4.45 Fill-in parameters for 2D infiltration polygon features on the Map on the table under the 'Infiltration details' section. Use the 'Review infiltration file on map' button to view the configured 2D infiltration data layer on the Map.

Use the '**Review infiltration file on map**' button to generate a 2D file from the infiltration configuration and view the generated file on the Map. The generated 2D file type depends on the defined 2D model type: *.DFS2 grid files for rectangular grid models, and *.DFSU unstructured files for flexible mesh models.

Background Layer

When the space-varying infiltration input source is from a 'Background Layer', the 'Infiltration details' section on the 2D infiltration editor displays:

- **Layer**: The drop-down list is populated with background layers on the Map valid for use as 2D infiltration data. A layer is valid if it is of a *.TAB or *.SHP polygon type. Beside the Layer list, use the ellipsis button to browse to the location of other *.TAB or *.SHP polygon files.
- **Infiltration rate item:** A drop-down list of numerical attributes from the selected layer, which may be used as infiltration rate values.
- **Unit**: Select the desired units from the drop-down list of supported units for infiltration rate in the program.
- **Default infiltration rate**: This value will be used over areas in the domain for which infiltration parameters are not defined with the background layer polygons.

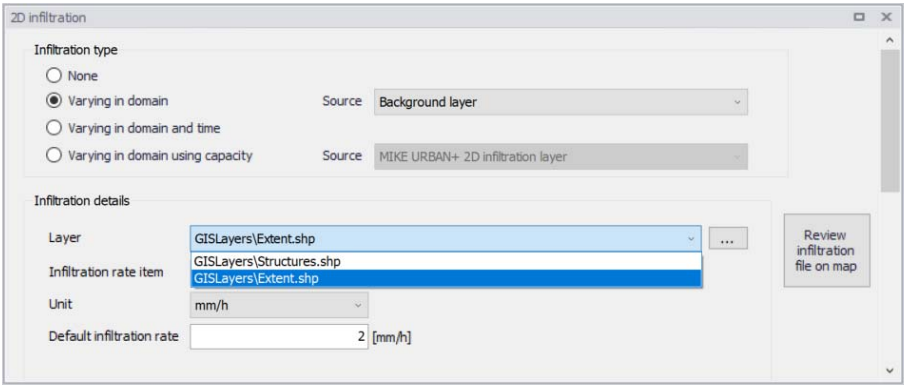


Figure 4.46 The program detects and offers valid Background Layers on the Map as sources of infiltration data. Use the 'Review infiltration file on map' button to view the configured 2D infiltration data layer on the Map.

Use the '**Review infiltration file on map**' button to generate a 2D file from the infiltration configuration and view the generated file on the Map. The generated 2D file type depends on the defined 2D model type: *.DFS2 grid files for rectangular grid models, and *.DFSU unstructured files for flexible mesh models.

2D File

The '2D File' option of specifying the source of spatially-varying infiltration input in the model, define the following in the editor:

- **File**: A drop-down list of valid 2D background layer files on the Map. A file is valid if it is a *.DFS2 or *.DFSU file with Item type = 'Infiltration', 'Seepage', or 'Recharge' (Figure 4.47).

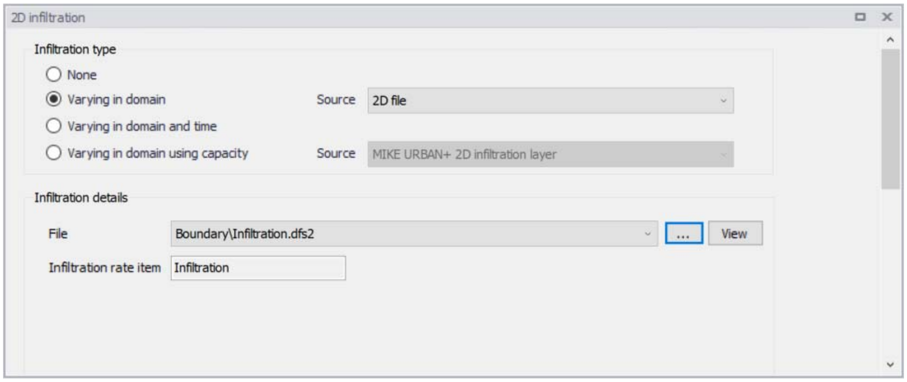


Figure 4.47 Point to a 2D infiltration file under the Infiltration details section on the editor

To the right side of the drop-down list is an ellipsis button that launches the 'Open a dfs file' dialog. Use the dialog to browse for a valid file (Figure 4.48). It filters on types '2D Grid file (*.DFS2)' or 'Unstructured data file (*.DFSU)', and has a drop-down list for selecting the Item to load.

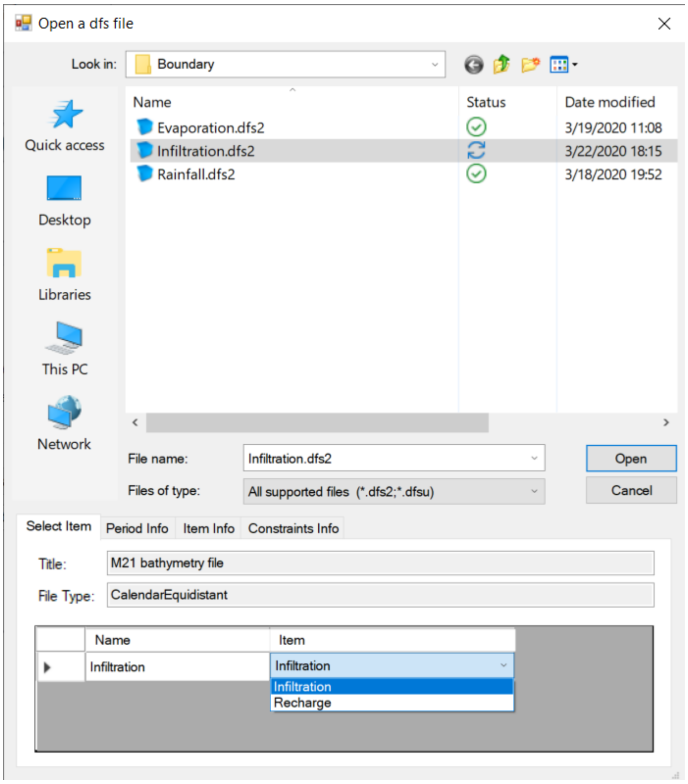


Figure 4.48 Browse to the valid 2D file via the ‘Open a dfs file’ dialog.

The '**View**' button opens the selected 2D file for viewing/editing. It launches the Grid Editor if a *.DFS2 file has been selected in the editor. If a *.DFSU file has been selected, it opens the file using the Data Manager.

- **Infiltration rate item**: Non-editable text box displaying the selected Item from the loaded file.

4.5.4 Varying in Domain and Time

Infiltration data that varies with time as well as domain is defined as a time-varying 2D file in the 2D overland model (Figure 4.49).

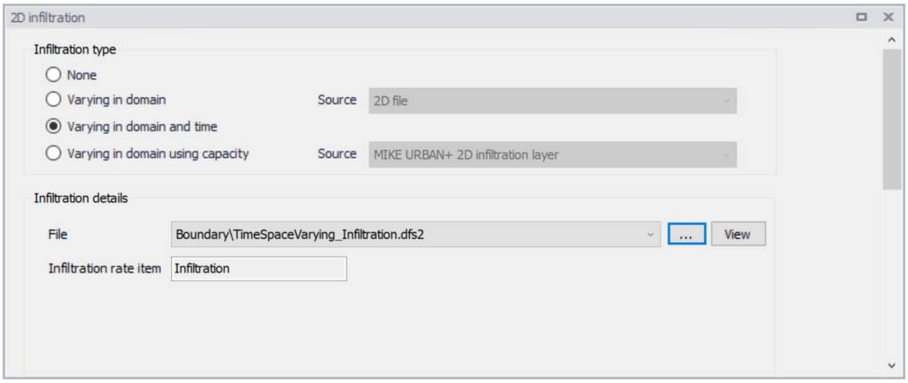


Figure 4.49 Browse to a time-varying 2D file for input infiltration data that varies in domain and time

- **File**: A drop-down list of valid 2D background layer files on the Map. A file is valid if it is a time-varying *.DFS2 or *.DFSU file with Item type = 'Infiltration', 'Seepage', or 'Recharge'

  The ellipsis button launches the 'Open a dfs file' dialog that is used to locate a valid time-varying 2D file. The dialog filters on types '2D Grid file (*.DFS2)' or 'Unstructured data file (*.DFSU)', and has a drop-down list for Items that may be loaded.

  The '**View**' button opens the selected time-varying 2D file for viewing/editing. It launches the Grid Editor if a *.DFS2 file has been selected in the editor. If a *.DFSU file has been selected, it opens the file using the Data Manager.

- **Infiltration rate item**: Non-editable text box displaying the selected Item from the loaded file.

4.5.5 Varying in Domain Using Capacity

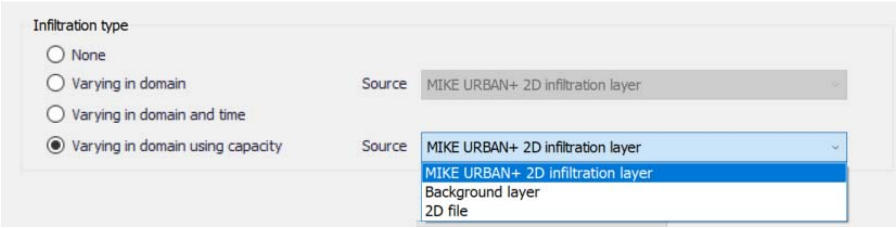


Figure 4.50 Source options for 2D Infiltration data of 'Varying in domain using capacity' type

Data for infiltration that is 'Varying in domain using capacity' may be defined in the 2D model with the following sources (Figure 4.50):

- **MIKE URBAN+ 2D infiltration layer.** 2D infiltration polygons defined on the Map in MIKE URBAN+.
- **Background layer**. Pre-loaded polygon background layer on the Map.
- **2D file**. Existing infiltration 2D file (*.DFS2 or *.DFSU types)

More details on configuration requirements for each source type are described in succeeding sections.

MIKE URBAN+ 2D infiltration layer

This option requires the definition of 2D infiltration polygon features on the Map.

One may use the Edit Features toolbox from the 2D Overland menu ribbon (Figure 4.43), or the Flooding Layer Editing Tools toolbar on the Map (Figure 4.44) to create 2D infiltration polygons. Records for each polygon feature are listed in a table on the 2D Infiltration editor (Figure 4.51).

Under the 'Infiltration Details' section, specify:

- **Bottom level of infiltration zone defined from**:
  - Depth below ground level
  - Specified level
- **Initial water volume in infiltration zone defined from**:
  - Percentage of the capacity
  - Percentage of the infiltration zone volume

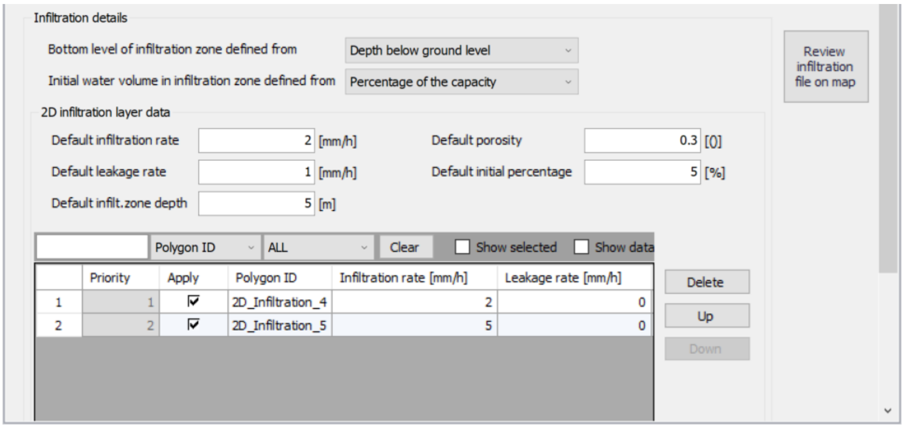


Figure 4.51 Configure 2D infiltration data layer properties under the Infiltration Details section on the editor. The table lists 2D infiltration polygons drawn on the Map.

Under the '2D infiltration layer data' section, define:

- **Default infiltration rate**. Flow between the free surface and infiltration zones.
- **Default leakage rate**. Flow between saturated and unsaturated zones.
- **Default infilt. zone depth/Default infilt. zone level**. ‘Default infilt. zone **depth**’ when 'Bottom level of infiltration zone defined from’ = '**Depth** below ground level'. ‘Default infilt. zone **level**’ when 'Bottom level of infiltration zone defined from’ = 'Specified **level**'.
- **Default porosity**. Porosity (void fraction) of the infiltration zone.
- **Default initial percentage**. Initial water volume in Percentage of capacity (interval: 0-100 [%]) OR as fraction of the infiltration zone volume (interval: 0-porosity[()])

These default values are used over areas not covered by the defined 2D infiltration layer polygon features on the Map.

The table listing the features for the 2D infiltration layer has the following data columns:

- **Infiltration rate.** Flow between the free surface and infiltration zones.
- **Leakage rate.** Flow between saturated and unsaturated zones.
- **Infilt. zone depth/Infilt. zone level**. The depth to or level of the bottom of the infiltration zone indicating its extent. When ‘Bottom level of infiltration defined from’ = 'Depth below ground level', the header name is 'Infilt. zone depth'. When ‘Bottom level of infiltration defined from’ = 'Specified level', the header name is 'Infilt. zone level'
- **Porosity**: Porosity (void fraction) of the infiltration zone.

- **Initial percentage**: Initial percentage of the capacity or fraction of the infiltration zone volume (depending on the selected option in the drop-down list 'Initial water volume in infiltration zone defined from'). Initial water volume in Percentage of capacity (interval: 0-100 [%]) OR as fraction of the infiltration zone volume (interval: 0-porosity[()])

Define desired values for the above parameters on the table for each 2D infiltration polygon.

Use the ‘**Review infiltration file on map**’ button to generate a 2D file from the infiltration configuration and view the generated file on the Map. The generated 2D file type depends on the defined 2D model type: *.DFS2 grid files for rectangular grid models, and *.DFSU unstructured files for flexible mesh models.

Background Layer

When a background layer shall be the source of the 2D infiltration (Varying in domain using capacity) input, the corresponding polygon data layer must be loaded onto the Map as a Background Layer.

Under the ‘Infiltration Details’ section, specify:

- **Bottom level of infiltration zone defined from**:
  - Depth below ground level
  - Specified level
- **Initial water volume in infiltration zone defined from**:
  - Percentage of the capacity
  - Percentage of the infiltration zone volume

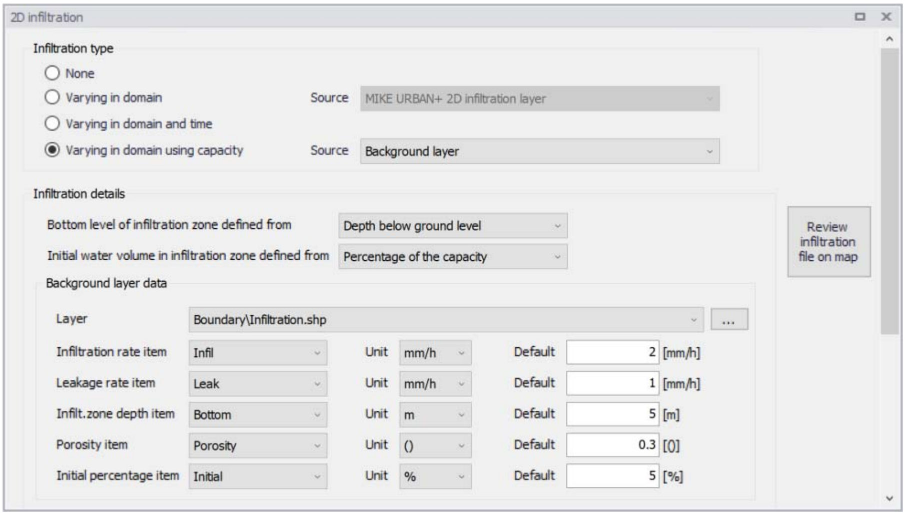


Figure 4.52 Use a polygon background layer with numerical attributes for infiltration zone parameters.

Define the following parameters under the ‘Background layer data' section:

- **Layer**. A drop-down list showing valid background layers loaded on the Map. Valid layers are *.TAB or *.SHP polygon types. The ellipsis button is used to browse to an external polygon *.TAB or *.SHP file.
- Below is a list of parameters that must be defined based on the loaded Layer:
  - **Infiltration rate item**. Flow between the free surface and infiltration zones.
  - **Leakage rate item**. Flow between saturated and unsaturated zones.
  - **Infilt. zone depth/level item**. The label uses the word ‘depth' if ‘Bottom level of infiltration zone defined from' = ‘Depth below ground level'. Otherwise, the label indicates ‘level' if ‘Bottom level of infiltration zone defined from' = ‘Specified level'.
  - **Porosity item**. Porosity (void fraction) of the infiltration zone.
  - **Initial percentage item**. Initial water volume in Percentage of capacity (interval: 0-100 [%]) OR as fraction of the infiltration zone volume (interval: 0-porosity[()])

  Select the appropriate numerical attribute from the loaded layer file for each parameter using the drop-down list. Each drop-down list is populated with the full list of (numerical) attributes from the selected layer.

  To the right of the parameter input boxes are '**Default**' value input boxes. These values are used over areas not covered by the input file layer features.

Use the ‘**Review infiltration file on map**’ button to generate a 2D file from the infiltration configuration and view the generated file on the Map. The generated 2D file type depends on the defined 2D model type: *.DFS2 grid files for rectangular grid models, and *.DFSU unstructured files for flexible mesh models.

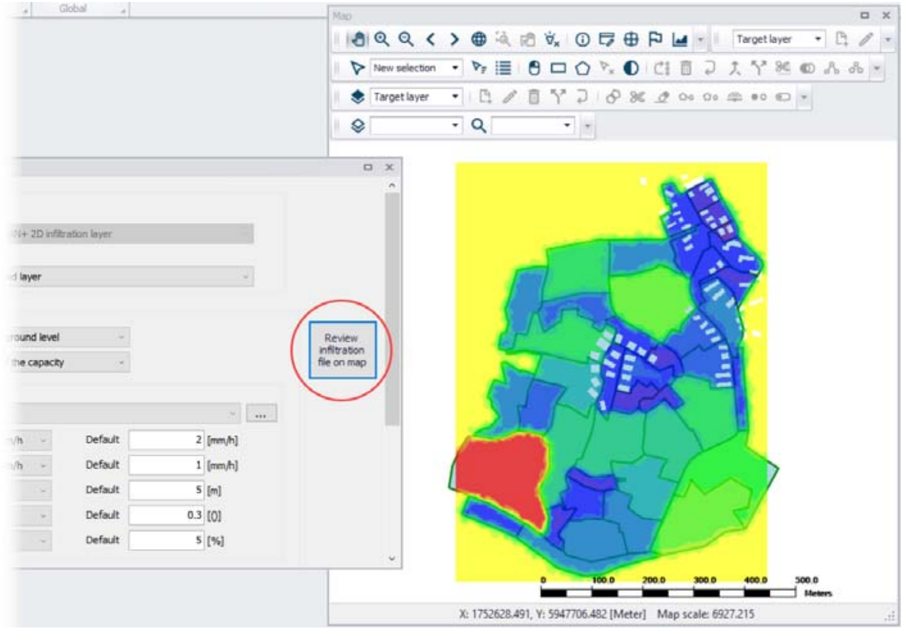


Figure 4.53 2D infiltration data layer (*.DFSU) generated from configuration using the ‘Review infiltration file on map’ functionality

2D File

This source option requires specifying a spatially-varying 2D infiltration input file containing 5 data items describing infiltration zone properties.

Under the ‘Infiltration Details’ section, specify:

- **Bottom level of infiltration zone defined from**:
  - Depth below ground level
  - Specified level
- **Initial water volume in infiltration zone defined from**:
  - Percentage of the capacity
  - Percentage of the infiltration zone volume

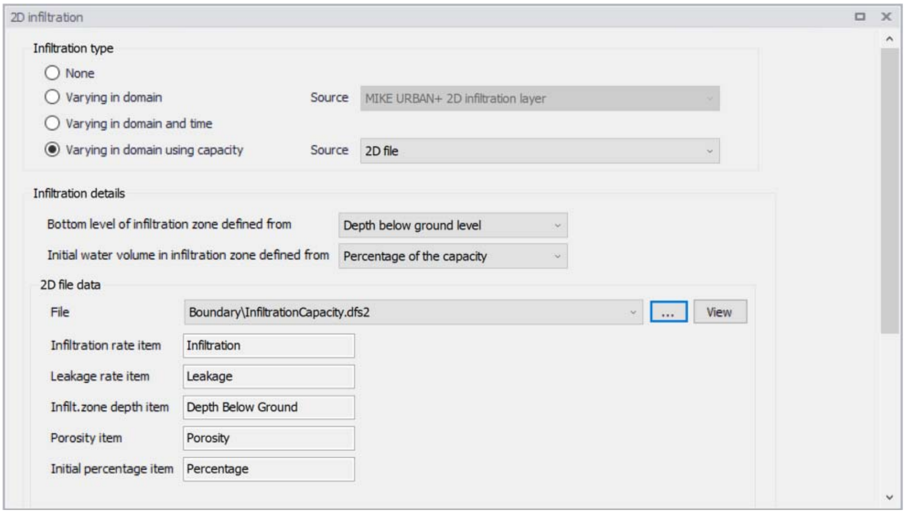


Figure 4.54 Define a 2D file containing data items on infiltration zone properties

Specify the 2D File under the ‘2D file data' section. The ‘**File**’ input box has a drop-down list of valid files loaded as background layers on the Map. Valid files are *.DFS2 or *.DFSU files containing the following data items:

- **Infiltration, Seepage, or Recharge**. Flow between the free surface and infiltration zones.
- **Leakage.** Flow between saturated and unsaturated zones.
- Bottom level of infiltration zone as:
  - **Depth Below Ground**. If ‘Bottom level of infiltration zone defined from’ = 'Depth below ground level'
  - **Elevation**. If ‘Bottom level of infiltration zone defined from’ = 'Specified level'.
- **Porosity, Porosity coefficient, Dimensionless factor, or Specific yield.** Porosity (void fraction) of the infiltration zone.
- Initial water volume in infiltration zone. Initial water volume in Percentage of capacity OR as fraction of the infiltration zone volume
  - **Percentage or Fraction**. (interval: 0-100 [%]). If ‘Initial water volume in infiltration zone define from’ = 'Percentage of the capacity'.
  - **Volumetric Water Content**. (interval: 0-porosity[()]). If ‘Initial water volume in infiltration zone define from’ = 'Percentage of the infiltration zone volume'.

To the right of the File drop-down list, use the ellipsis button to browse to an external 2D file. The 'Open a dfs file' dialog filters on types '2D Grid file (*.DFS2)' or 'Unstructured data file (*.DFSU)', and has a drop-down list from where the item to load may be selected (also see Figure 4.48).

The '**View**' button opens the specified 2D file for viewing/editing. It launches the Grid Editor if a *.DFS2 file has been selected in the editor. If a *.DFSU file has been selected, it opens the file using the Data Manager.

Below the **File** input box are non-editable text boxes displaying the relevant Items in the specified file corresponding to the:

- **Infiltration rate item.** Flow between the free surface and infiltration zones.
- **Leakage rate item**. Flow between saturated and unsaturated zones.
- **Infilt. zone depth/level item**. The label indicates 'depth' if 'Bottom level of infiltration zone defined from'= 'Depth below ground level'. Otherwise, the label uses 'level' if 'Bottom level of infiltration zone defined from'= 'Specified level'.
- **Porosity item**. Porosity (void fraction) of the infiltration zone.
- **Initial percentage item**. Initial water volume in Percentage of capacity (interval: 0-100 [%]) OR as fraction of the infiltration zone volume (interval: 0-porosity[()])

4.6 2D WQ Infiltration

Water quality associated with infiltration conditions may be modelled in MIKE URBAN+ 2D Overland.

These options are available in MIKE URBAN+ if the 2D Overland modules 'Hydrodynamic (HD)' and 'Water quality (AD)' are active AND 2D Infiltration boundaries are included in the model setup.

Infiltration water quality conditions can be included as:

- **Ambient Concentration**. The concentration of the infiltrated water mass is set equal to the concentration of the ambient water in the domain.
- The concentration of the infiltrated water mass may be specified explicitly as:
  - **Uniform**
  - **Varying in Time**
  - **Varying in Domain**
  - **Varying in Domain and Time**

Configure these WQ characteristics individually for each WQ component defined in the setup.

Use the 2D WQ Infiltration editor (Figure 4.55) to configure water quality properties for Infiltration boundaries in the model.

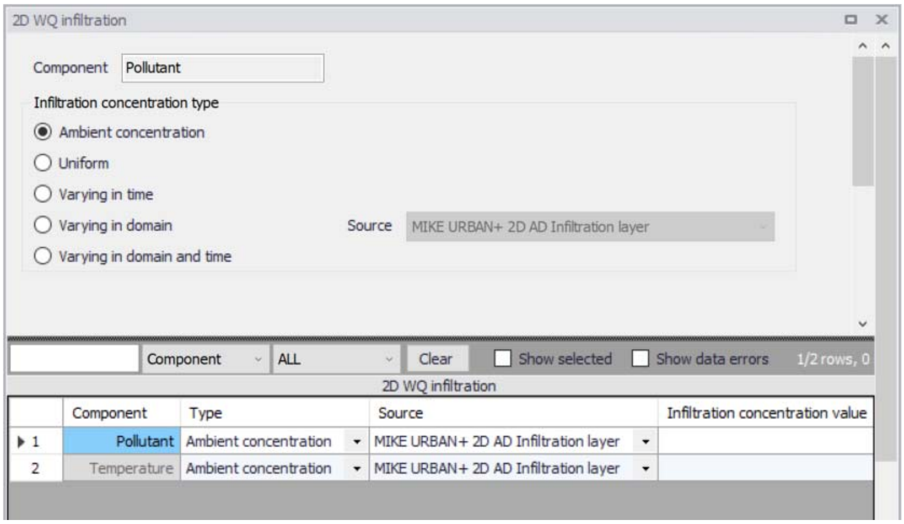


Figure 4.55 The 2D WQ Infiltration editor

The following sections describe data requirements for specifying 2D water quality characteristics for 2D Infiltration boundaries in MIKE URBAN+.

4.6.1 Ambient Concentration

For this type of 2D WQ infiltration input, the concentration of the evaporated water mass is set equal to the concentration of the ambient water in the domain.

Thus, this type of WQ boundary condition does not require additional details on infiltration concentrations.

4.6.2 Uniform

Use this option if infiltration boundary concentrations shall remain constant (in time and domain).

Under the Infiltration Concentration Details section define:

- **Infiltration concentration value**. Specified concentration of the selected/active pollutant component in the infiltration boundary condition.
- **Soft start interval**. Soft start interval during which WQ boundary values are increased from 0 to the specified value in order to avoid shock waves being generated in the computations.

Specify parameters for Uniform 2D WQ Infiltration conditions under the Infiltration Concentration Details section on the editor. A list of WQ components in the model setup is shown in the table overview at the bottom of the editor.

4.6.3 Varying in Time

The case with infiltration concentrations varying in time but constant in domain requires a (*.DFS0) data file containing timeseries data on concentrations (in component unit).

The data must cover the complete simulation period, but the time step of the input data file does not need to be the same as the time step of the hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

4.6.4 Varying in Domain

2D WQ infiltration boundary characteristics for various WQ components may also be modelled as varying in domain (constant in time). Input for this type of 2D WQ boundary may be defined using:

- **MIKE URBAN+ 2D AD Infiltration layer**
- **Background layer**

The data requirements for each are described in the following sections.

MIKE URBAN+ 2D AD Infiltration layer

This option requires definition of '2D AD infiltration' polygon features on the Map.

One may use the Edit Features toolbox from the 2D Overland menu ribbon, or the Flooding Layer Editing Tools toolbar on the Map (Figure 4.56). Use the 'Create' tool to draw features on the Map. Right-click to remove the last vertex added, and double-click to finish polygon drawing.

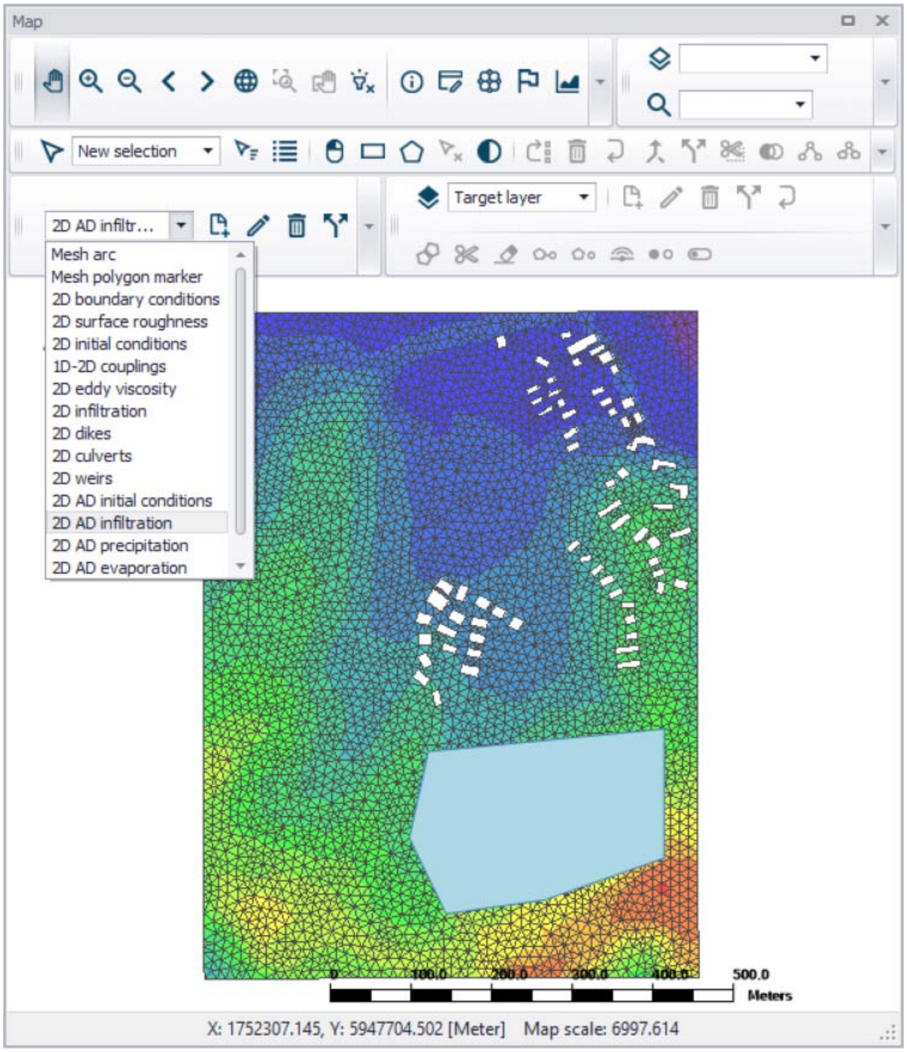


Figure 4.56 Define ‘2D AD infiltration’ layer polygons on the Map

Under the ‘Infiltration Concentration Details’ section on the editor, define the '**Default infiltration concentration**'. This value will be used over areas in the domain for which 2D AD infiltration parameters are not defined via the ‘2D AD infiltration’ polygons on the Map.

Records for 2D AD infiltration polygons drawn on the Map are listed in the table on the 2D WQ Infiltration editor. New records cannot be added from the tabular view. They are only added when drawing a new polygon on the map.

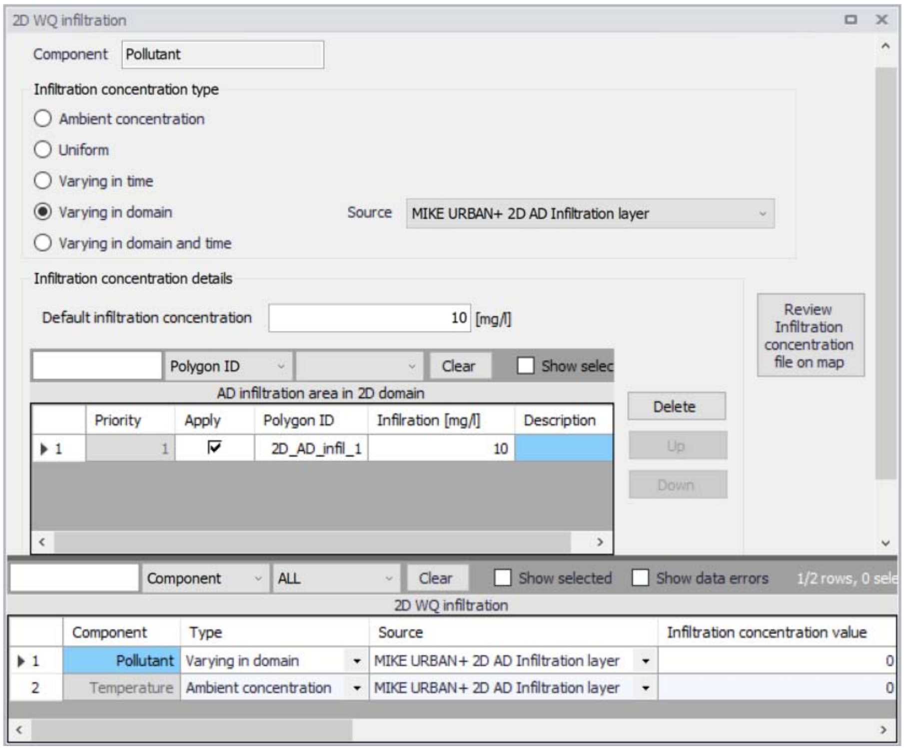


Figure 4.57 Define infiltration concentration properties for each '2D AD infiltration' layer polygon record in the table

Specify the concentration value (related to the active/selected WQ component) for each polygon feature on the table. All items from the table are WQ component-specific. The table has the following columns:

- **Priority**: This is equivalent to the row number. Indicates the order with which values for overlapping features will be prioritized for use in the model.
- **Apply**: Check box allowing activation/deactivation of individual polygon features without deleting the polygon and its properties from the Map.
- **Polygon ID**: This is a text string used to identify the polygon.
- **Infiltration**: This is a numerical field containing the infiltration concentration value assigned to the polygon. The unit is shown in the header (same unit as for the numerical field above the table).
- **Description**: This is an optional text string used to save a free user description for a polygon feature.

Use the '**Review infiltration concentration file on map**' button to generate a 2D file from the infiltration concentration configuration and view the generated file on the Map. The generated 2D file type depends on the defined 2D model type: *.DFS2 grid files for rectangular grid models, and *.DFSU unstructured files for flexible mesh models.

When the file is created, it is then added as a layer on the Map (i.e. it is listed in the tree view and visible on the Map). When the file contains multiple items/components, the component which is drawn on the map is the active component in the overview table. If the file was already loaded as a layer, it is only refreshed (with new data, and with the last active component number).

Background layer

This option requires specification of a background polygon layer to represent domain-varying 2D infiltration concentrations.

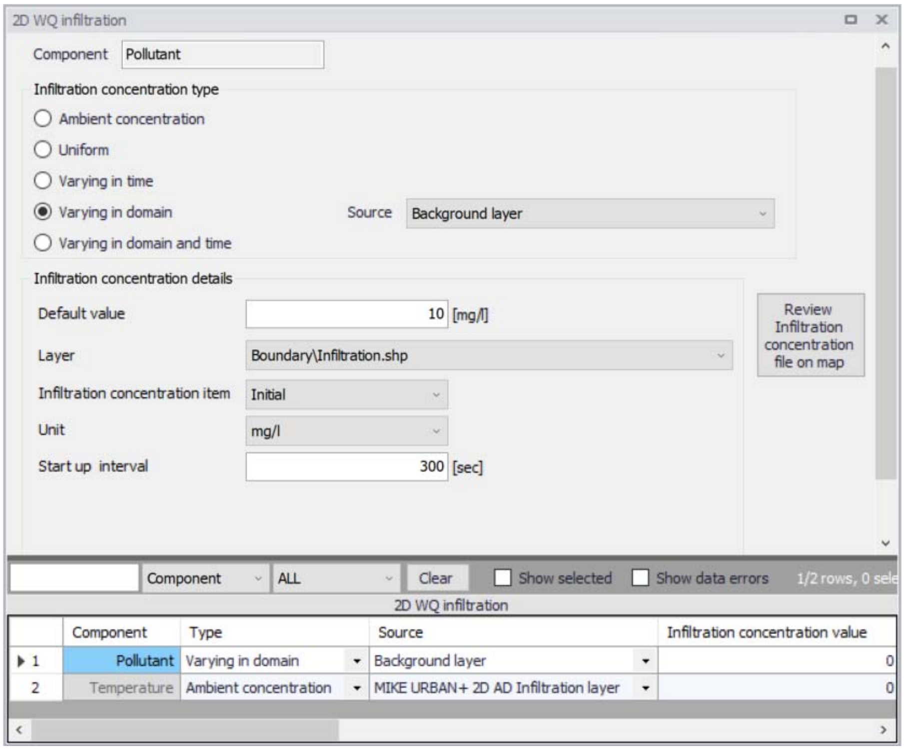


Figure 4.58 Specify domain-varying infiltration concentration parameters based on Background layer data

The ‘Infiltration concentration details’ section on the 2D WQ Evaporation editor displays (Figure 4.58):

- **Default value**: This value will be used as infiltration concentration over areas in the domain for which parameters are not defined with the background layer polygons.

- **Layer**: A drop-down list populated with valid layers loaded as background layers on the Map. A layer is valid if it is a *.TAB, or *.SHP polygon layer.
- **Infiltration concentration item**: Drop-down list populated with the full list of attributes from the selected layer.
- **Unit**: Drop-down list showing unit options depending on the 'Type' of WQ component as specified in the 'WQ components' editor:
  - For Type = 'Pollutant', it shows the list of all supported units for concentration (e.g. mg/l, g/m3, kg/m3, g/l, mu-g/l, pound/(feet US)^3, pound/feet^3, pound/(yard US)^3, pound/yard^3, etc.)
  - For Type = 'Microorganism', it shows the list of all supported units for bacteria / micro-organism concentrations (e.g. million/100 ml, per 100 ml, per liter, etc.)
  - For Type = 'Temperature', it shows the list of all supported units for temperature (e.g. degree Celsius, degree Fahrenheit, degree kelvin, etc.)
  - For Type = 'pH', it shows a single undefined unit [].
  - For Type = 'Salinity', it shows the list of all supported units for salinity (e.g. PSU, per thousand, etc.)
  - For types 'Water age' and 'Water blend', it shows a single undefined unit [].
- **Start up interval**: Interval during which WQ boundary values are increased from 0 to the specified value in order to avoid shock waves in the computations.

The '**Review infiltration concentration file on map**' button generates a 2D file from the infiltration concentration configuration and loads the generated file on the Map. The generated 2D file type depends on the defined 2D model type: *.DFS2 grid files for rectangular grid models, and *.DFSU unstructured files for flexible mesh models.

When the file is created, it is then added as a layer on the Map (i.e. it is listed in the tree view and visible on the Map). When the file contains multiple items/components, the component which is drawn on the map is the active component in the overview table. If the file was already loaded as a layer, it is only refreshed (with new data, and with the last active component number).

4.6.5 Varying in Domain and Time

The case with infiltration concentrations varying both in time and domain requires a 2D file with time-varying concentration values (Figure 4.59). The file must be a 2D unstructured data file (*.DFSU) or a 2D grid data file (*.DFS2).

The area in the 2D data file must cover the model domain. If a *.DFSU file is defined, piecewise constant interpolation is used to map the data to the domain mesh. If a *.DFS2 file is defined, bilinear interpolation is used to map the data to the domain grid.

The data must cover the complete simulation period, although the time step of the input data file does not have to be the same as the time step of the hydrodynamic simulation. A linear interpolation will be applied if the time steps differ.

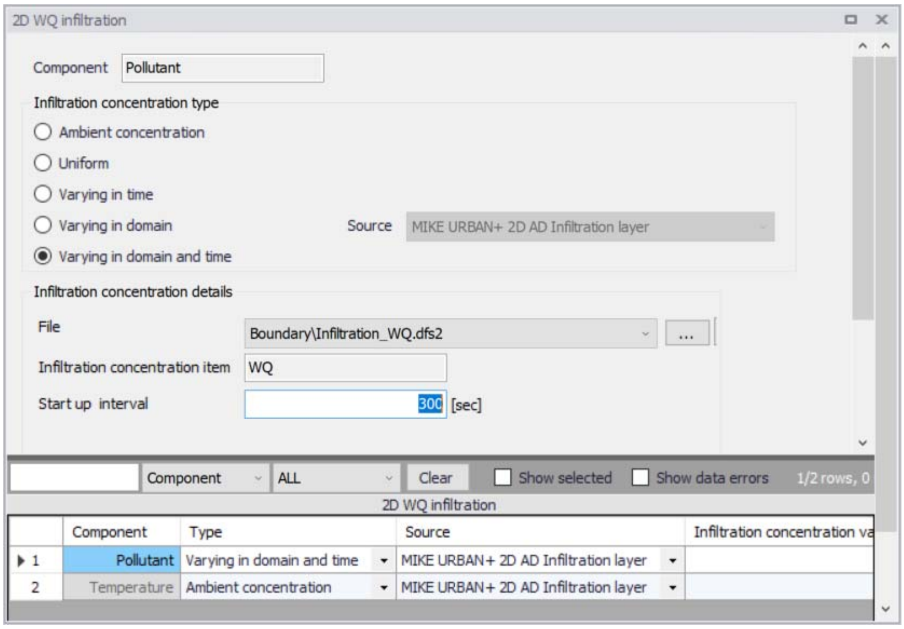


Figure 4.59 Specify the time-varying 2D data file under the Infiltration Concentration Details section on the editor.

On the editor, define the 2D data file under the 'Infiltration Concentration Details' section:

- **File**: A drop-down list of valid 2D layer files on the Map. A file is valid if it is a time-varying *.DFS2 or *.DFSU file.

  The ellipsis button launches the 'Open a dfs file' dialog that is used to locate a valid time-varying 2D file. The dialog filters on types '2D Grid file (*.DFS2)' or 'Unstructured data file (*.DFSU)', and has a drop-down list for Items that may be loaded.

  The 'View' button opens the selected time-varying 2D file for viewing/editing. It launches the Grid Editor if a *.DFS2 file has been selected in the editor. If a *.DFSU file has been selected, it opens the file using the Data Manager.

- **Infiltration concentration item**: Non-editable text box displaying the selected Item from the loaded file.

The database stores one file name and item for each WQ component.

- **Start up interval**: Interval during which WQ boundary values are increased from 0 to the specified value in order to avoid shock waves in the computations.

5 Model Couplings

Through MIKE URBAN+, the benefits of 2D and 1D modelling can be utilised in an integrated model where complex networks and channels represented in 1D are coupled to an overland representation in 2D.

5.1 River model

When 'Coupling to MIKE HYDRO River' is active in the 'Modules' page, the desired MIKE HYDRO model setup file must be selected in the 'River model' page.

The '...' button allows for browsing of the required MIKE HYDRO model file.

Once selected, the main features from the MIKE HYDRO River model are visible on the map in MIKE URBAN+, however they cannot be edited from within MIKE URBAN+.

5.2 1D-2D Couplings

5.2.1 Coupling types

Defining the couplings in a combined 1D - 2D model is done between the stormwater pipes, open channels, streams, rivers and the surface. Different linkage types and options are available and are used as appropriate for the different 1D model components:

- Manhole
- Basin
- Outlet
- Soakaway
- Pump
- Weir
- Natural channel

Coupling definitions can be individually specified in the 1D-2D couplings table, or generated using coupling tools.

5.2.2 Coupling tools

In the ribbon (activated when the 2D Overland Hydrodynamic module is turned on), tools are available to define 1D-2D coupling.

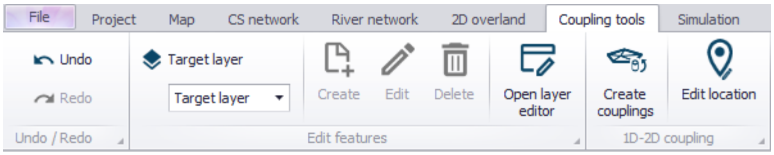


Figure 5.1 Coupling tools ribbon

Create couplings

Click on "Create couplings" to launch a work flow tool that steps through configuring couplings between 1D and 2D model elements. New coupling definitions will be created using this tool, with new rows with unique IDs automatically generated in the 1D-2D couplings table.

Tip: If you intend to couple your model in several smaller steps, you can dock the coupling configuration dialog within your workspace for easier access.

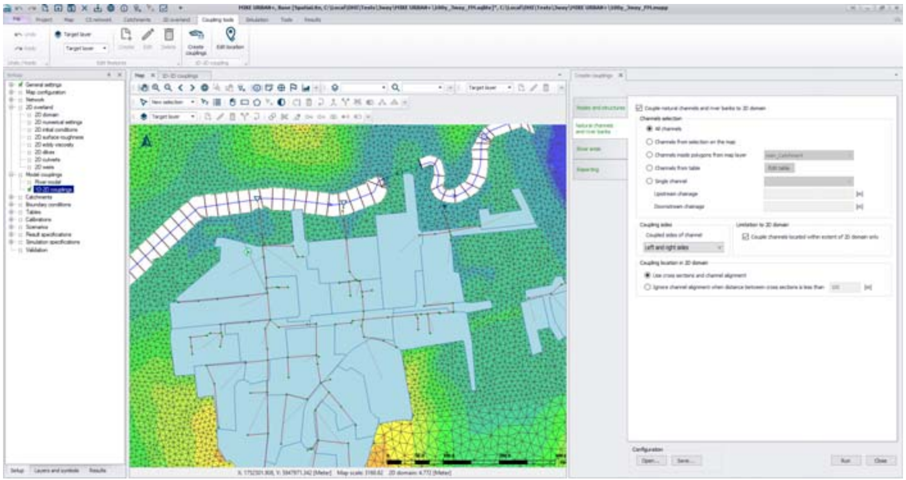


Figure 5.2 Docking the coupling tool in the workspace

Nodes and structures

The first tab in the create couplings tool enable nodes and structures to be coupled based on a selection by ticking on "Couple nodes and structures to 2D domain". If only couplings to other item types than nodes and structures are needed, leave this unticked.

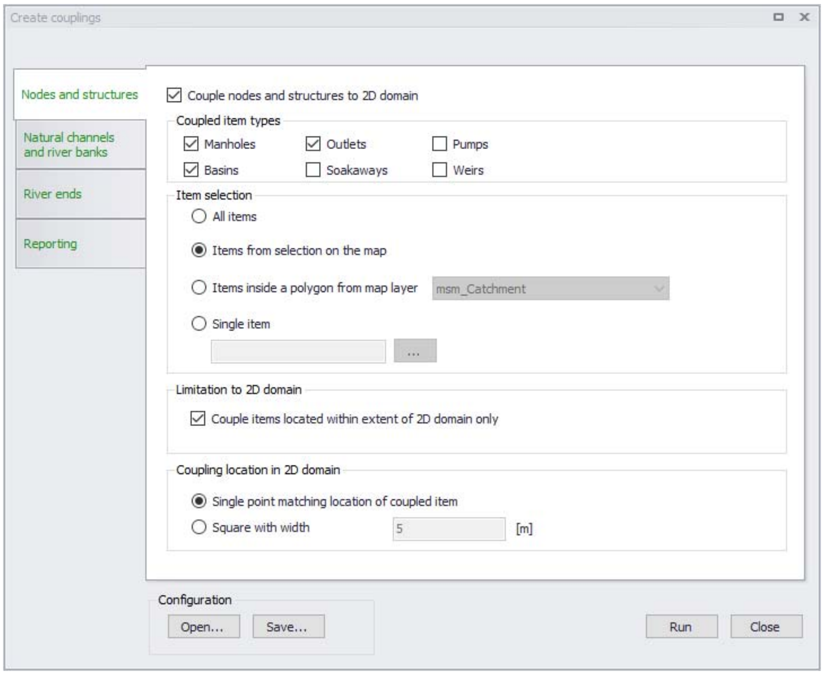


Figure 5.3 Create couplings tool - couple Nodes and structures

Coupled item types

Select the 1D model components to be coupled in this instance.
Tip: every time this create couplings tool is "Run", coupling definitions are added to the 1D-2D couplings table, so can be defined in smaller steps.

Item selection

A number of selection methods are available to define which individual 1D item types are to be coupled.

- All items: creates a coupling definition for all components of the selected item types. E.g. all manholes
- Items from selection on the map: select nodes, pumps or weirs (depending on the selected coupled item types) via the Map view or network table which will be selected in the map). Currently active selection will be coupled.
- Items inside a polygon from map layer: limit the coupling to items that are located within an area (for example a catchment). This polygon layer must be part of the model (msm_Catchment) or loaded into the model as a background layer to be selected.
- Single item: only couple one node or structure. In this case, click on the button "…", specify the item type and select the ID of the node, pump or weir to be coupled.

Limitation to 2D domain

Ensure that only nodes and structures that lie within the 2D domain are considered when coupling (recommended), tick on "Couple items located within extent of 2D domain only.

Coupling location in 2D domain

Select the location/s of the 2D cells to be coupled to the nodes and structures.

- Single point matching location of coupled item - the coupling is defined with the coordinates of the coupled item, except for weirs and pumps for which it is defined at the location of the 'From node'. During the simulation, this item will be coupled to the 2D cell in which it lies.
- Square with width - this will couple the node/structure to 2D cells with centers geographically within the square with specified meters around the node/structure location. In this case more than one 2D cell is coupled to each 1D item, and the flow will be distributed amongst the cells when flowing from the 1D item. This feature is useful if you expect high flows that could cause instabilities.

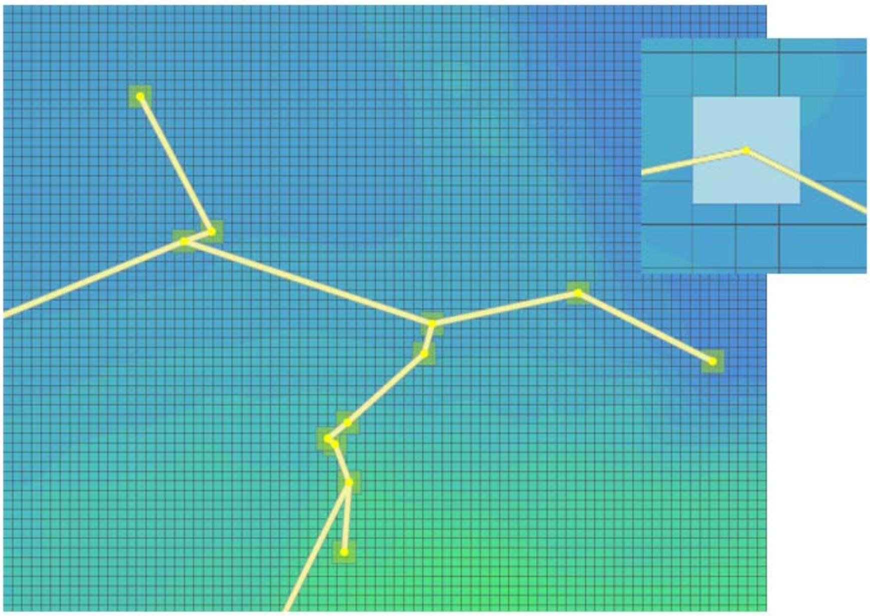


Figure 5.4 Coupling 2D cells by square with width.

Natural channels

The second tab in the create couplings tool enables natural channels and river banks to be coupled by ticking on "Couple natural channels to 2D domain". If only coupling to other item types than natural channels and rivers are needed, leave this unticked.

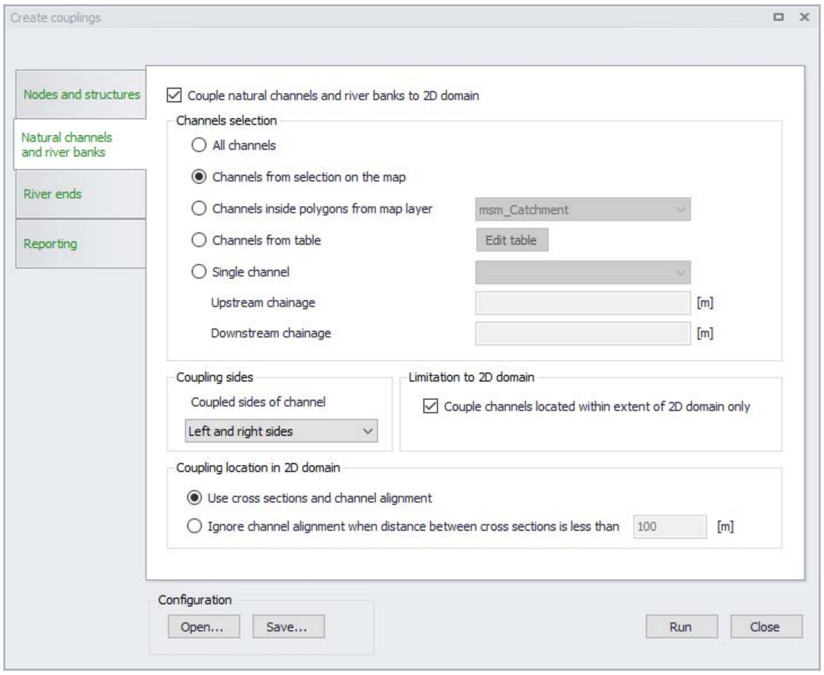


Figure 5.5 Create couplings tool - couple Natural Channels and River Banks

Channels selection

A number of selection methods are available to define which channels are to be coupled.

- All channels: creates a coupling definition for every natural channel (specified in the CS network | Pipes and canals table) or river (specified in River network | Rivers or in MIKE HYDRO).
- Channels from selection on the map: select natural channels via the Map or network tables (which will be shown as selected in the map). The currently active selections will be coupled.
- Channels inside polygons from map layer: limit the coupling to items that are located within an area (for example a catchment). This polygon layer must be part of the model (msm_Catchment) or loaded into the model as a background layer to be selected.
- Channels from table: click on the "Edit table" button to import individual or all natural channels (specified in CS network | Pipes and canals) or river (specified in River network | Rivers or in MIKE HYDRO) and tick on/off the channels to be coupled. Alternatively import a list of channels in text format via "Import from file". ". The text file must contain three columns with the upstream chainage value, the downstream chainage, and the channel ID. The columns should be space-delimited and provided in this order.

- Single channel: only couple one channel. In this case, select the ID of the channel to be coupled as well as which part of the channel to couple (upstream and downstream chainage of the channel).

Coupling sides

The natural channel / river bank coupling allows a string of cells/elements to be laterally linked to a given channel link section (between specified upstream and downstream chainages) or an entire natural channel.

The user must specify whether the actual linkage is to be made along the "right side" of the channel, the "left side", or the "left and right sides". In the latter case, two couplings will be created.

Limitation to 2D domain

To ensure that only channels that lie within the 2D domain are considered when coupling (recommended), tick on "Couple channels located within extent of 2D domain only”. When a channel is partly within the 2D domain but extends beyond the 2D domain borders, the coupling is created only within the extent of the 2D domain.

Coupling location in 2D domain

The determination of which 2D elements faces are to be coupled is made from an automatic selection based on the extent of the cross sections in the channel. Two methods can be used:

- Use cross-sections and channel alignment: The location of the coupling line is based on the extent of the cross sections in the river reach. as well as on the direction of the channel. A vertex is added to this coupling line at each cross section location, and at each change of direction (i.e. at each vertex) of channel. This method ensures that the coupling follows the channel's direction between cross sections, which is convenient when the distance between cross sections is long compared to the channel's width.
- Ignore channel alignment when distance between cross sections is less than a specified distance: The location of the coupling line is solely based on the extent of the cross sections in the river reach. The coupling line has a straight shape between two consecutive cross sections. This method is convenient when the distance between cross sections is small compared to the channel's width, as the number of cross sections is sufficient to correctly define the bank's location.

The determination of which 2D cells/elements are to be coupled is then made from the coupling line. The flow from/to the 1D channel is coupled and distributed to the closest faces of the mesh along this line.

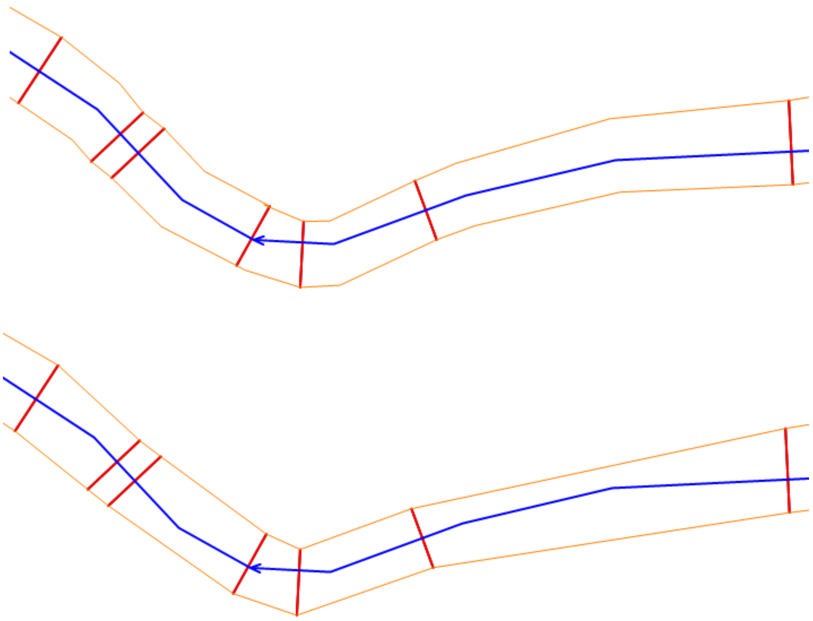


Figure 5.6 Location in 2D domain - Using (at the top) and ignoring (at the bottom) channel alignment

River ends

The third tab in the create couplings tool enables river ends from a MIKE HYDRO River model setup to be coupled by ticking on "Couple river ends to 2D domain". If only coupling to other item types than river ends are needed, leave this unticked.

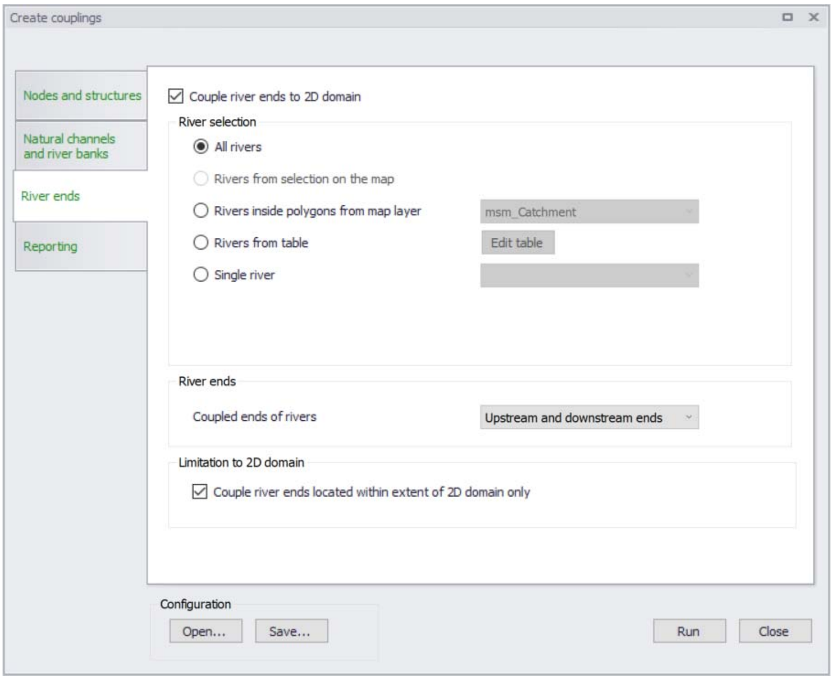


Figure 5.7 Create couplings tool - couple River ends

River selection

A number of selection methods are available to define which rivers are to be coupled.

- All rivers: creates a coupling definition for every river.
- Rivers from selection on the map: select rivers via the Map or Rivers table, and the currently active selection will be coupled (Option not available with MIKE HYDRO).
- Rivers inside polygons from map layer: limit the coupling to rivers that are located within an area (for example a catchment). This polygon layer must be part of the model (msm_Catchment) or loaded into the model as a background layer to be selected.
- Rivers from table: click on the "Edit table" button to select which rivers to be coupled. Alternatively import a list of channels in text format via "Import from file". The text file must contain two columns with a value representing the coupled end (1 for Upstream, 2 for Downstream) and the river ID. The columns should be space-delimited and provided in this order.
- Single river: only couple one river. In this case, select the ID of the river to be coupled.

River ends

The user must specify whether the linkage is to be made at the upstream or downstream end of the rivers, or both. In the latter case, two couplings will be created.

For the "Rivers from table" option, this choice is ignored and is overridden by the selections made in the table.

Limitation to 2D domain

To ensure that only the river ends lying within the 2D domain are considered when coupling (recommended), tick on "Couple river ends located within extent of 2D domain only".

Configuration

'Save…': Configuration settings can be saved in .xml format to be reused later or in another model

'Open…': A previously saved configuration file can be loaded.

Once your configuration is completed, press "Run" to create your couplings.

Reporting

Once a coupling configuration is run, a summary of the coupling will be reported in this section of the tool. This report can be saved for further inspection.

Edit location

Click on "Edit location" to launch a work flow tool that steps through editing existing couplings between 1D and 2D model elements. The tool replicates the functionality of the "Create couplings tool" (see Create couplings, page 136) but unlike the "Create couplings" tool, this tool will update the location (i.e. coordinates) of existing coupling definitions in the 1D-2D couplings table.

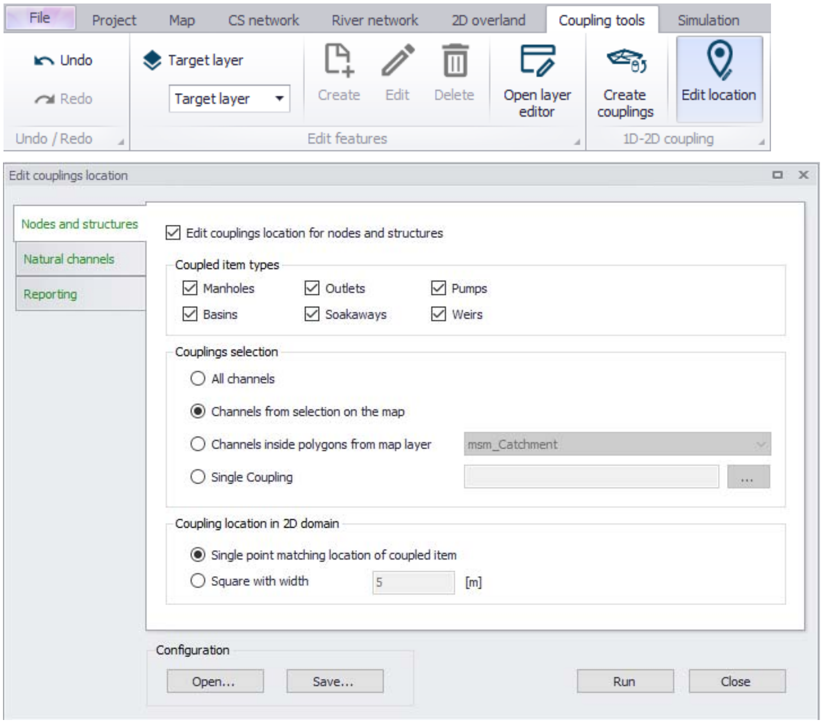
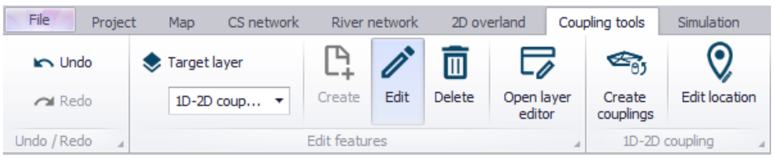


Figure 5.8 Coupling tools - edit location

This tool allows a fast update of the coupling location for multiple items at once. It is especially useful when the flow from the 1D items is high and creates instabilities in the 2D overland model. The tool can be used to distribute the flow from the 1D items to more cells/elements and therefore reduce the velocities in the coupled cells/elements.

Edit, Delete



The location of couplings can be manually edited on the map. Select the '1D-2D couplings' target layer in the 'Coupling tools' tab of the ribbon, and activate the 'Edit' button. Click a coupling on the map to edit its location. For a point coupling, it is possible to drag the coupling to a new location. For a line or polygon coupling, vertices can be added, deleted or moved.

Couplings can also be deleted from the map by activating the 'Delete' button and clicking on the desired coupling.

It is not possible to create new couplings from the map.

5.2.3 1D-2D Couplings table

In the 1D-2D Couplings table it is possible to click on the "Insert" button and individually specify the Location and Attributes for the 1D-2D coupling or edit existing coupling definitions.

Identification

Each coupling definition has a unique ID (editable), the 1D network type (manhole, basin, river bank, pump, etc) that is coupled, and a switch to control which coupling are activated during the simulation using the "Apply" tick box.

When coupling to MIKE HYDRO River, some coupling types can be coupled to either the 2D overland model or to a 1D network model. A 'Couple to' option allows selected whether the item is to be coupled to the collection system network, to the MIKE HYDRO River network or to the 2D overland model.

Location

Specify the location of the 1D and 2D components to be coupled.

Nodes and structures

ID

The node or structure ID to be coupled that can be selected from a drop-down list that corresponds to the selected "Type".

Please note: For the coupling types where the "Inlet area" applies in the "Attributes" tab, the area is recomputed to a default value based on the node's geometry, whenever the node ID is re-selected from this list.

Location in 2D domain

When coupling a node or structure to the 2D overland model, the location of the coupling is defined in the 'Location in 2D domain' table. The default location is a point corresponding to the location of the coupled item. For a weir or pump, it corresponds to the location of its 'From node'. During the simulation, an item coupled to a single point location will be coupled to the 2D cell in which it lies.

The coupling may instead be defined with a polygon, in which case the 'Location in 2D domain' table must contain at least three points /locations. In this case more than one 2D cell is coupled to each 1D item, and the flow will be distributed amongst the cells when flowing from the 1D item. This feature is useful if you expect high flows that could cause instabilities. Points defining the polygon can be added / removed / ordered using the buttons next to the table. In general, using multiple cells or an area as opposed to coupling to only one point will enhance stability.

The 'Square width' option automatically updates the coupling to a polygon with a square shape and with specified width, when the button "Update" is clicked.

Please note: The location is reset to a default location every time the coupled item is changed.

If a node/pump/weir link couples to multiple cells in the 2D overland model, then the ground level in the 2D overland model is taken as the average level from coupled cells/elements. In each cell/element, the coupling acts as a simple source where only the mass is taken into consideration (no momentum).

Location on coupled river

If the collection system network is coupled to both the 2D overland model and to MIKE HYDRO River, coupling an outlet, a pump or a weir requires to specify whether the selected item is coupled to the 2D model or to MIKE HYDRO. If coupled to the 2D overland model, the description from the previous chapter applies. If coupling to MIKE HYDRO, the coupling location is only defined by a chainage along a river, and no other attributes are required.

Natural Channel, River bank, MIKE HYDRO River bank

Coupling types "Natural Channel", "River bank" and "MIKE HYDRO River bank" all describe lateral coupling of a channel, describing overbank spilling towards the floodplain in the 2D overland model. The parameters required for these three coupling types are identical: only the source of the channel differs.

ID

This is the ID of the natural channel or river to be coupled that can be selected from a list.

For the coupling type "Natural Channel" only the channels defined in Network | Pipes and Canals, where Link type = Natural Channel, will be available in the list.

From upstream/To downstream chainage

Specify the section of the channel, from the upstream to the downstream chainages, that is to be coupled to 2D.

Side

Define on which side of the channel over the flow will spill in/out between the 1D and 2D domain.

Note: If both sides of the channel need to be coupled, two couplings need to be specified.

The choice of the side controls the location of the created coupling line in the 2D domain, and therefore controls which cells/elements will be coupled to the 2D domain. The location of the coupling line is based on the markers specified in the cross sections of the channel, where marker 1 is the left bank and marker 3 is the right bank.

Location in 2D domain

One or more cells/elements in the 2D overland topography may be linked to the channel.

The table is automatically populated with the default location, when the channel ID, chainages as well as coupled side are specified.

The following methods are also available to control the 2D element faces to be coupled:

- Insert/Delete: manually insert, edit and delete the vertices of the coupling line controlling which 2D cells/elements will be coupled.
- Up/Down: control the order of the vertices. The coupling line must stretch continuously from upstream to downstream.
- Insert from file: import the x and y coordinates from an external file (XYZ, Shape or Tab files)
- Coupling tools ribbon: the 'Edit' tool in the ribbon can be used to edit the coupling line graphically on the map.

If multiple couplings have to be created, the process can be automated by use of the coupling tool "Create couplings" (see Create couplings page 136).

Tip: If cross sections of the channels in a model set-up are generally wider than the width of the cells/elements in the 2D domain, then it may be appropriate to exclude the cells in the 2D domain lying within the channel bed and hence avoid that any flow calculation is performed by the 2D model in these cells. Thereby, it can be ensured that the water body within the main river is not included twice in the coupled model simulations. Utilise the "Exclude natural channels" tool available via the 2D overland ribbon to exclude the river extent from the 2D computation.

MIKE HYDRO River end

River ID

The river branch's name from the coupled MIKE HYDRO model setup, to be coupled that can be selected from a list.

River end

The selection between the upstream or downstream end of the river, which is being coupled.

Location in 2D domain

When the end of a river from a MIKE HYDRO River model setup is coupled to the 2D overland model, the river end is coupled to a string of elements faces from the 2D domain. The coupled elements faces are the closest faces from the coupling line, defined in the 'Location in 2D domain' table.

The table is automatically populated with the default location corresponding to the end cross section on the river, when the channel ID and Upstream/Downstream end are specified.

The following methods are also available to control the 2D element faces to be coupled:

- Insert/Delete: manually insert, edit and delete the vertices of the coupling line controlling which 2D cells/elements will be coupled.
- Up/Down: control the order of the vertices. The coupling line must stretch continuously from one side of the river to the other.
- Coupling tools ribbon: the 'Edit' tool in the ribbon can be used to edit the coupling line graphically on the map.

If multiple couplings have to be created, the process can be automated by use of the coupling tool "Create couplings" (see Create couplings page 136).

Node ID

When the end of a river from a MIKE HYDRO River model setup is coupled to a node from the collection system network, the node ID must be selected. The node can be selected from the "…" button, or using the "Get closest" which will find the closest node on the collection system network.

Side structure

River ID

The river branch's name from the coupled MIKE HYDRO model setup, where the coupled structure is located.

Structure ID

The ID of the structure being coupled. Note that only the structures with the "Side structure" type on the selected river will show up in the list.

Side

The choice between left and right sides controls on which side of the river the structure will be coupled in the 2D domain.

Length and distance from river

This value is used to control the default location of the coupling line, provided in the "Location in 2D domain" table, as described below. The value is not used in the simulation, i.e. if the table is manually adjusted, only the coordinates from the table are considered for the simulation whereas the "Length and distance from the river" value will be ignored.

Location in 2D domain

When a side structure from a MIKE HYDRO River model setup is coupled to the 2D overland model, the structure is coupled to a string of elements faces from the 2D domain. The coupled elements faces are the closest faces from the coupling line, defined in the 'Location in 2D domain' table.

The table is automatically populated with a default location when the structure ID and "Length and distance from the river" value are specified. The default location is a straight line parallel to the river with a length equal to "Length and distance from the river" value, and placed at a distance from the river also equal to "Length and distance from the river" value.

The following methods are also available to control and edit the 2D element faces to be coupled:

- Insert/Delete: manually insert, edit and delete the vertices of the coupling line controlling which 2D cells/elements will be coupled.
- Up/Down: control the order of the vertices.
- Coupling tools ribbon: the 'Edit' tool in the ribbon can be used to edit the coupling line graphically on the map.

Attributes

Define how water is transferred between the 1D and 2D models through the coupling definition.

Nodes

The coupling to nodes is designed to describe the interaction of water when a manhole, basin or soakage node is overtopped or when overland flow enters and exits a sewer/storm water network.

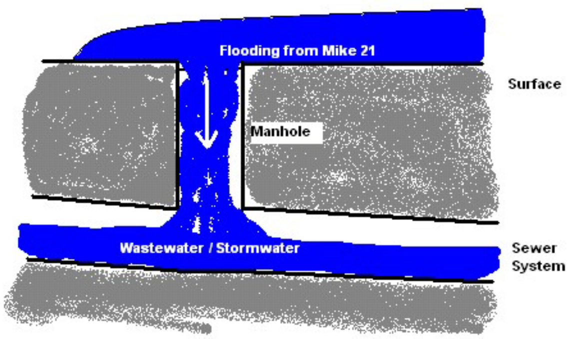


Figure 5.9 Flooding from the overland flow into a non-surcharged sewer system

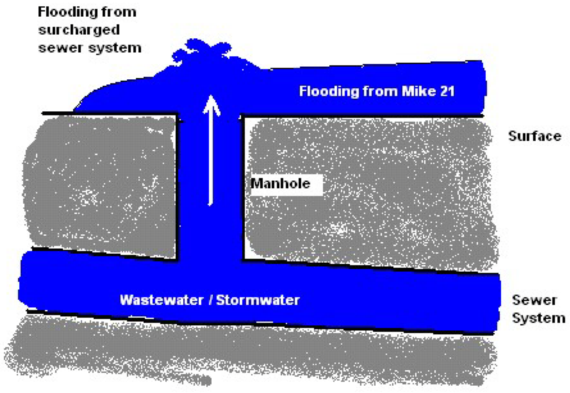


Figure 5.10 Flooding from a surcharged sewer system into the 2D overland topography

The coupling to nodes may also be used for linking a sewer outlet node with the 2D overland topography. For example, to describe the dynamic interaction between a sewer system and a collection basin described using the 2D topography as opposed to describing the basin using an area elevation curve.

Smoothing factor

This parameter introduces an exponential smoothing of the water level values transferred from the 1D to 2D model. A value of 0 indicates no smoothing whereas a value closer to 1 creates strong smoothing in the model. Values are only valid in the range (0 to 1).

- For couplings to manholes, basins and soakaways in the sewer model the smoothing is applied to the node water level and the 2D water level used for calculating the interacting discharge.
- For outlets in the sewer model the smoothing is applied to the 2D water level transferred to the collection system network.

The higher the value the greater the smoothing will be. The parameter impacts the dynamics by smoothing out steep gradients (in time) through the couplings. In general, the exponential smoothing factor should be adjusted when a structure exhibits unstable behaviour (oscillates wildly).

A high smoothing factor will cause a time lag in the transfer of the values from one model component to the other model component.

Delta depth for dampening

This parameter indicates the water level difference from which the calculated discharge should be progressively suppressed. The delta depth may be used for stabilizing set-ups where the instability is caused by a small difference in the water level in the two coupled models.

If a value of 0.0 is given then no suppression is applied. When a positive value is given, the discharge is multiplied by a specified suppression factor between 0 and 1. When the water level difference between the node and 2D overland cell/s is greater than the delta depth then the factor is 1. For water level differences *dh* less than the delta depth, the suppression factor is:

$$\text{Suppressionfactor} = 1 - \left(\frac{DeltaDepth - dh}{DeltaDepth}\right)^2 \tag{5.1}$$

Max Q

The coupling gives the option of placing an upper limit on the flow. This is only available for couplings to manholes, basins and soakaways. Care must be taken with this option. A simulation indicating that the max value is in effect constantly may be hiding an oscillating flow being overridden by a fixed value.

When a time series boundary condition is applied as a network load to a collection system node that is also coupled to the 2D domain, as network load it is fully applied to the network (not to the coupling), i.e. all of the discharge enters the network first and only then spills onto 2D domain.

However, when a rainfall runoff catchment load is applied to a node that is also coupled to the 2D domain, the runoff discharge is applied to the coupling which results in the discharge to the network being restricted by the max Q parameter. The remaining discharge is applied to the 2D domain.

Flow description

The exchange of water at the coupling can be computed in four different ways:

- Orifice equation:
  The flow between 1D and 2D is governed by a standard orifice equation.
  - Discharge coefficient: a non-dimensional factor that may be used to scale the orifice flow. By default the parameter should be close to 1.
  - Inlet area: describes the flow exchange between the urban and the 2D grid/mesh. The greater the cross sectional area the greater the conveyance capacity of the coupling. This parameter corresponds physically to the area of the manhole cover or stormwater inlet. The default inlet area is recomputed based on the manhole's diameter, or set equal to the last (highest) As value from the geometry table for a basin, when the node ID is selected. The max Q value must be considered in conjunction with this parameter.

- Weir equation:
  The flow between 1D and 2D is described through a weir equation.
  - Discharge coefficient: a non-dimensional factor that may be used to scale the weir flow. By default the parameter should be close to 1.

- Weir width: describes the flow exchange between the urban and the 2D grid/mesh based on the weir formula. This parameter corresponds physically to the circumference of the manhole cover.

- Exponential function:
  The flow is governed by a simple exponential function.

  An increase in the exponent factor has a strong impact on the discharge. An increase in this value will generate a larger flow for a certain water level difference in the urban and the 2D model.

- Curb inlet function:

  This function is used when the flow between 1D and 2D represents flow transferred at a grate or inlet from a surface overland flow network to the sub-surface pipe network. The transfer capacity of the connection is specified as a DQ-relation (tabular data type). When water is transferred to the surface from the sewer pipe network the flow is calculated with the orifice equation.

  - Discharge coefficient: as per the orifice flow description
  - Inlet area: as per the orifice flow description
  - Freeboard: defines a critical water level (Invert level + Freeboard) at the connection node in the pipe network below which the defined DQ relation applies. For submerged and reverse flow (surcharge), the transfer capacity of the connection reverts to a standard orifice relationship.
  - DQ relation ID: click on the button "…" to select from predefined DQ relationships specified in Tables | Curves and relations. The DQ relation specifies the depth-based capacity curve for the curb inlet. Values must be monotonically increasing in depth and discharge starting at (0,0). For depths in excess of the maximum value specified in the last row of the table, the last corresponding discharge is used. Positive discharge is considered to flow from 2D into 1D (overland to sewer network).

Pumps and Weirs

The coupling to the 2D overland model is also applicable to situations where the sewer system is discharging into the surrounding area through a pump or over a weir. In these cases the pump or the weir must be defined as having no downstream node in the 1D network.

Smoothing factor

This parameter introduces an exponential smoothing of the water level values transferred from one model to the other model. A value of 0 indicates no smoothing whereas a value closer to 1 creates strong smoothing in the model. Values are only valid in the range (0 to 1).

Natural channels, River bank, MIKE HYDRO River bank

The transfer of flow between a string of 2D cells/elements laterally linked to a given reach in the channel is described by the following attributes:

Smoothing factor

This parameter introduces an exponential smoothing of the water level values transferred from 1D to 2D. A value of 0 indicates no smoothing whereas a value closer to 1 creates strong smoothing in the model. Values are only valid in the range (0 to 1).

The greater the value the greater the smoothing will be. The parameter impacts the dynamics by smoothing out steep gradients (in time) through the couplings. In general, the exponential smoothing factor should be adjusted when there is unstable behaviour in the coupled flow (oscillates wildly).

A high smoothing factor will cause a time lag in the transfer of the values from one model component to the other model component.

Delta depth for dampening

This parameter indicates the water level difference from which the discharge gradients should be progressively suppressed. The delta depth may be used for stabilizing set-ups where the instability is caused by a small difference in the water level in the two coupled models.

If a value of 0.0 is given then no suppression is applied and the model may experience numerical instability. The suppression varies with the water level difference and is only active when the water level difference is less than the depth tolerance.

Weir description

Defines the type of weir flow calculation to be performed between 1D and 2D.

- Weir type: Either the Villemonte or Honma formula is used to describe the transfer of flow between the 1D channel and the 2D mesh/grid.
- Weir coefficient: required for the weir formulas
- Crest source: determines where the crest level geometry information comes from that is used for the lateral coupling, i.e. where the levee height should be extracted:
  - Highest - Takes the maximum of the 2D topography and the 1D channel's cross section markers (marker 1 for left coupling side and 3 for right coupling side).
  - 2D Topography: The levee height is defined by the topography in the 2D topography (for stability reasons the drying depth is added to the topography values when defining the levee height).
  - Table: A table of values can be created or edited that defines the crest level profile. Chainages along the channel and crest level are specified. This table is saved in Tables | Curves and relations.

– External file: Information on link elevations as a function of distance along the coupling is introduced through an external dfs1-file. This file can be time varying and hence, can be used to simulate e.g. embankment levees changes and breaches during a simulation. Browse to an existing file, edit the profile in a profile editor, or create a new profile using the button "...".

Please note: The crest level is not allowed to be below the 2D topography nor below the bed level in the channel. Thus if the selection is such that this occurs MIKE URBAN+ will use the limiting value.

MIKE HYDRO River end, Side structure

When coupling the end of a river branch or a side structure from MIKE HYDRO to the 2D overland model, the following attributes must be specified.

- Smoothing factor: This parameter introduces an exponential smoothing of the water level values transferred from one model to the other model. A value of 0 indicates no smoothing whereas a value closer to 1 creates strong smoothing in the model. Values are only valid in the range (0 to 1).
- Flow distribution: If the distribution is non-uniform, flow from the river is distributed into the coupled cells/elements according to their respective water depth. This is considered appropriate for couplings to natural terrain or channels, but may not always be applicable around structures. If the distribution is uniform, the discharge is distributed uniformly in the coupled cells/elements.

6 Water Quality

MIKE URBAN+ provides several modules for the simulation of and water quality for both urban catchments surfaces and sewer systems. Since pollutants are carried by sediment, sediment transport process and water quality in sewer systems are closely interconnected. This is important for understanding phenomena like the first flush effect, which can only be simulated with a description of the temporal and spatial distribution of sediment deposits on the catchment surface and in the sewer system. MIKE 1D can model complex mechanisms such as Surface Runoff Quality (SRQ), Pipe Sediment Transport (ST), Pipe Advection-Dispersion (AD), and Biological Processes (BP) by means of the coupling to the MIKE ECO Lab. Output from this, such as pollutant graphs from combined sewer overflows, can then be applied directly to DHI's receiving waters models MIKE 1D and MIKE 21. Using the integrated urban water modelling environment of MIKE URBAN+ allows assessment of water quality for the water bodies receiving these sewer overflows, such as rivers, streams, lakes and coastal waters.

6.1 WQ Components

The Advection-Dispersion model can be used for calculation of the transport of dissolved or suspended substances, age of water, blend in percent between two sources, and for modelling of water temperature variation within the sewer network. The model is based on the one-dimensional transport equations for dissolved material. The equations reflect two transport mechanisms: the advective (or convective) transport with the mean flow velocity and the dispersive transport due to concentration gradients in the water. The transport equations are solved by use of an implicit finite difference scheme, which is fully time and space centred, in order to minimize the numerical dispersion. The main assumptions of the model are:

- The considered substance is completely mixed over the cross-sections. This implies that a source/sink term is considered to mix instantaneously over the cross-section;
- The substance is conservative or subject to a first order reaction (linear decay);
- Fick's diffusion law can be applied, i.e. the dispersive transport is proportional to the concentration gradient.

Special considerations have been given to the transport at manholes and other structures. More information on the technical background of the model is given in the ”Pollution Transport Reference” manual.

The Advection-Dispersion model requires two types of data: time series of concentrations at the model boundaries and data for full definition of the components to be modeled, e.g. initial concentrations, dispersion coefficients and decay rates.

6.1.1 Water Quality | AD Components

Each of the components (substances) to be included in the Advection-Dispersion computations must be specified in the WQ Components. The naming of component is absolutely flexible, and no "reserved" or "standard" component names are prescribed.

The user can specified several AD components to be included in the simulation using the MIKE 1D engine.

The specified pollution components can be declared as 'Dissolved', 'Suspended', 'Total', 'Bacteria', 'Temperature' ‘Water Age’, ‘Water Blend’ and 'Other'. This categorization is needed for correct handling of units: mass per volume (dissolved, suspended, total, other), counts per volume (bacteria), water age in hours, water blend in %, and degrees (temperature). When working with water-quality model, this categorization is further supplemented with other parameters, in order to apply the components in the WQ model properly. Practically, each of the specified components can be connected to a 'standard' component in the WQ module. By these means the WQ module 'knows' how to treat each component.

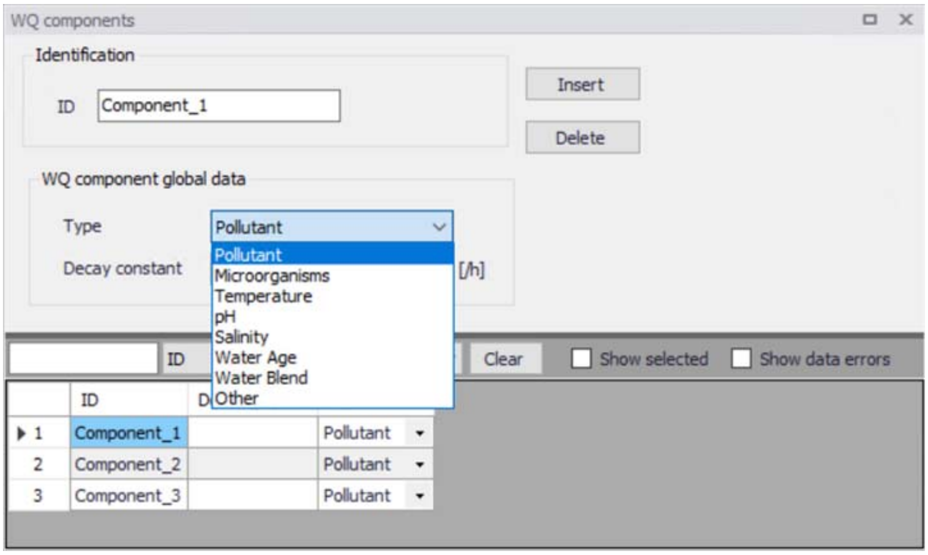


Figure 6.1 WQ Components

For each component, specification of an initial condition and decay coefficient can be specified. The decay coefficient cannot be given for water age and water blend type. If the specification of the initial concentration for a certain component has been omitted, a zero concentration is automatically applied. Water blend concentration must always be given as a number between 0 and 100 percent, and the sum of the two blend components must add up to 100 percent.

Local initial conditions for dissolved or suspended pollutants, water blend, water age and temperature can be specified under “Water Quality|AD Initial Conditions'.

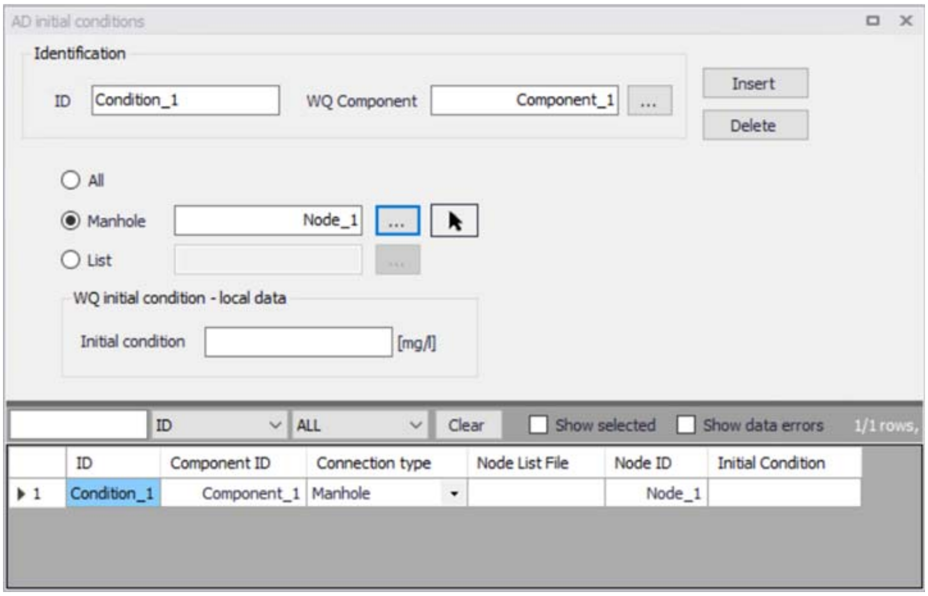


Figure 6.2 AD Initial Conditions

The initial conditions can be specified for individual nodes, a list of nodes or for all of them. The initial conditions in the connected conduits is calculated by linear interpolation of the concentrations specified in the upstream and downstream nodes.

By defining decay constants, non-conservative components can be specified. For such non-conservative component the concentration is assumed to decay according to the first order expression:

$$\frac{dC}{dt} = KC \tag{6.1}$$

where:

K = the decay coefficient (h^{-1})

C = the concentration

The decay constant is defined as an uniform decay over the entire model.

The AD model can be run with the components specifications only. In this case all model specific parameters (decay constant, dispersion coefficient, initial concentration) as well as boundary conditions are set to zero.

Please not that the sum of the two blend components must always add up to 100.

6.1.2 Water Quality | AD Dispersion

The dispersion coefficient is specified as a function of the flow velocity. The function is given as:

$$D = au^b \tag{6.2}$$

where:

D = the dispersion coefficient (m^2/s),

a = the dispersion factor,

u = the flow velocity (m/s),

b = a dimensionless exponent.

If the exponent is set equal to zero, then the dispersion coefficient is constant and independent of the flow velocity. The unit for the dispersion factor will then be m^2/s. If the exponent is 1, i.e. the dispersion coefficient is a linear function of the flow velocity, then the unit of the dispersion factor will be meter, and the dispersion factor will in this case be equal to what is generally termed the dispersivity. It is possible to specify values of the minimum and the maximum dispersion coefficients, in order to limit the range of the dispersion coefficient calculated during the simulation.

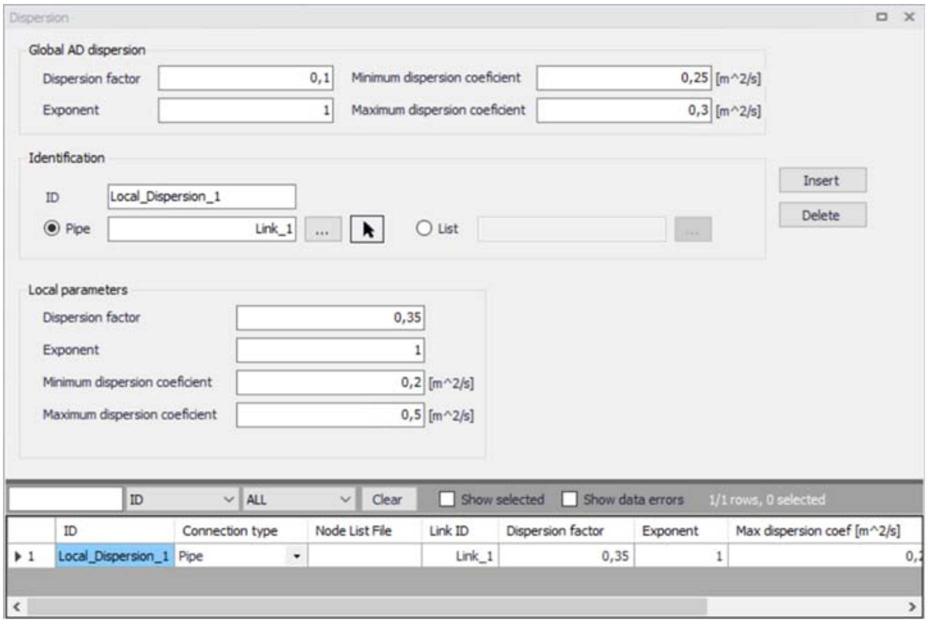


Figure 6.3 AD dispersion Global and Local Parameters

The dispersion coefficient can be given either globally or locally. The global description will be used at all locations except for those pipes where local conditions have been specified in the menu. In the example above the global statement indicates that a constant dispersion coefficient of 2.00 m2/s is applicable, and independent of the flow velocity (the exponent is zero).

For two conduits (see the example below), the dispersion coefficients specified locally, 'overrule' the global specification and prescribe a constant dispersion coefficient of 4 m2/s.

6.2 2D Decay

The Advection/Dispersion (AD) module provides the advection/dispersion basis for the computations performed in the 2D Overland Transport Module and the MIKE ECO Lab Module (ecological modelling).

The advection/dispersion module simulates the spreading of dissolved substances subject to advection and dispersion processes in lakes, estuaries and coastal regions. The effects and facilities include:

- linear decay
- heat dissipation to the atmosphere

The Advection/Dispersion (AD) module can be applied to a wide range of hydraulic and related phenomena. This includes the spreading of dissolved substances such as:

- salt
- heat
- coliform bacteria
- xenobiotic compounds

Typical applications include water quality studies.

Decay

You can specify the decay factors in one of two ways:

- as a constant value applied at all time steps
- by means of time series data file, which will be interpolated to match simulation time steps

Heat Dissipation

You can specify the reference temperature in one of two ways:

- as a constant value applied at all time steps
- by means of a time series data file, which will be interpolated to match simulation time steps

6.3 2D AD Initial Conditions

The format of the initial concentration (in component unit) for each component can be specified as

- Constant (in domain)
- Varying in domain

For the case with varying in domain you have to prepare a data file containing the concentration (in component unit) before you set up the hydrodynamic simulation.

The file must be a 2D unstructured data file (dfsu) or a 2D grid data file (dfs2).

The area in the data file must cover the model area. If a dfsu-file is used piecewise constant interpolation is used to map the data. If a dfs2-file is used bilinear interpolation is used to map the data. In case the input data file contains a single time step, this field is used. In case the file contains several time steps, e.g. from the results of a previous simulation, the actual starting time of the simulation is used to interpolate the field in time. Therefore the starting time must be between the start and end time of the file.

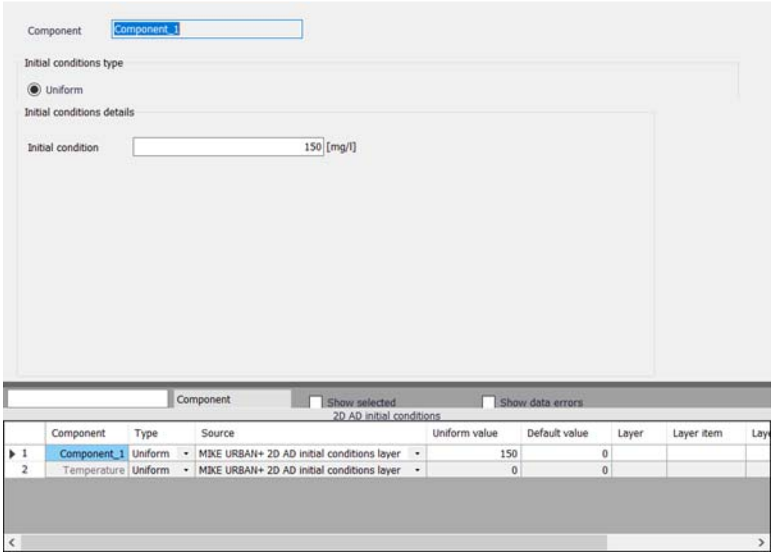


Figure 6.4 2D AD initial conditions

6.4 2D AD Dispersion

The horizontal dispersion can be formulated in three different ways:

- No dispersion
- Dispersion coefficient formulation
- Scaled eddy viscosity formulation

Selecting the dispersion coefficient formulation you must specify the dispersion coefficient.

Using the scaled eddy viscosity formulation the dispersion coefficient is calculated as the eddy viscosity used in solution of the flow equations multiplied by a scaling factor.

Dispersion coefficient

When selecting the Dispersion coefficient option the format of the dispersion coefficient can be specified as:

- Constant (in both time and domain)
- Varying in domain

For the case with dispersion coefficient varying in domain you have to prepare a data file containing the dispersion coefficient, before you set up the hydrodynamic simulation. The file must be a 2D unstructured data file (dfsu) or a 2D grid file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used, piecewise constant interpolation is used to map the data. If a dfs2-file is used, bilinear interpolation is used to map the data.

Scaled eddy viscosity

When selecting Scaled eddy viscosity option the format of the scaling factor can be specified as:

- Constant
- Varying in domain

For the case with values varying in domain you have to prepare a data file containing the scaling factor, before you set up the hydrodynamic simulation. The file must be a 2D unstructured data file (dfsu) or a 2D data grid file (dfs2). The area in the data file must cover the model area. If a dfsu-file is used, piecewise constant interpolation is used to map the data. If a dfs2-file is used, bilinear interpolation is used to map the data.

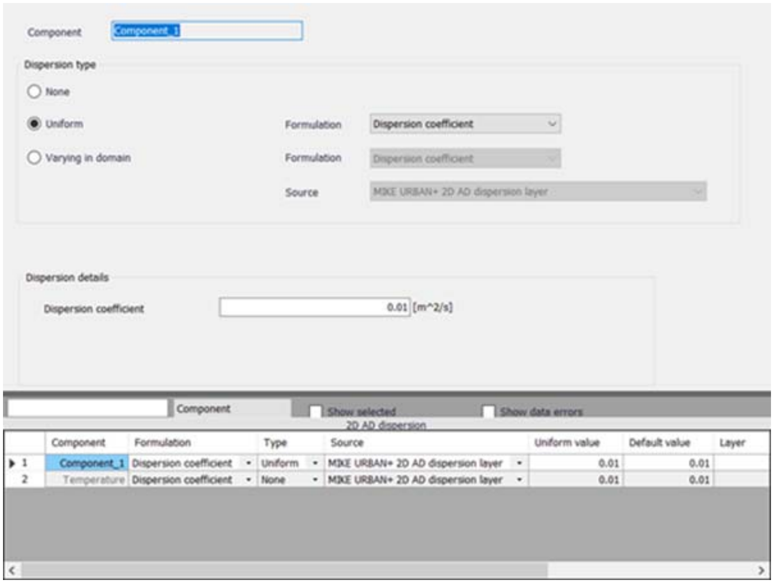


Figure 6.5 2D AD Dispersion

7 Tables

The Tables Section in MIKE URBAN+ holds data for the following tabular data types:

- Curves and Relations
- Materials

7.1 Curves and Relations

In Tables|Curves and Relations (Figure 7.1), a number of tabular data used in other data dialogs are specified. These different types of tabular data are:

- Capacity Curve QH (used for Pumps)
- Capacity Curve QdH (used for Pumps)
- Pump Acceleration Curve (used for Pumps)
- Regulation Qmax(H)
- Regulation Qmax(dH)
- QH Relation (when specifying a QH relation for a node)
- Valve Rating Curve
- Time-Area Curve (used in Time-Area runoff model)
- Removal Efficiency (used for the efficiency curve for removal weirs)
- DQ Relation (used for Curb Inlets)
- QQ Relation (used for Curb Inlets and On Grade Captures)
- Capacity Curve QdH & Power
- Runoff Pollutants (used for SWQ)
- Basin Geometry (used for Basins)
- RTC (used for RTC Action Set Points)
- RTC Time (used for RTC Action Set Points)
- Undefined (general placeholder)

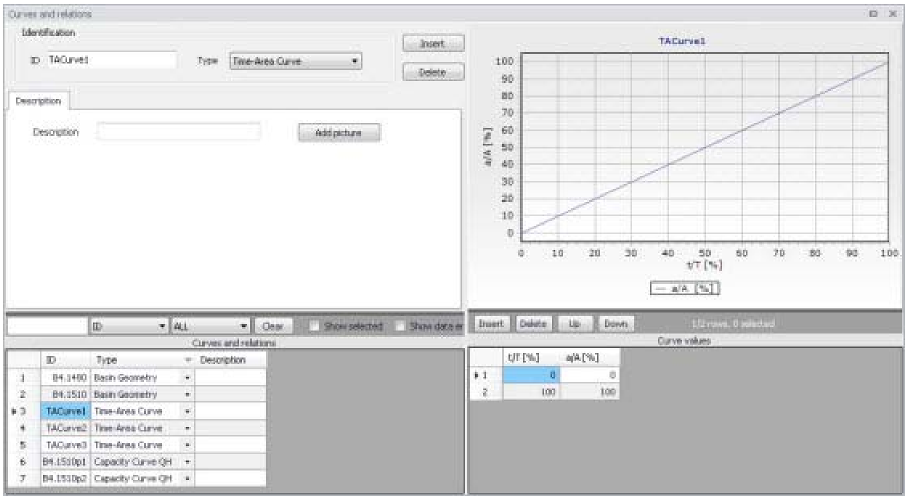


Figure 7.1 Curves and Relations Editor

There are 3 pre-defined Time-Area Curves in the database (TACurve1, TACurve2 and TACurve3), which should not be deleted.

Additional curves and relations are inserted under the Curves and Relations list (i.e. the primary table on the lower left corner of the editor) using the 'Insert' button at the top of the editor (Figure 7.2).

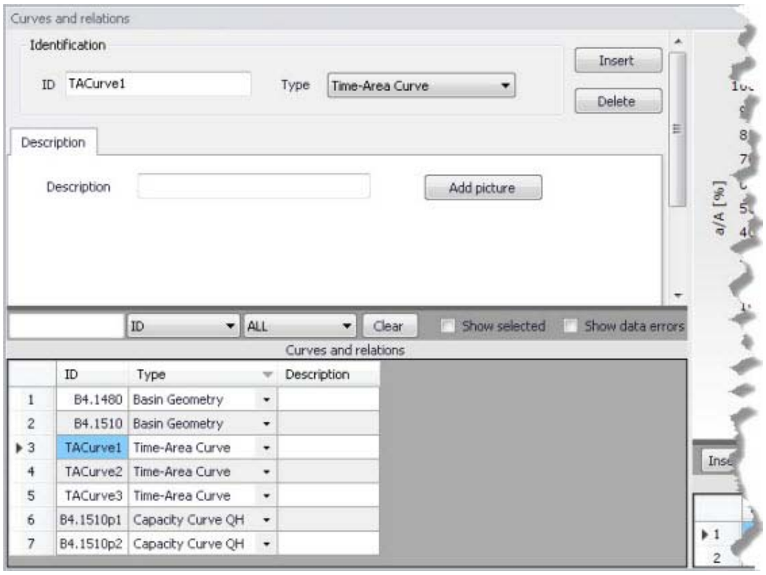


Figure 7.2 Primary table with the Curves and Relations list on the lower left side of the Curves and Relations Editor

After inserting a new tabular data item, define the corresponding data values under the Curve Values table (i.e. secondary table to the right of primary table) (Figure 7.3). Secondary table parameters/columns that should be filled vary depending on the curve and relation type.

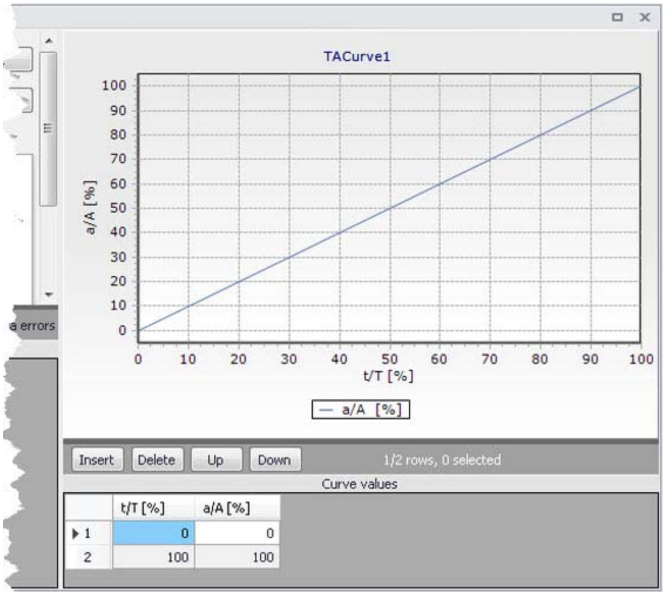


Figure 7.3 Secondary table containing Curves Values on the lower right side of the Curves and Relations Editor. Also shown is the tabular data plot above the secondary Curve Values table.

A plot of the tabular data is also shown on the upper right corner of the editor (Figure 7.3).

Table 7.1 Overview of Curves and Relations Editor attributes (Table ms_Tab)

Edit field	Description	Used or required by simulations	Field name in datastructure
ID	Tabular data identi-fier	Yes	MUID
Type	Dropdown menu for selecting tabular data type	Yes	TypeNo
Description	User's descriptive information on the tabular data	Optional	Description

7.1.1 Capacity Curves

It is possible to define two types of capacity curves in MIKE URBAN+; both are used to define pump operation.

The capacity curve can be a 'Capacity Curve QH' relation (for screw pumps) or 'Capacity Curve QdH' relation (for differential head pumps).

‘H' is the absolute water level in the pump's wet well (i.e. From Node), and 'dH' is the water level difference between the (downstream) 'To Node' and (upstream) 'From Node' locations.

If an offset is specified, this will be added to the capacity curve relation.

Also note that one may specify a pump capacity curve with energy consumption (i.e. Capacity Curve QdH & Power).

7.1.2 Pump Acceleration Curve

Pumps may be RTC controlled. For PID-controlled RTC pumps, the acceleration of a pump can be specified as dependent on the actual flow. This pump acceleration curve is then specified as a number of ‘dQ, dQ/dt’ values.

7.1.3 Regulation Curves Qmax(H) and Qmax(dH)

The regulation curves Qmax(H) and Qmax(dH) are used in the regulation of the maximum discharge in links. The regulation can either be a maximum discharge as a function of the water level in a user-specified node, or a maximum discharge as a function of the water level difference between two user-specified nodes.

7.1.4 QH Relation

QH relations can be used for outlets. Using a QH relation in an outlet means that you specify the discharge out of the outlet based on the water level in the outlet.

7.1.5 Valve Rating Curve

A valve is a functional relation between two nodes of a sewer network. The valve rating curve specifies the relationship between the valve opening (%) and resistance (k).

7.1.6 Time-Area Curve

The Time-Area curve is used in the Time-Area runoff model. A Time-Area curve represents the percentage contributing part of the catchment surface as a function of time.

MIKE URBAN+ comes with three default Time-Area curves - TACurve1, TACurve2 and TACurve3 - applicable for rectangular, divergent and convergent catchments, respectively.

One can define other Time-Area curves. Each Time-Area value table must start with a pair of values (0,0) and must end with a pair of values representing the whole catchment contribution. MIKE URBAN+ maintains T-A curves in percent (%), and the last pair of values in the table must be (100,100).

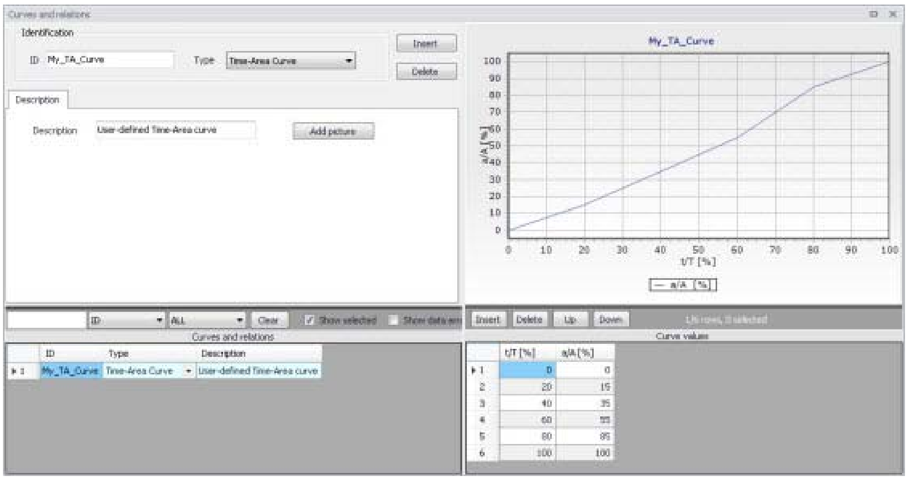


Figure 7.4 Example of user-defined Time-Area curve

7.1.7 Removal Efficiency

There are three methods available for the removal of sediments in weirs. In one of these methods you specify the relation between discharge towards the weir and the removal efficiency, i.e. the efficiency curve. The removal efficiency is hence a function of Q and the efficiency (dimensionless 1/1).

7.1.8 Curb Inlet DQ and QQ Relations

Two curve types can be specified for two different types of Curb Inlets:

- DQ Relation (depth-discharge relation specified in the Curb Inlets dialog)
- QQ Relation (Qapproach-Qcapture relation specified in the On Grade Captures editor)

The DQ relation specifies the depth-based capacity curve for a SAG Type Curb Inlet. Values must be monotonously increasing in depth and discharge and starting at (0,0). For depths in excess of the maximum value specified in the last row of the table, the last corresponding discharge value is used.

The QQ relation specifies the relationship between approach flow in the overland flow network (Qapp) and the captured flow at the connection node for an On Grade Type Curb Inlet (Qcap). Values must be monotonously increasing and starting at (0,0). For approach discharges in excess of the maximum value specified in the last row of the table, the last corresponding capture discharge value is used.

7.1.9 Capacity Curve QdH & Power

If specific power consumption in relation to pump levels is known, it is possible to include this in the model using the 'Capacity Curve QdH & Power' curve type.

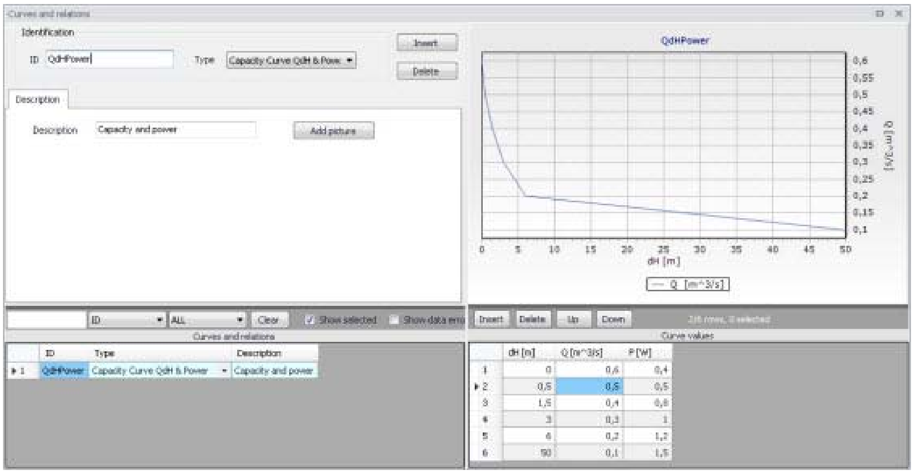


Figure 7.5 Pump capacity curve including power consumption

After the simulation with a 'Capacity Curve QdH & Power' the summary will contain information on the power consumption during the simulation period.

7.1.10 Runoff Pollutants

This type of table is used in surface water quality (SWQ) boundary conditions as a way to define the Temporal Variation of surface stormwater loads as well as RDI stormwater loads.

The table serves as a lookup table for the boundary condition, where corresponding concentration values are determined based on runoff intensity. The tabular data set shall contain values for runoff intensity (i.e. the runoff divided by the total catchment area) and corresponding concentrations.

7.1.11 Basin Geometry

Basin geometries are tabulated area-elevation functions. One specifies values for the parameters H, Ac, and As.

Ac is the cross-section area perpendicular to the main flow direction in the basin, which is used to calculate the velocity. As is the surface area of the basin (used to calculate the volume). Both parameters are specified as functions of the water level, H, in the basin.

The H-column for the basin geometry can start at any value, e.g. 0 for interpretation of H as depth in the basin. MIKE URBAN+ associates the first H-value to the bottom level of the node. This means that the same geometry can be reused in several places in the model.

The maximum level before flooding at a basin is either the highest H value of the geometry or the ground level. If the top of the basin geometry is below the ground level, the specified basin geometry is extended with additional points to allow for flooding.

7.1.12 RTC

RTC tables are lookup tables defining the functional relation between an actual input value (e.g. sensor reading or difference between sensor readings, etc.) and the set point value (or setting). The tabulated values are linearly interpolated between defined relations.

7.1.13 RTC Time

RTC Time tables are lookup tables explicitly defining the set-point value (or setting) for particular time periods (i.e. date/time). The tabulated values are linearly interpolated between defined values.

7.1.14 Undefined Type

The Undefined table type is an extra generic type of table used as a placeholder for potential future functionality.

7.2 Materials

In MIKE URBAN+, a link is characterised by material, which determines the Manning friction coefficient (Manning), the Colebrook-White coefficient (EQ Roughness), or Hazen-Williams coefficient (H-W Coefficient) for the conduit.

It is optional to use either the default roughness values for specific materials or local values.

Specification of the different kind of materials and roughness coefficients is done through the Materials editor (Tables | Materials).

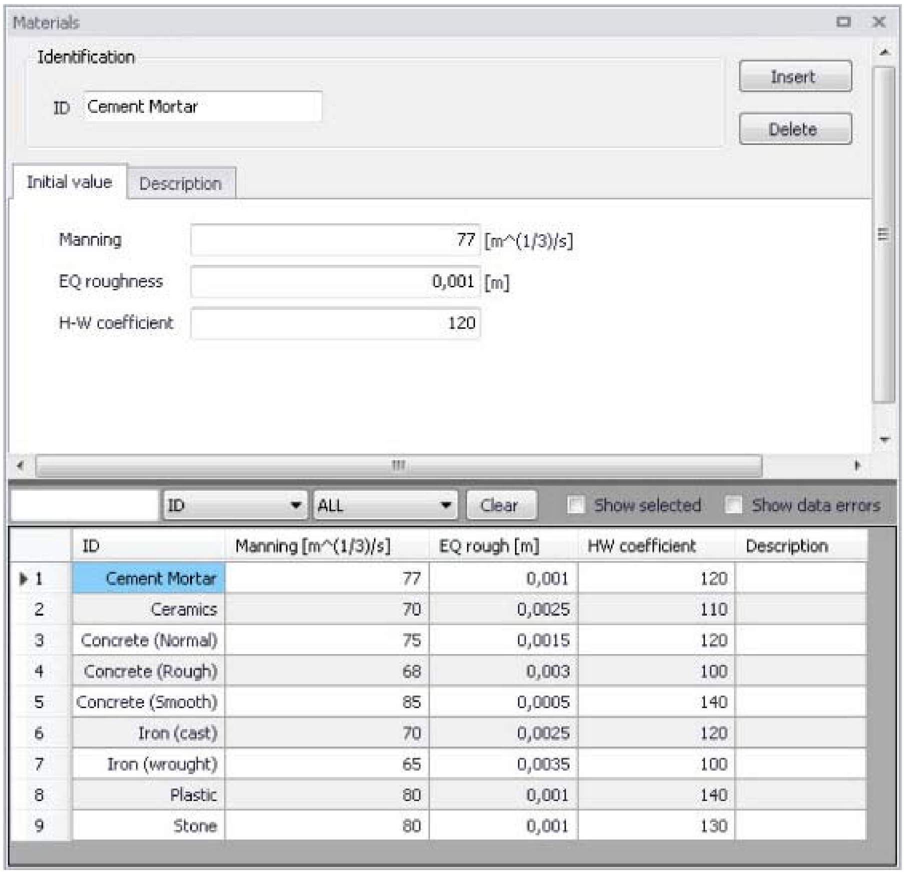


Figure 7.6 Materials Editor

MIKE URBAN+ has the following pre-defined Material types with friction loss properties:

- Cement Mortar
- Ceramics
- Concrete (Normal)
- Concrete (Rough)
- Concrete (Smooth)
- Iron (cast)
- Iron (wrought)

- Plastic
- Stone

Table 7.2 Overview of the Materials Editor attributes (Table ms_Material)

Edit field	Description	Used or required by simulations	Field name in datastructure
ID	Material type ID	Yes	MUID
Manning	Manning roughness value	Yes If 'Manning Explicit' or 'Manning Implicit' is used	Manning
EQ Roughness	Equivalent rough-ness	Yes If 'Colebrook-White' formulation is used	EQRough
H-W Coefficient	Hazen-Williams roughness coeffi-cient	Yes If 'Hazen-Williams' is used	HWCoef
Description	User's descriptive information on the material	Optional	Description

8 Scenarios

The water distribution and wastewater collection data models are commonly used for the system performance analysis and in the planning process. The complexity of the involved systems, the various uncertainties about the future conditions and usually huge costs associated with maintenance, rehabilitation and development necessitate a thorough investigation of alternative system configurations in a search for the technically feasible, environmentally sound and economically efficient solution. These alternative configurations - scenarios - may differ by the system's physical layout, loading conditions, operational strategies, etc. Various projects, such as development of a Sewerage Master Plan, Wastewater Transportation Strategy, an Overflow Abatement Strategy, etc. would typically result in a large number of scenarios, either representing alternative system configurations at a given time and/or representing the system at various development stages. Test of each scenario against the prescribed legislation or the standards of service that the authorities provide requires a numerical model on its own.

These scenarios are always related to each other through the common origin ('Base') and the differences typically epitomize a smaller part of the total data. Moreover, scenarios representing a development of the system through time are subject to the dependencies propagating along with the timeline. Analysis of the scenarios as separate projects creates major inconveniences, such as:

- Large number of models, even when differences between them are minor
- Missing the efficient overview over the entire set of solutions
- Inability to maintain the existing dependencies between the individual scenarios automatically. Thus, the updating of the models with additional information requires editing of multiple files to change the same element, e.g. if a pipe diameter is found to have been incorrectly registered in the GIS database, it will have to be updated multiple times in each of the scenario project files.

8.1 What are Scenarios

The MIKE URBAN+ Scenarios is a user interface for a set of MIKE URBAN+ features, enabling the definition, organisation management and reporting of alternative scenarios, such as:

- Augmentation of existing trunk sewer mains;
- Increased wastewater loading from increased population;
- Increased water demands from increased population;
- Alternative design loads, e.g. rainfall-runoff of different return period;
- Alternative alignment of sewer and storm mains;

- Building of a new sewer trunk and water supply mains in order to cater for a new development area within the same MIKE URBAN+ project.

8.2 Design of the MIKE URBAN+ Scenario Manager

The MIKE URBAN+ Scenario group is based on the concept of Data Groups, Alternatives and Scenarios. In this context, a Data Group is a set of database tables which form a meaningful set. E.g. all database tables containing collection system network data belong to the data group 'Network Data'. Every database table relevant for the scenario manager is included in one of the Data Groups.

Each Data Group can appear in the MIKE URBAN project in any number of Alternatives. The initial alternative is named with a default name 'Base'. Any further alternative is created upon user request and gets a user-specified name. The Alternatives for a certain data group are organised in a tree-like structure, where dependencies propagate along the branches -from the 'parent' to all the 'heirs' i.e. 'children'.

A scenario represents a complete set of consistent data, featuring the system configuration for a given situation. In other words, a scenario contains one alternative of each Data Group. Actually, individual alternatives are used as building blocks for constructing scenarios. A moderate number of data groups (eight for collection system and nine for water distribution) allows for a manageable structure of scenarios, while ensuring the high level of flexibility.

The initial scenario is named with a default name 'Base', and consists of the 'Base' alternative of each data group. Any further scenario is created upon user request and gets a user-specified name. The scenarios are organised in a tree-like structure of 'parents' and 'children'.

Alternatives

The alternatives represent the components of the scenarios. The various alternatives contain the actual data belonging to a certain data group. Actually, each subsequent alternative only contains the information on the differences relative to its immediate 'parent', while the rest of data is inherited from the 'parent' through the principles of inheritance.

Grouping of various alternatives belonging to different data groups into scenarios is sometimes subject to limitations, because the data groups have not been formed on the basis of data independency, but rather following the logical data grouping. E.g. the alternative of the 'Catchment connections' CS data group, which specifies a catchment connection to node 'A', cannot be used with the alternative of the 'Network data' data group where node 'A' has been renamed or deleted. Obviously, the catchment would remain disconnected.

Collection System Alternatives

For Collection Systems, the scenarios are composed of the following data groups:

- Network
- Loads and Boundary
- Catchment & Hydrological
- WQ
- Operational RTC
- LTS
- Profiles and Curves

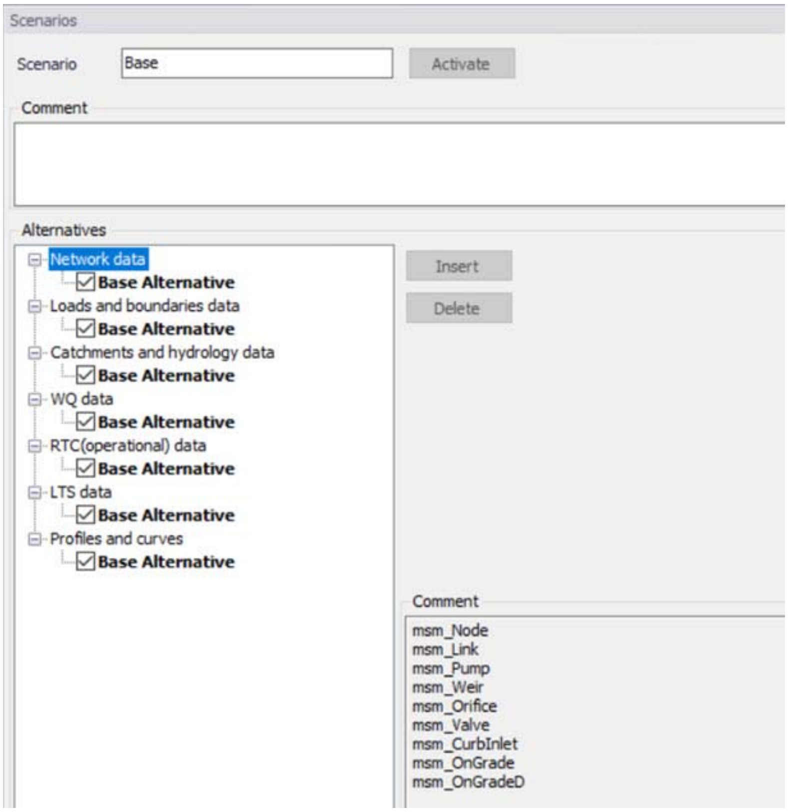


Figure 8.1 Scenarios Alternatives for Collection System

Water Distribution Alternatives

For Water Distribution Systems input data can be grouped the following way, corresponding to the different types of available alternatives:

- Network
- Water Demands
- Control
- Water Quality
- Pattern and Curves

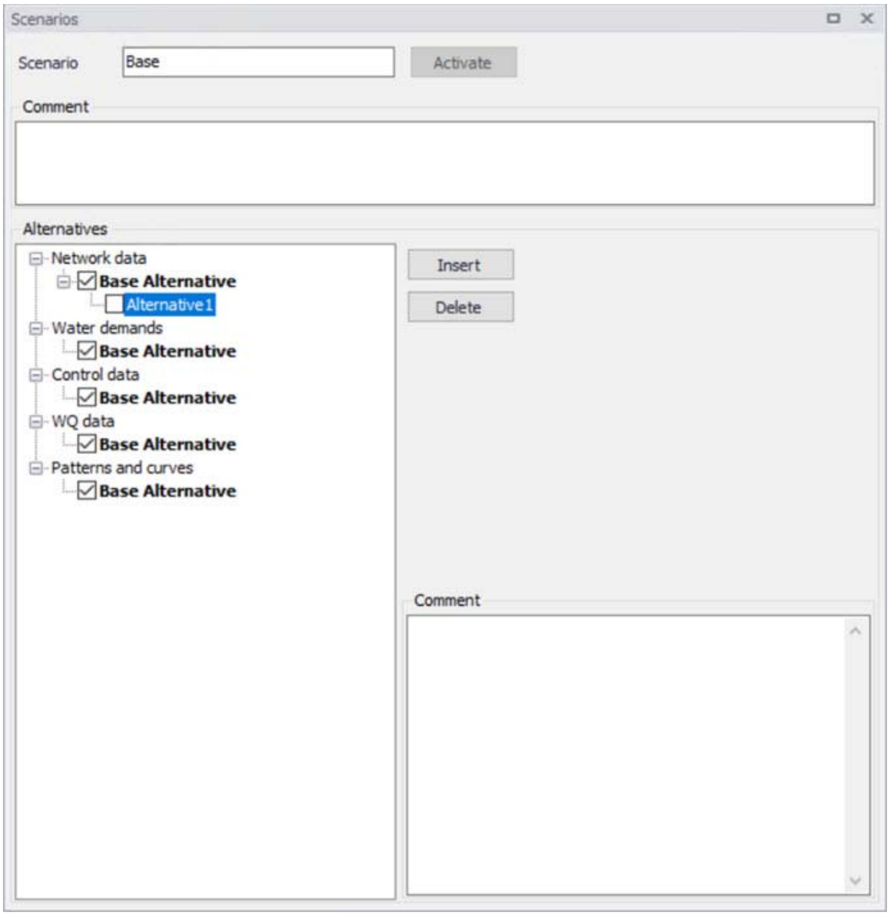


Figure 8.2 Scenarios Alternatives for Water Distribution

Inheritance principles

With the inheritance from 'parent' alternatives to 'child' alternatives, some specific items must be kept in mind.

- Making a change to an alternative will affect all descendent ('child') alternatives of that alternative. This means that it will impact all the scenarios where either the alternative or the children of that alternative are applied. This also ensures that if one value needs updating it will be updated in all the scenarios where the alternative is applied (e.g. if a pipe diameter is found to have been incorrectly registered in the GIS data during the course of a project then the pipe diameter can be changed one place only, regardless of the number of scenarios and alternatives that reference to this alternative).
- The chain of inheritance for a certain data record stops where any change (or delete) of that element has occurred in earlier work. E.g. if a bottom level of a node 'A' has been edited in some child alternative, some later update of the bottom level in 'Base' will only propagate through the alternative tree until the alternative containing the old change.
- Adding an element (e.g. a node) in the 'parent' with an ID that already exists in one or more of its descendants ('children') will overwrite the content of the 'child' element
- If adding an element (e.g. pump/link) in the parent that cannot be added to all the children (because some parts may have been deleted/changed there), the element is added where possible and omitted elsewhere.

8.3 Managing Scenarios and Alternatives

The Scenario Manager has two parts:

- The Scenario part
- The Alternatives part

The scenario part is for creating, editing and managing scenarios, while the alternatives part is for creating, editing and managing alternatives.

Under Scenarios in the tree view one can “right-click” to get the option of creating a new child scenario.

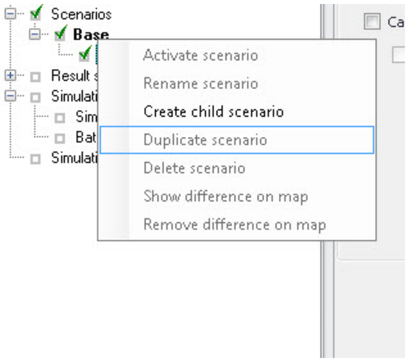


Figure 8.3 Create a new child scenario

Right-clicking on a child scenario enables the options to activate, rename, duplicate, delete, create a new child scenario and visualize the differences in the map.

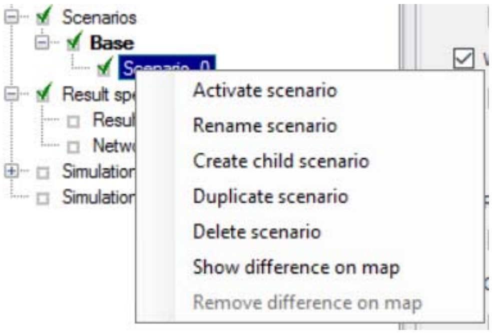


Figure 8.4 Management of child scenarios

Scenarios

The scenario part is used for creating, editing, and managing scenarios. Per default there will one built-in scenario, i.e. the Base scenario. The Base scenario cannot be edited or deleted. An unlimited number of additional scenarios can then be added to cover the various 'What if' scenarios.

Create a Child Scenario

This option adds a scenario that is a child of the highlighted (not to be confused with the active/current scenario), i.e. to begin with the alternatives of a new scenario will be that of the highlighted scenario. A name for the new scenario is suggested by default. The name can be changed through the option of Rename scenario or by editing directly in the ID field in the Scenario editor.

Remove

The remove option will delete the highlighted scenario. The Base scenario cannot be deleted. Note that deleting a scenario will not delete any data as the alternatives hold the data (the scenarios just refer to alternatives). The comments for the scenario being deleted, however, will also be deleted.

Rename

The rename option will make the scenario name active so it can be easily renamed.

Activate

The activate button will load the scenario, i.e. the project data is manipulated so that all editors contain the appropriate data. Depending on the size of the project this may take some time.

Alternatives

Alternatives can be edited only if the appropriate scenario is made active. Alternatives can, however, be added regardless of the active scenario. When a scenario is loaded, the project data is manipulated so that all editors contain the appropriate data.

Alternatives can be activated, inserted and deleted in the editor through the visual buttons.

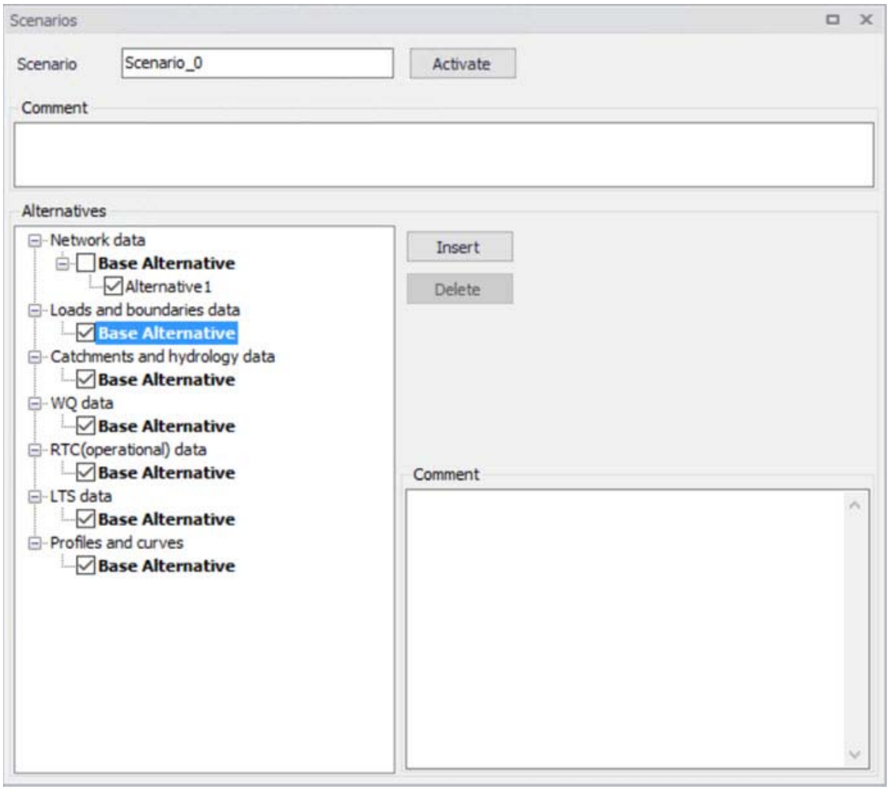


Figure 8.5 Alternative options

8.4 How to Start Working with Scenarios

Creating alternatives and scenarios

- Right click on Scenario in the Tree view
- Create a child scenario
- Select the alternative group that you wish to add an alternative to and press the ‘Insert’ button
- You can now rename it and/or continue to make alternatives
- Select the alternative and click on Activate

The activated scenario is displayed in bold font. Equally, all the alternatives belonging to the active scenario, are displayed in bold in the 'Alternatives' page.

Both the data tables, the graphics and any MIKE URBAN+ tools work ONLY with the data belonging to the currently active scenario. Access to the data belonging to other scenarios is not possible through MIKE URBAN+ interface.

9 Result Specifications

MIKE URBAN+ allows flexible generation of model simulation result files and summaries. Various result file setups may be specified in the Result Specifications section, which may then be used in a simulation as needed by the user.

Result files obtained from simulations may be customised with respect to the number of result files generated, file types, spatial extent of saved results, and types of result items included in the files.

9.1 Result Files

The Result Files editor (Figure 9.1) provides a facility for viewing and specifying result file setups in a project. The types or results available depend on the active Modules for the project (General Settings | Modules).

Result setups are a mix of Default and User-specified results.

Default results

The editor is initially filled with Default result setups. A Default set is shown for each active Module in the project. These records are distinguished by the "Default_" prefix in their IDs.

The following table shows an overview of the various Default result files that are possible in MIKE URBAN+.

Table 9.1 Overview of possible Default result files in MU+

Default ID	Model Type	Format	Content Type
Default_Surface_runoff	Catchments	.RES1D	Surface runoff
Default_RDII	Catchments	.RES1D	RDII
Default_Storm_water_quality	Catchments	.RES1D	Storm water quality
Default_Storm_water_sediments	Catchments	.RES1D	Storm water sediments
Default_LIDs	Catchments	.DFS0 (currently hard-coded)	LIDs
Default_Catchment_discharge	Catchments	.RES1D	Catchment discharge
Default_Catchment_discharge_quality	Catchments	.RES1D	Catchment discharge quality

Table 9.1 Overview of possible Default result files in MU+

Default ID	Model Type	Format	Content Type
Default_Network_HD	Network	.RES1D	Hydrodynamic
Default_Network_RTC	Network	.RES1D	Real time control
Default_Network_AD	Network	.RES1D	Pollution transport
Default_Network_MIKE_ECOLab	Network	.RES1D	MIKE ECO Lab
Default_LTS_extreme_statistics	Network	.RES1D	LTS extreme statistics
Default_LTS_chronological_statistics	Network	.RES1D	LTS chronological statistics
Default_2D_overland	2D Overland	DFSU / DFS2	2D area
Default_2D_overland_AD	2D Overland	DFSU / DFS2	2D area, AD
Default_2D_Flood_statistics	2D Overland	DFSU / DFS2	2D flood statistics
Default_2D_Volume_balance	2D Overland	DFS0	Volume balance

Default result sets include results in all model elements, saving values for the basic calculation parameters.

Default result sets may be modified with respect to:

- File format: .RES1D or .DFS0 file format.
- Result items: Calculated parameters to be included in the file. Only basic result items are initially included in Default result setups.

Note that the saving locations may not be modified for Default results, and results will always be saved in all elements.

Also note that for some content types, only one file format is allowed and cannot be changed.

User-specified results

The user may also define custom result file setups according to their modelling needs.

The following properties may be customised for user-specified results:

- File format

- Location: Spatial extent or network elements for which results are saved in the file.
- Result items: Calculated parameters to be included in the file.

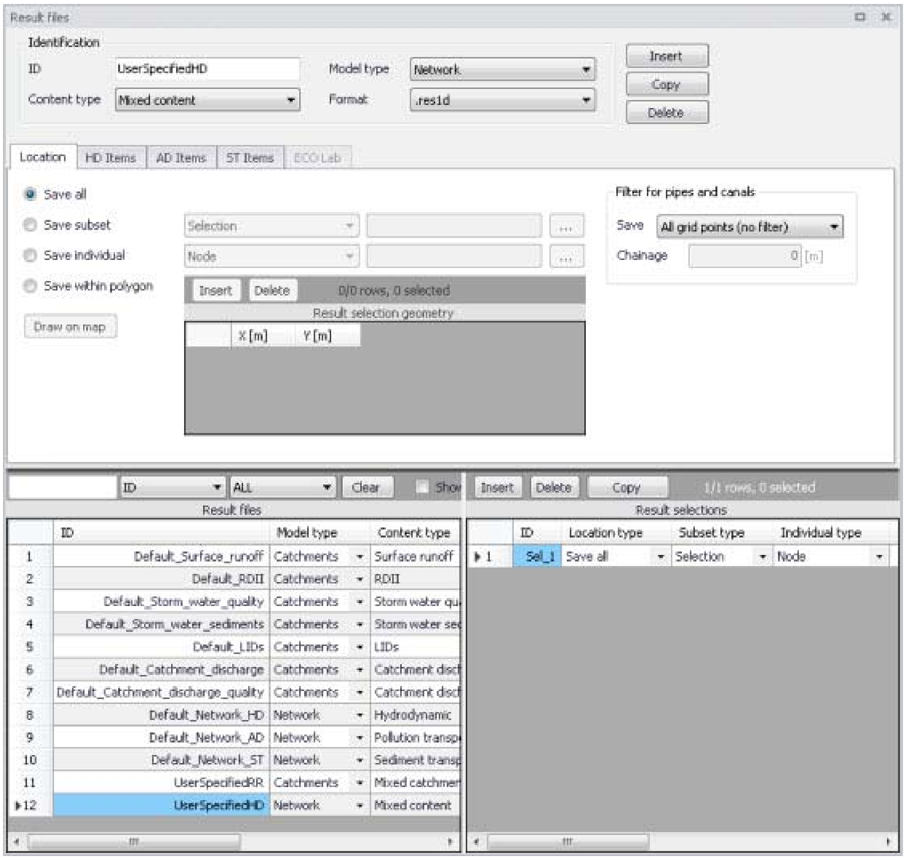


Figure 9.1 The Result Files Editor

The various data tabs and components of the Result Files editor are described in succeeding sections.

9.1.1 Identification

The Identification group box on the Result Files editor contains general information on a result file item.

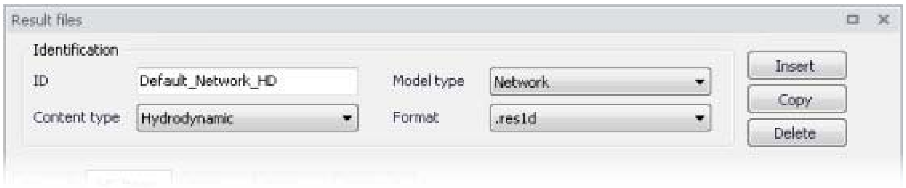


Figure 9.2 The Identification group box on the Result Files Editor

Model Type

Each result file item is categorised based on the type of model from which it is generated. The model may either be a Catchment model, a Network model (comprising CS network and/or River network) or a 2D Overland model.

Content Type

This parameter characterises result files according to the calculation modules, and filters the available result items that may be included in the result file setup. The available categories depend on the selected Model Type for a result file setup.

Content Type can be:

- For 'Catchments' Model Type:
  - Mixed catchment contents. Content type for which result items across various active catchment model-related computational modules may be included.
  - Surface runoff
  - RDII
  - Storm water quality
  - Storm water sediments
  - LIDs
  - Catchment discharge
  - Catchment discharge quality
- For 'Network' Model Type:
  - Mixed content. Content type for which result items across various active network model-related computational modules may be included (e.g. HD, AD, ST, and MIKE ECO Lab).
  - Hydrodynamic
  - Real time control. If the 'Real Time Control (RTC)' module is active.
  - Pollution transport. If the 'Water Quality (AD)' module is active.
  - MIKE ECO Lab. If the 'Water Quality (MIKE ECO Lab)' module is active.
  - LTS extreme statistics. If the 'Long Term Statistics (LTS)' module is active.
  - LTS chronological statistics. If the 'Long Term Statistics (LTS)' module is active.
- For '2D Overland' Model Type:

- 2D area: a map result file containing instantaneous results at regular time intervals
- 2D flood statistics: a map result file containing a single time step with statistical results (e.g. maximum values over time)
- Time series: time series results from one or more cells from the 2D domain
- Volume balance: a time series providing total volumes over the 2D domain
- Section discharge: a time series providing the total discharge computed through a cross section
- 2D area, AD: a map result file containing instantaneous water quality results at regular time intervals
- Time series, AD: time series of water quality results from one or more cells from the 2D domain

Format

Some result files may be saved in various file formats.

An overview of the Identification group attributes in the Result Files editor is shown in Table 9.2 below.

Table 9.2 Overview of the Identification group box attributes (Table msm_RS)

Edit field	Description	Used or required by simulations	Field name in datastructure
ID	Unique identifier for the result file setup	Yes	MUID
Model Type	Categorises the model used to generate the result file as either: - Catchment model, or - Network model	Yes	ModelTypeNo
Content Type	The type of result set under which the result file item falls under. Lists of possible content types are shown below.	Yes	ContentTypeNo
Format	The file format for the generated result file is either: - .RES1D -.DFS0 - DFSU - DFS2	Yes	FormatNo

The buttons to the right of the Identification group box control the data rows in the left overview table in the Result Files editor.

‘Insert' button

Adds a new result item in the Result Files table.

‘Copy' button

Creates a copy of an active result file item. The ID of the copied item is set the same as the original item’s ID plus the suffix ‘_Copy’.

'Delete' button

Deletes the current selected rows from the left overview table in the editor.

9.1.2 Location

The flexibility in results-saving in MIKE URBAN+ extends to possibilities for selecting elements or specifying locations for which to save results in the file.

Result locations may be defined for user-specified result files, but not for Default results. Note that Default result files will always save results in all model elements (i.e. ‘Save all’ option).

Result saving locations are specified in the Location tab in the Result Files editor (Figure 9.3).

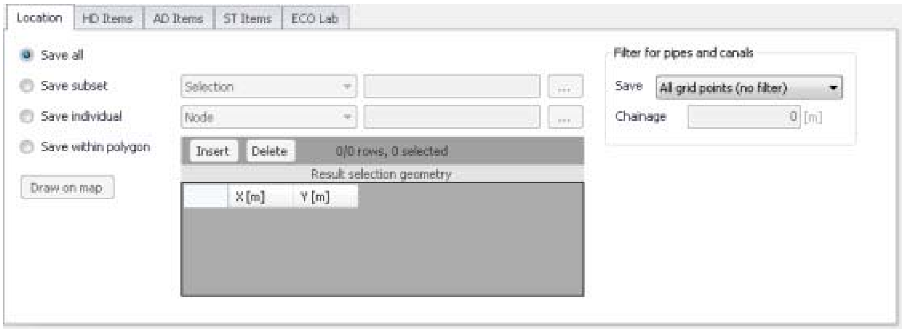


Figure 9.3 The Location Tab in the Result Files Editor

Table 9.3 Overview of the Location tab attributes in the Result Files editor (Table msm_RSS)

Edit field	Description	Used or required by simulations	Field name in datastructure
[Location radio buttons]	Radio button for selection result saving location: - Save all - Save subset - Save individual - Save within polygon	Yes	SelectionNo
[Save Subset dropdown menu]	Dropdown menu for selecting the type of subset	If Location = Save subset	SubsetNo
[Selection list input box]	Input box for a Selection List	If Location = Save subset and Subset = Selection	SelectionListID
[Save Individual dropdown menu]	Dropdown menu for selecting the type of model element for which to save results	If Location = Save individual	IndividualNo
[Element input box]	Input box for an element selection	If Location = Save individual	ElementID
Save [Filter for Pipes and Canals]	Option for selecting the calculation grid point(s) along pipes and canals for which to save results	Yes If Model Type = Network, and results are saved in Pipes and Canals	GridPointNo
Chainage [Filter for Pipes and Canals]	Input box for specifying the chainage (i.e. distance from upstream node) of grid point along the pipe or canal for which to save results	Yes If Model Type = Network, results are saved in Pipes and Canals, and Save = User-specified chainage	Chainage

Save all

This option saves results in all model elements. All Default result files (see Default results section, page 181) use this option, which may not be modified. Also note that this option is not available when results are saved in .DFS0 format (see Format section).

Save subset

This option offers a dropdown list of possible subset groups for which to save results. The list varies according to Model Type associated with the results. A selection list must be defined (Figure 9.4) when the subset is a ‘Selection’.

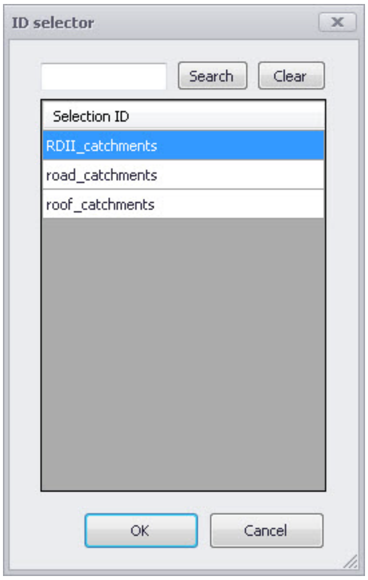


Figure 9.4 Selection lists in the ID Selector window

Save individual

This option offers a dropdown list of model elements for which to save results. The list varies according to the Model Type associated with the results. An element ID must be defined for the selected model element.

Save within polygon

Results from only the network elements or the 2D domain elements within a specified polygon are saved in the file. The polygon is characterised by vertex XY coordinates defined in the secondary grid on the Location tab (Figure 9.5).

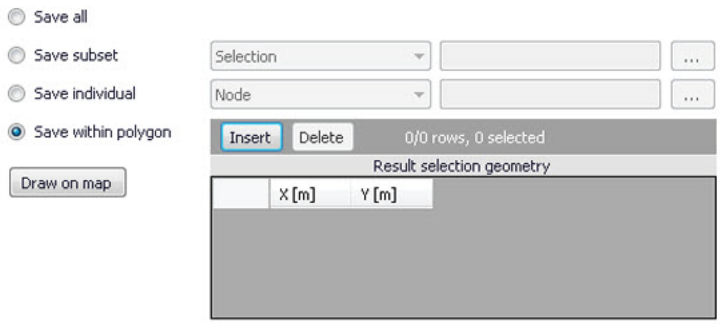


Figure 9.5 The Result Selection Geometry secondary grid in the Location Tab

The location polygon may be defined by:

- Defining values in the secondary table.
  Polygon vertex locations are added and removed from the table using the 'Insert' and 'Delete' buttons at the top of the secondary grid.

- Drawing the polygon on the map using the 'Draw on map' button to the left of the table.
  Use the 'Draw on map' button to define a new polygon on the map. The ‘Draw on map’ button activates the Map view (Figure 12.5). Draw a polygon on the map by defining vertex locations. Double-click on the map to finish the polygon editing. The polygon coordinates are then shown in the secondary table in the Location tab of the Result Files editor.

  The 'Draw on map' button is renamed 'Edit on map' as soon as a polygon has been defined and the secondary table is filled.

  Note that the location polygon is shown on the Map only while drawing. The polygon is no longer shown on the map once the polygon has been drawn.

  When the polygon already exists (i.e. when the Location tab secondary table is not empty), the 'Edit on map' button allows for editing the existing polygon. The Map is shown, where polygon vertices may be moved, deleted, or added.

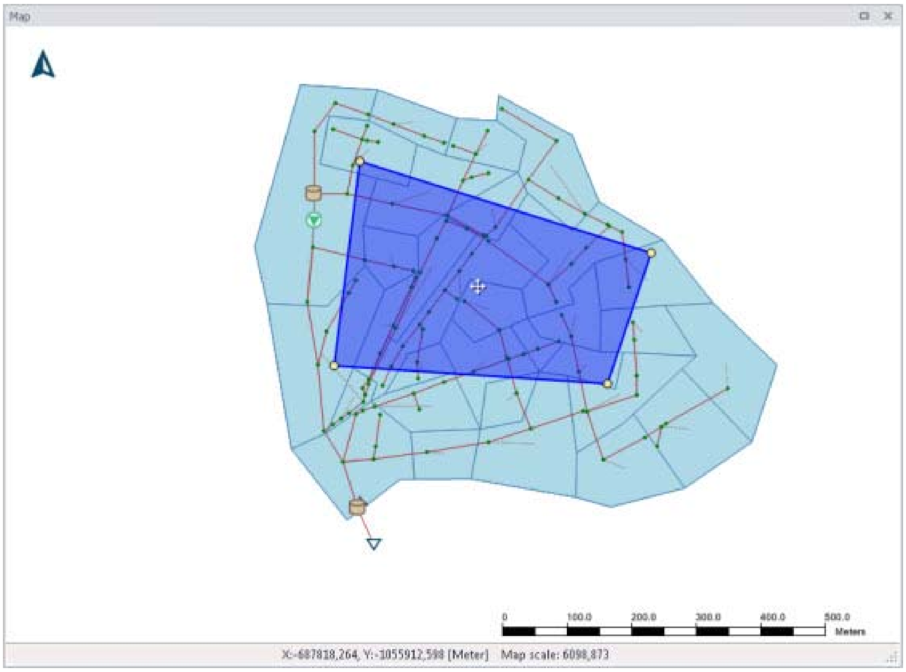


Figure 9.6 Defining a result location polygon on the Map

Coordinates

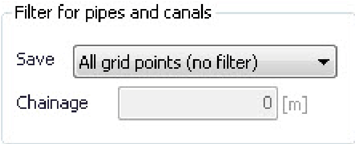
For 2D time series data, the X and Y coordinates of the time series must be specified in the table. When multiple coordinates are specified for the same result file, each location will be saved as an individual item in this file. For each time series, the raw results for the 2D domain element in which the coordinates fall will be saved.

For a 2D section discharge result file, coordinates must be specified for the start and end of the cross section line, through which the discharge will be computed.

Defining saving grid points

The Filter for Pipes and Canals group box is shown on the right side of the Location tab when the result is from a network model (i.e. Model Type = Network).

This section is used to select grid points along pipes and canals for which results are saved in the result file.



The 'Save' dropdown list offers the following options:

- All grid points (no filter)
- Upstream grid point
- Downstream grid point
- Up- and downstream points
- Middle grid point
- User specified chainage. If ‘User specified chainage’ is selected, specify the grid point location for which results are saved in the 'Chainage' field below. Results are saved for the grid point closest to specified chainage value.

9.1.3 Items

Tabs in the Result Files editor are used to select items that will be stored in the result file. Different tabs are shown depending on the Model Type and active project Modules.

Note that customising Items related to LIDs is currently not available. Also, modifying items related to MIKE ECO Lab results is done in the MIKE ECO Lab template (.ECOLAB) and not the MU+ interface.

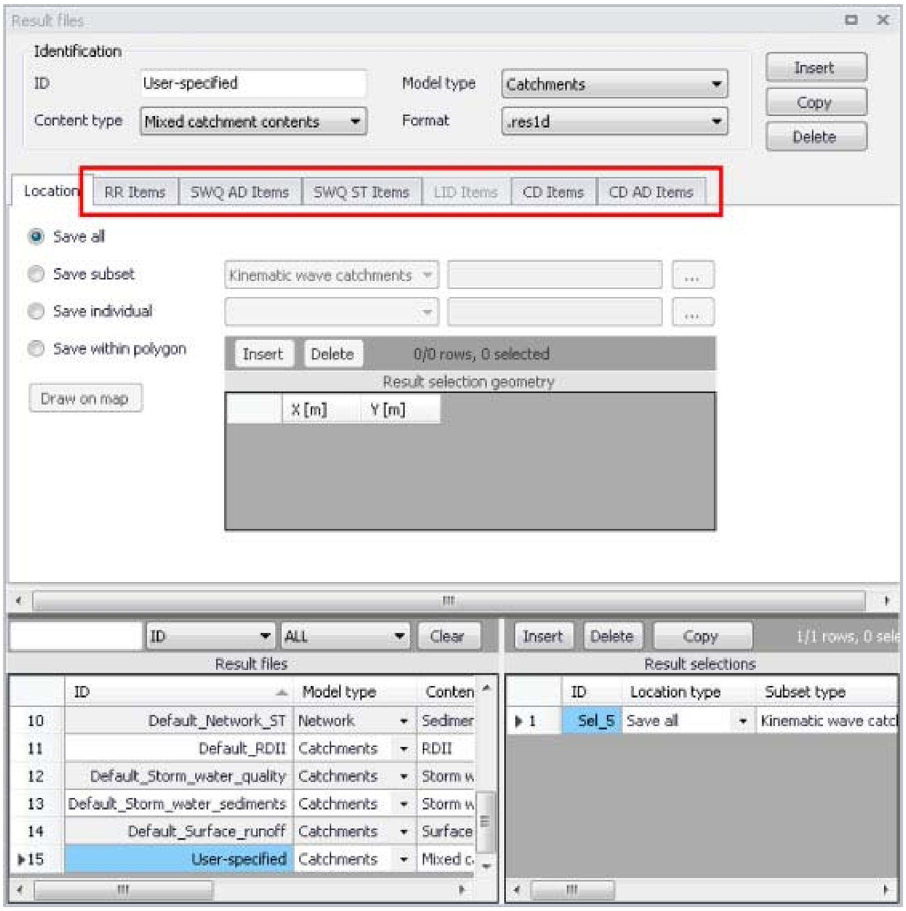


Figure 9.7 Item Tabs in the Result Files Editor. This figure shows available tabs for results from Catchment models.

Each tab shows items related to a computation Module, and the items are categorised as:

- Basic items. Primary result parameters for a simulated process.
- Additional items. Additional result items that provide greater detail on the simulated processes for the system.

Interpolation

For 2D time series results, the spatial interpolation type must be specified.

If ‘Discrete values’ is selected, the values saved to the time series result file are the cell-averaged values.

If ‘Interpolated values’ is selected, the values saved to the result file are determined by second order interpolation. The 2D domain element in which the point is located is determined and the point value is obtained by linear interpolation using the vertex (node) values for the actual element. The vertex values are calculated from the cell-averaged values using the pseudo-Laplacian procedure proposed by Holmes and Connell (1989).

Note that all adjacent elements, including dry elements, are considered in the interpolation calculation.

10 Simulation Specifications

MIKE URBAN+ simulations are started from the Simulation Specifications section. This section includes the following menus:

- **Simulation setup**. Where various combinations of different types of simulations may be setup and run.
- **Batch Simulation**. Controls batch simulations involving the automatic sequential launch of several simulation jobs.

10.1 Simulation setup

The Simulation Setup editor has several tabs, which are shown depending on the active Modules for the project:

- **General.** Includes general parameters, such as definition of the simulation period, selection of simulation types, and free text description of the simulation setup.
- **HD**. Includes parameters specific for HD simulation.
- **AD and WQ**. Includes parameters specific for network AD simulation and MIKE ECO Lab.
- **Results**. Includes specification of results (output) to be generated by the simulation.

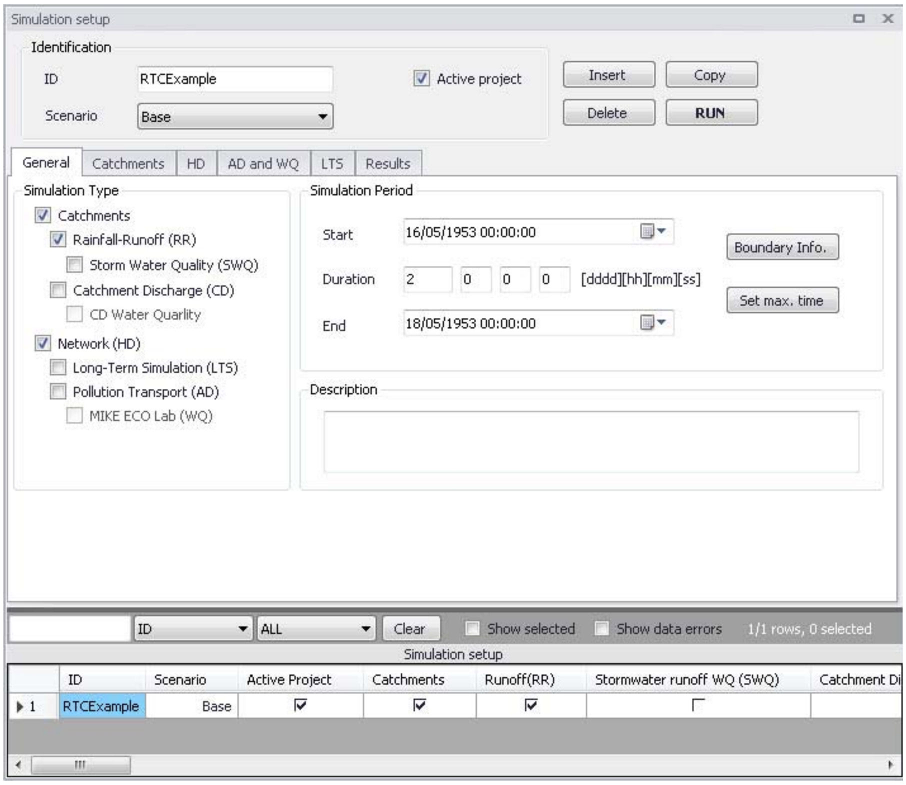


Figure 10.1 The Simulation Setup Editor

The Identification group at the top and the scrollable grid table at the bottom of the editor are common across all tabs.

Table 10.1 Overview of the Simulation Setup Identification data group (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
ID	User-specified ID of simulation. ID will be reflected in the name of result files	Yes	MUID

Table 10.1 Overview of the Simulation Setup Identification data group (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Scenario	Dropdown menu for selecting ID of Scenario for the simulation	Yes	ScenarioName
Active Project checkbox	Defines a simulation setup as "in the tray", i.e. as the one chosen among simulation setups to be used when extracting data by external application, e.g. MIKE Flood, and for running directly from the Simulation toolbar. Only one job at a time may be set as “Active”.	Yes	ActiveProject

The following buttons are also located at the top of the editor with the Identification group:

‘Insert’ button

Inserts a new record in the Simulation Setup editor with a default unique MUID.

‘Copy’ button

Duplicates an existing (currently active) simulation setup record.

‘Delete’ button

Deletes a currently active simulation record.

‘RUN’ button

Triggers export of the currently active simulation job and execution of the simulation.

10.1.1 General

The General tab includes parameters relevant for the entire simulation setup. The following parameters are specified in the General tab:

- Simulation type
- Simulation period

- Description

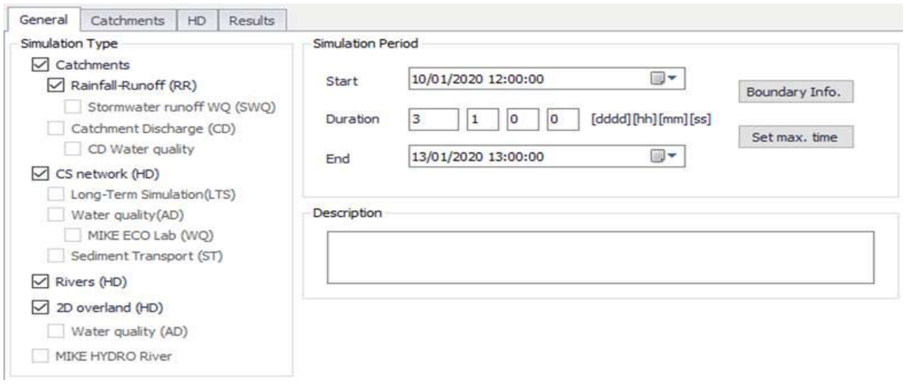


Figure 10.2 The Simulation Setup Editor General tab

An overview of the editor fields and corresponding database attributes is provided in Table 10.2 below.

Table 10.2 Overview of the Simulation Setup General tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Catchments	Activates catchment-related simulation types	Yes	CatchmentComputationNo
Rainfall-Runoff (RR)	Activates runoff simulation	Yes	RRComputationNo
Storm Water Quality (SWQ)	Activates SWQ simulation	Yes	SWQComputationNo
Catchment Discharge (CD)	Activates catchment discharge computations	Yes	CDComputationNo
CD Water Quality	Activates water quality computations for catchment discharges	Yes	CDWQComputationNo
CS network (HD)	Activates CS network HD simulation	Yes	HDComputationNo
Long Term Simulation (LTS)	Activates LTS simulation	Yes	LTSComputationNo
Water quality (AD)	Activates Network AD simulation	Yes	ADComputationNo

Table 10.2 Overview of the Simulation Setup General tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
MIKE ECO Lab (WQ)	Activates Network ECO Lab simulation	Yes	ELComputationNo
Sediment Transport (ST)	Activates CS network sediment transport simulation	Yes	STComputationNo
Rivers (HD)	Activates River network HD simulation	Yes	RiverHDComputationNo
2D overland (HD)	Activates 2D overland HD simulation	Yes	M2DComputationNo
Water quality (AD)	Activates 2D overland AD simulation	Yes	M2DADComputationNo
MIKE HYDRO River	Activates coupling to MIKE HYDRO River. HD coupling is always activated. AD coupling is activated if AD module is activated for another simulation module.	Yes	MHRiverCompulationNo
Start	Specifies start date and time for the simulation.	Yes	ComputationBegin
Duration	Displays the duration of the simulation in days, hours. minutes and seconds. Automatically adjusted based on Start and End time/date. May be edited, adjusting End date/time accordingly.	Yes	-

Table 10.2 Overview of the Simulation Setup General tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
End	Specifies end date and time for the simulation. Adjusted automatically according to user's specification of duration.	Yes	ComputationEnd
Description	Free text description of the simulation setup	Optional	Description

‘Boundary Info.’ button

The Boundary Info. button opens the Boundary Overview window with a horizontal bar chart showing time extent of all active boundary conditions from all included modules.

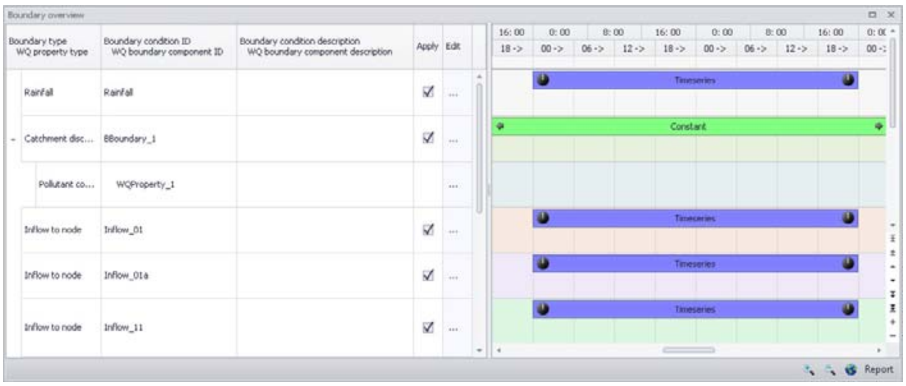


Figure 10.3 The Boundary Overview appears when pressing the ‘Boundary Info.’ button

‘Set max. time’ button

The ‘Set max. time’ button sets the maximum simulation time by filling in the start and end times of the simulation. The start time of a simulation is considered the latest start time of all boundaries. Likewise the end time for the simulation is considered the earliest end time of all boundaries.

Each boundary contains a number of items which can cover different parts of the simulation.

If a limited validity interval is specified for a boundary condition, this specifies the start and end time. If a validity is not specified, only items specified as timeseries have a start and end time. If either a constant or cyclic value is given without validity interval, the item is not included in the evaluation.

10.1.2 HD

For a network (CS network and/or river network) simulation, the tab holds parameters specific to the hydrodynamic simulation setup:

- Fixed simulation time step, or
- Adaptive simulation time step settings
- Network initial condition type
- Additional parameters

For a simulation including 2D overland, the time step parameters are changed to:

- A fixed simulation time step, used by the network simulation. This time step is also used to determine the saving frequency of 2D overland result files, and to synchronize the HD and AD modules for the 2D overland simulation.
- Adaptive time step settings, applying only to the 2D overland simulation.

The tab is active if a hydrodynamic (HD) module is activated and if relevant data exist in the project (e.g. if at least 1 conduit is specified).

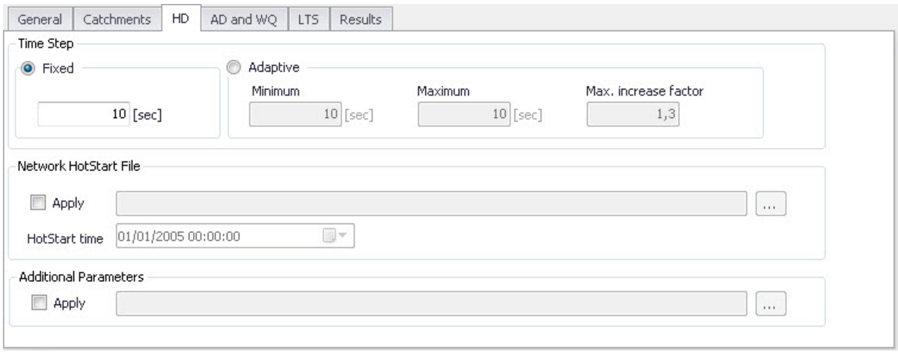


Figure 10.4 HD Tab of the Simulation Setup Editor

Table 10.3 The Simulation Setup HD tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Fixed/Adaptive radio buttons	Toggles between alternative time step type	Yes	HDTimeStepType
Fixed	Specifies a fixed time step for the network simulation	Yes if Fixed time step type or if including 2D overland	HDDtFixed
Minimum	Specifies a minimum time step for network or 2D overland model	Yes if Adaptive time step type or if including 2D overland	HDDtMin
Maximum	Specifies a maximum time step for network or 2D overland model	Yes if Adaptive time step type or if including 2D overland	HDDtMax
Max. CFL number	Specifies the expected maximum CFL number in the simulation, to control the adaptive 2D overland time step	Yes if including 2D overland	M2DHDMaxCFL
Max. Increase Factor	Specifies maximum increase factor for adaptive time step for network model	Yes if Adaptive time step type	HDDtIncreaseFactor
Network initial conditions type	Specifies if the network (CS and/or river) is initially empty, or using user-defined initial conditions	Yes if including CS network or River network simulation	HDInitCondTypeNo
Initial conditions ID	Specifies the ID of the set of initial conditions, defined in the 'Initial conditions' page	Yes if user-specified type	HDInitCondID
Additional Parameters Apply checkbox	Activates *.ADP file with network-relevant input. Define *.ADP file name and path if activated.	Optional	ADPNetworkFileNo

The additional parameter file (*.ADP file) is a separate file with additional settings for the simulation. Please refer to the seperate documentation on this file for further information.

10.1.3 AD and WQ

The "AD and WQ" tab includes parameters specific for the AD and MIKE ECO Lab (WQ) simulation setup. The tab is available if the Water Quality (AD) module is activated in the project, and if at least 1 AD component is specified; otherwise it is hidden.

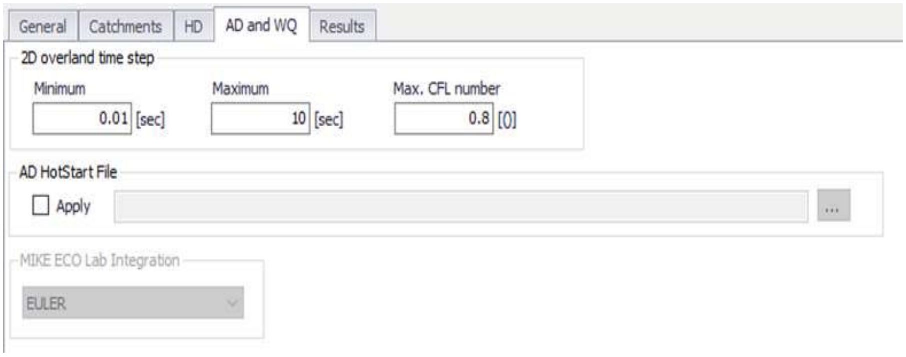


Figure 10.5 The Simulation Setup AD and WQ Tab

Table 10.4 The Simulation Setup AD and WQ tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Minimum	Specifies a minimum time step for 2D overland model	Yes if including 2D overland	M2DADDtMin
Maximum	Specifies a maximum time step for 2D overland model	Yes if including 2D overland	M2DADDtMax
Max. CFL number	Specifies the expected maximum CFL number in the AD simulation, to control the adaptive 2D overland AD time step	Yes if including 2D overland	M2DADMaxCFL
AD Hotstart File Apply checkbox	Checkbox activates hotstart for network AD computations	Optional	ADHotStartFileNo

Table 10.4 The Simulation Setup AD and WQ tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
[Parameter beside the AD Hotstart File Apply checkbox]	The Network AD hotstart file name and path	Yes if AD Hotstart File = Apply	ADHotStatrFile-Name
MIKE ECO Lab Integration	Specified ECOLab integration method	Yes If Simulation Type = MIKE ECO Lab (WQ)	ELIntegrationNo

Note that the hotstart date/time for the AD hotstart is the same as the Network HD hotstart date/time (i.e. HD hotstart must be active if AD hotstart is used).

10.1.4 Results

The Results tab includes parameters for defining output from a simulation setup.

Multiple result files may be specified for each simulation setup.

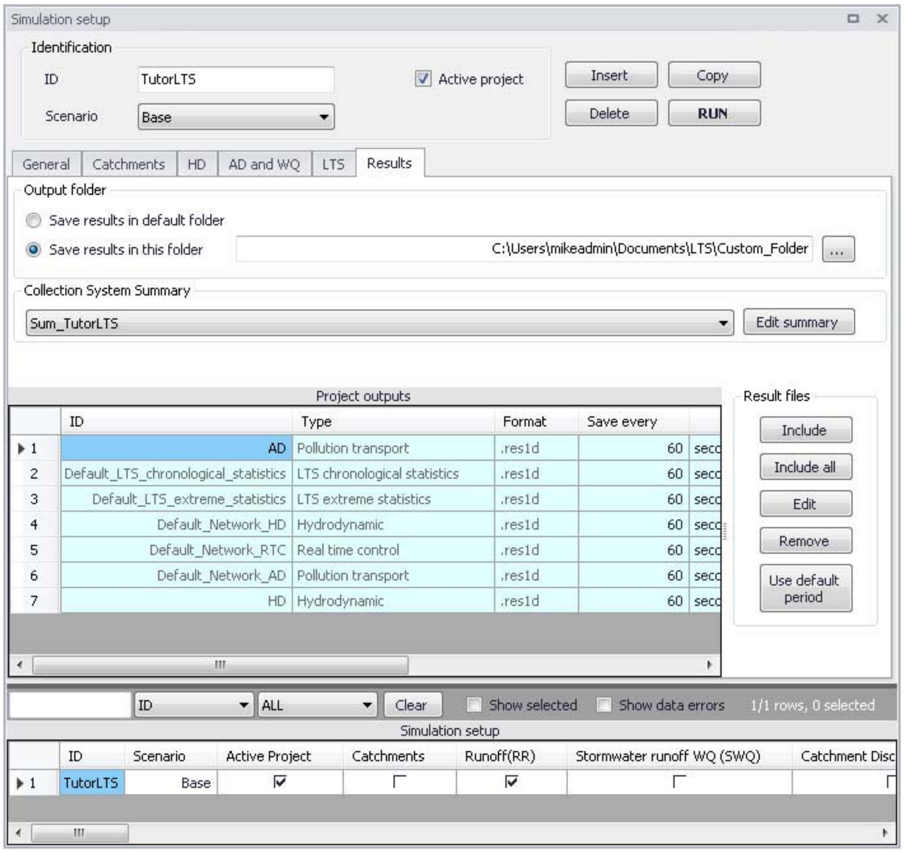


Figure 10.6 Results Tab of the Simulation Setup Editor showing multiple result items

Table 10.5 Overview of Simulation Setup Results tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Save Results in Default Folder/Save Results in this Folder [Output Folder radio buttons]	Toggles between Default folder and user-specified folder for output file location	Yes	HDOutputNo
[Input box beside ‘Save Results in this Folder ‘option]	Contains the path for user-specified output destination folder	Yes if ‘Save Results in this Folder’ activated	HDFolderPath

Table 10.5 Overview of Simulation Setup Results tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Collection System Summary dropdown menu	Specifies a MIKE1D simulation summary. User selects from the list of available summaries. Only one network summary per simulation job is possible.	Yes if including a network simulation	SummaryID
Edit Summary button	Opens the Network Summary editor with the current summary in focus. Allows for editing summary contents. If user has specified a non-existing summary ID, program automatically creates a new summary record with default contents and opens the summary editor with the new summary in focus.	Yes	-

A secondary grid in the Simulation Setup Results tab displays a list of the output files selected for the simulation setup (Figure 10.7). The grid retrieves the information from the Result Files editor (Result Specifications| Result Files).

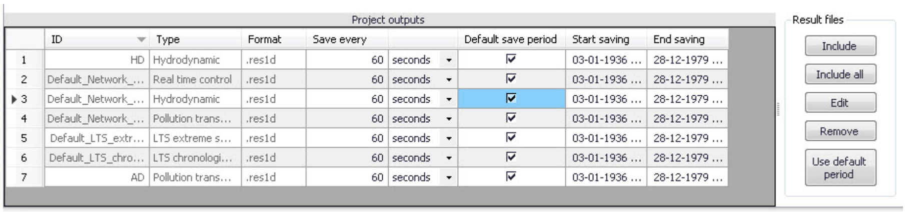


Figure 10.7 The Project Outputs secondary grid in the Results Tab

The list of outputs is controlled by the user using functional buttons to the right of the grid.

The user selects output definitions to include from among those available in the project associated with the modules included in the simulation (and specified in the Result Files editor). The options are filtered according to the contents of the actual simulation.

The list will include at least one "Default" result file definition for each module (Runoff, Network HD, etc.) containing the most usual results for the entire model domain (e.g. runoff for all catchments, discharges and water levels for all model elements, etc.). The list of result file definitions may be extended by "user-specified" output definitions.

An overview of the attributes of the Project Outputs secondary grid is shown in Table 10.6 below.

Table 10.6 Overview of the Project Outputs secondary grid attributes (Table msm_ProjectOutput)

Edit field	Description	Used or required by simulations	Field name in datastructure
ID	MUID of the selected output file definition	Yes	OutputID
Type	Shows type and default contents of the output file (read only)	Yes	ContentsTypeNo
Format	Shows the file format (read only)	Yes	FormatNo
Save Every	Specifies results saving frequency	Yes	DtSave
[Column to the right of 'Save Every' column]	Specified unit for result saving frequency	Yes	DtSaveUnitNo
Default Save Period checkbox	Specify to save results for the entire simulation (check), or for user-specified period only (uncheck)	Yes	DefaultSavePeriodNo
Start Saving	Defines start date and time for saving results	Yes if user-specified save period	SaveStartDate
End Saving	Defines end date and time for saving results	if user-specified save period	SaveEndDate

The following functional buttons are available for controlling the list of outputs in the Project Outputs secondary grid:

'Include' button

Opens a list of all relevant output definitions for the simulation and allows the user to choose those which are to be added to the list.

'Include all' button

Fills the list with all relevant pre-defined output definitions found in the database. Relevance is determined by the modules included in the simulation. If the list is not empty, it only adds those outputs which are not already in the list.

'Edit' button

Opens the Result Files editor with the current output in focus.

'Remove' button

Removes selected output(s) from the list.

'Use Default Period' button

This function finds the full period available for an output item. The information is reflected in the Start Saving and End Saving secondary grid attributes. This tool may be used in defining user-specified saving periods to ensure they fall within valid periods.

10.2 Batch Simulation

If you need to run more simulations sequentially, you can choose to do so by including these to a batch simulation. This is done through the Batch Simulation editor.

The Batch Simulation editor includes functionalities allowing control and execution of batch simulations.

The 'Batch Run' button executes all simulations that have the 'Include to batch' flag set in the sequence that they are specified in the grid table. This means that multiple simulations and scenarios can be simulated in batch without user interaction.

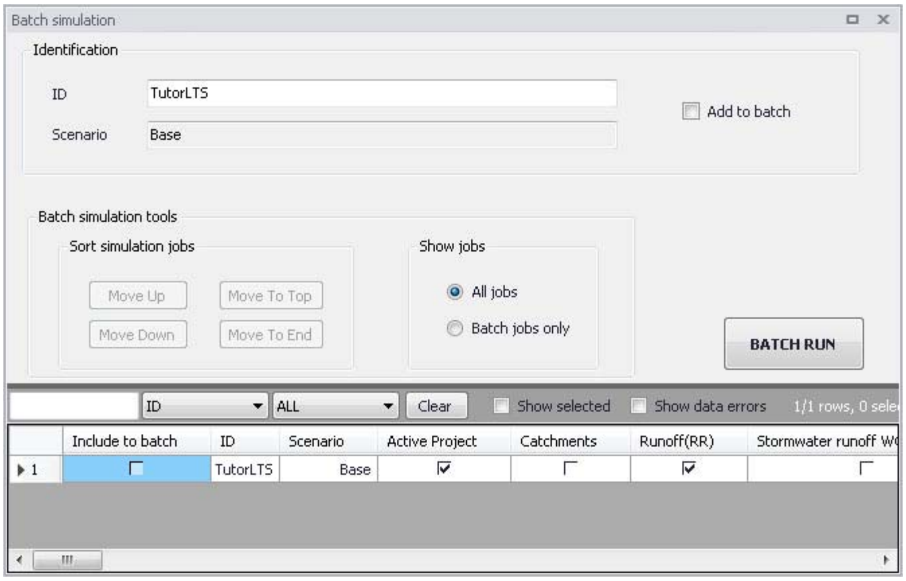


Figure 10.8 The Batch Simulation Editor

The Batch Simulation editor manages the same data from the Simulation Setup editor. The grid table shows the same entries as the grid in the Simulation Setup editor, but built-in tools allow reordering and filtering of simulation job records for batch execution.

Table 10.7 Overview of Batch Simulation editor fields (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
ID	ID of the simulation setup	Yes	MUID
Scenario	Scenario for the simulation setup	Yes	ScenarioName
Add to Batch checkbox	Option for including a simulation setup to batch	Yes	IncludeToBatchNo

The following functionalities are available on the editor:

Move Up

Moves the active record one position up in the grid.

Move Down

Moves the active record one position down in the grid.

Move To Top

Moves the active record to the top of the table.

Move To End

Moves the active record to the bottom of the table.

‘All jobs’ and ‘Batch jobs only’ radio buttons

This filters the list of simulation jobs displayed in the table. A complete list of simulation jobs (i.e. All jobs) is shown by Default, but the display can be reduced to show only those jobs included in the batch (i.e. Batch jobs only).

‘Batch Run’ button

This starts a batch job execution following the sequence of the simulation jobs on the list. Each consecutive job must wait until the previous job has been fully completed. All user prompts are suppressed during the batch job execution, i.e. the simulations are automatically executed without user prompts.

The expert in **WATER ENVIRONMENTS**