MIKE+

User Guide

2D Overland

PLEASE NOTE

COPYRIGHT

This document refers to proprietary computer software which is protected by copyright. All rights are reserved. Copying or other reproduction of this manual or the related programs is prohibited without prior written consent of DHI A/S (hereinafter referred to as "DHI"). For details please refer to your 'DHI Software Licence Agreement'.

LIMITED LIABILITY

The liability of DHI is limited as specified in your DHI Software Licence Agreement:

In no event shall DHI or its representatives (agents and suppliers) be liable for any damages whatsoever including, without limitation, special, indirect, incidental or consequential damages or damages for loss of business profits or savings, business interruption, loss of business information or other pecuniary loss arising in connection with the Agreement, e.g. out of Licensee's use of or the inability to use the Software, even if DHI has been advised of the possibility of such damages.

This limitation shall apply to claims of personal injury to the extent permitted by law. Some jurisdictions do not allow the exclusion or limitation of liability for consequential, special, indirect, incidental damages and, accordingly, some portions of these limitations may not apply.

Notwithstanding the above, DHI's total liability (whether in contract, tort, including negligence, or otherwise) under or in connection with the Agreement shall in aggregate during the term not exceed the lesser of EUR 10.000 or the fees paid by Licensee under the Agreement during the 12 months' period previous to the event giving rise to a claim.

Licensee acknowledge that the liability limitations and exclusions set out in the Agreement reflect the allocation of risk negotiated and agreed by the parties and that DHI would not enter into the Agreement without these limitations and exclusions on its liability. These limitations and exclusions will apply notwithstanding any failure of essential purpose of any limited remedy.

CONTENTS

1 General Settings

MIKE+ 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 MIKE+ 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 options are made available in MIKE+ under the 'Rivers, collection system and overland flows' and 'SWMM5 collection system and overland flows' model types.

Access the Model type and Description editors for 2D overland modelling under the General Settings section.

1.1 Model Type

The 'Model type' editor provides an 'at a glance' view of which MIKE+ features and modules are active. Active features and modules control the menus being accessible from the Setup tree view.

Rivers, Collection System and Overland Flows

Within the 'Rivers, collection system and overland flows' working mode, you can choose to have the following features activated:

- Catchments
- Collection system network
- River network
- 2D overland

Catchments can be connected to both Collection system and River networks. Collection system and River networks can also be connected together. When 2D overland is enabled with Collection system network and/or river network, couplings between the various features are enabled automatically.

You can then activate the following modules, which will apply to all selected features whenever relevant:

- Rainfall Runoff (RR): The RR module simulates hydrological processes on catchments. Enabling the Catchments feature but without including the RR module allows applying catchment discharge to the collection system and river networks.

- Hydrodynamic (HD): The HD module simulates the flows in collection system networks, river networks and/or 2D overland, depending on the selected features. The HD module includes the two sub-modules Control Rules (only apply applicable to Collection system and River networks) and Long Term Statistics (LTS), only applicable to the Collection system network.

- Transport (AD, SWQ): The transport module simulates the Advection-Dispersion processes (for e.g. pollution transport) in collection system networks, river networks and/or 2D overland, depending on the selected features. The transport option also enables stormwater quality (SWQ) simulations on catchments, when Catchment feature and Rainfall-Runoff module are active.

- Water quality (MIKE ECO Lab): The MIKE ECO Lab module simulates the advanced water quality processes (e.g. biological processes) in collection system networks, river networks and/or 2D overland, depending on the selected features.

- Sediment transport (ST): The ST module simulates transport of sediments in collection system and river networks. It is not available for 2D overland.

- Data assimilation (DA): The DA module includes an updating feature for continuous processing and updating of model states during simulations. Data assimilation is used for many types of applications, but in particular for real-time forecasting where the forecasted values of network state conditions (water levels, flows, etc.) are much more accurate and reliable when update and error estimation are included. It is designed for river networks.

Figure 1.1   Model Type editor for 'Rivers, collection system and overland flows' models

The following couplings to other MIKE products are also available:

- Coupling to MIKE HYDRO River: When this 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 the 'Transport (AD, SWQ)' module is also included in the simulation setup. AD components' names in MIKE HYDRO must, however, be identical to the components' names in MIKE+.

- Coupling to MIKE 21 or MIKE 3 Flow Model FM: When this is active, the desired MIKE 21 FM or MIKE 3 FM model setup file must be selected in the 2D / 3D overland model' page. Note that this coupling cannot be enabled at the same time as the 2D overland feature, since only one source of overland model can be used at a time.

- Coupling to MIKE SHE or FEFLOW: When this is active, it is possible to define parts of the river network to be coupled to a MIKE SHE or FEFLOW model. The network simulation file (*.m1dx file) must then be exported from MIKE+ for use in MIKE SHE or FEFLOW. This also allows coupling the collection system network to MIKE SHE: in that case, the corresponding coupling file (*.couple) must also be exported from MIKE+ for use in MIKE SHE.

When the different features and modules are activated, the corresponding menus will be added in the Setup view. Note that the features and modules are actually included in a simulation only if they are also included in the simulation definition, from the 'Simulation setup' editor.

The 'Pump emergency storage estimation' option enables a special type of analysis for collection system networks, enabling extra simulation options under the 'Special analyses' group in the Setup tree view. It estimates the time and volume available prior to an overflow at the wastewater pump station assets in the event of system failure.

SWMM5 Collection System and Overland Flows

Under the 'SWMM5 collection system and overland flows' working mode, you can choose to have the following features activated:

- Catchment
- Collection system network
- 2D overland

Activation of relevant features allows the following modelling capabilities:

- Catchment: Various hydrological processes that produce runoff from urban areas can be modelled with SWMM.
- Collection System Network: This allows for the hydrodynamic simulation of flows and water levels in the urban storm drainage and wastewater collection networks, thus providing an accurate information about the network functionality under a variety of boundary conditions. The model can be enhanced by the variety of real-time control functions.
- 2D Overland: Allows for the simulation of flows and water levels over 2D overland surfaces, as well as hydrodynamic coupling with a SWMM5 collections system network for integrated flood modelling.
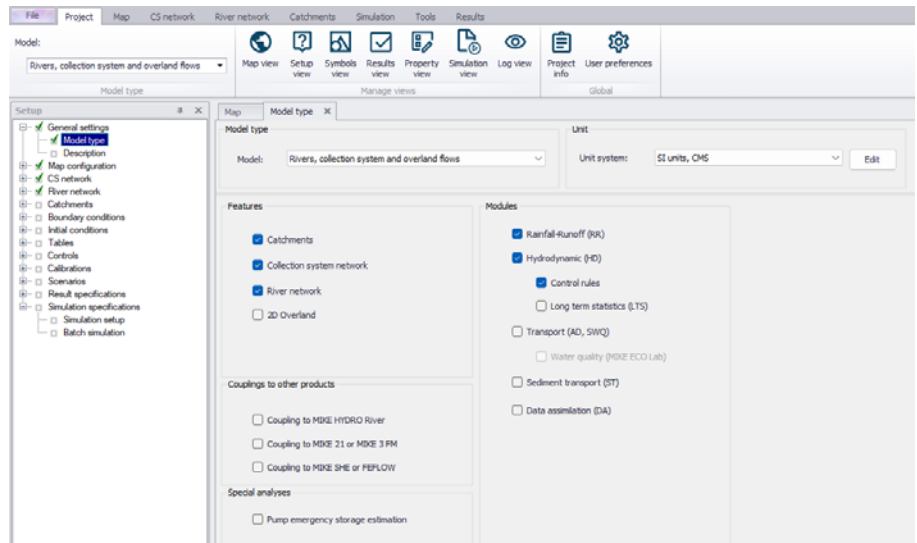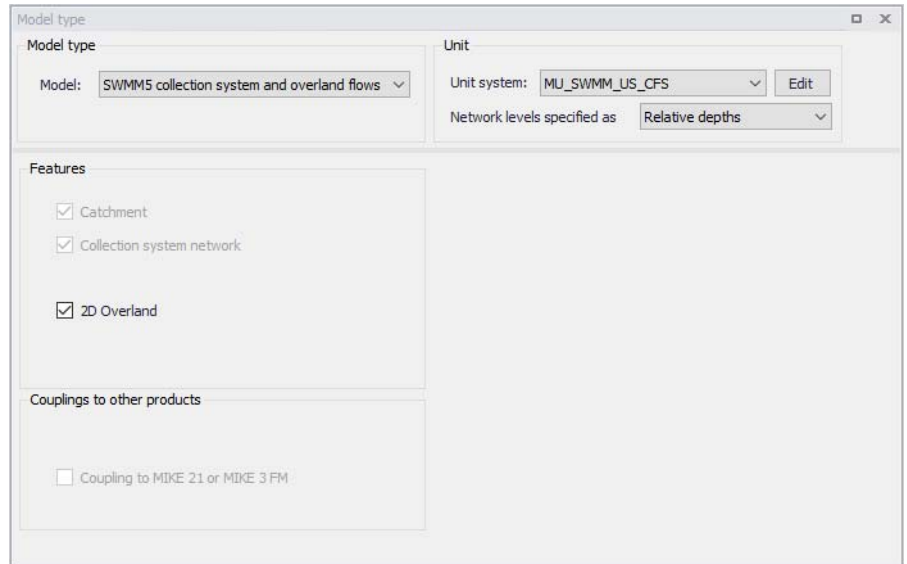
Figure 1.2   Model Type editor for 'SWMM5 collection system and overland flows' models

The following couplings to other MIKE products are also available:

- Coupling to MIKE 21 or MIKE 3 Flow Model FM: When this is active, the desired MIKE 21 FM or MIKE 3 FM model setup file must be selected in the 2D / 3D overland model' page. Note that this coupling cannot be enabled at the same time as the 2D overland feature since only one source of overland model can be used at a time.

The corresponding menus are added in the Setup view when relevant features are active. Note that features and modules are included in a simulation only if they are also activated in the simulation definition from the 'Simulation Setup' editor.

1.2   Description

The General Settings section contains the Description editor. This editor allows for addition of information about the project and a free text description of the model. It may also be used as a model build log to make notes on updates and amendments to the model.

Figure 1.3   The Description editor

2 Map Configuration

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

2.1 Coordinate System

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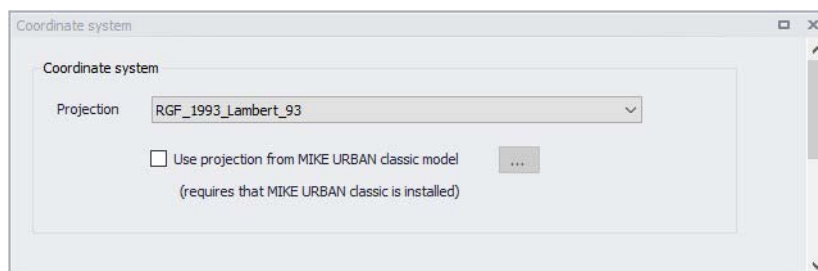
Figure 2.1 The Coordinate System dialog showing information on the projection system used in the project

The Projection can be selected from the short list, or by searching the projection amongst all the map projections available in MIKE+. The latter is achieved by selecting the <Browse…> option at the bMIKE+ottom of the list: this will open a window listing the available projections, and where it is also possible to import new projections from a projection file (*.prj file).

Alternatively, the map projection may be read from a MIKE URBAN classic file. If MIKE URBAN classic is installed on your computer, you can tick the corresponding option, select a MIKE URBAN classic file and the same projection will be used afterwards in MIKE+.

When changing the map projection, it is possible to reproject geographical data in the project, for example to convert the coordinates of the network and catchments data, or mesh arcs used for the creation of the 2D domain. Some data files used as input for the simulation can however not be re-projected: this is especially the case for an external 2D domain file (*.mesh or *.dfs2) or external 2D data file used to map input parameters (e.g. *.dfsu or *.dfs2 file used to map the 2D surface roughness).

The same options for selecting the Projection are also used in the 'New Module Setup' window when a new MIKE+ project is created (Figure 2.2).
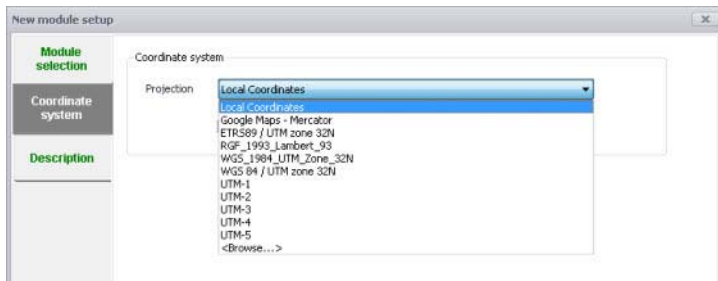
Figure 2.2   Specify the projection system for a new MIKE+ project on the New Module Setup window

2.2   Background Map

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

Activate a background map overlay by ticking the 'Visible' checkbox on the editor.
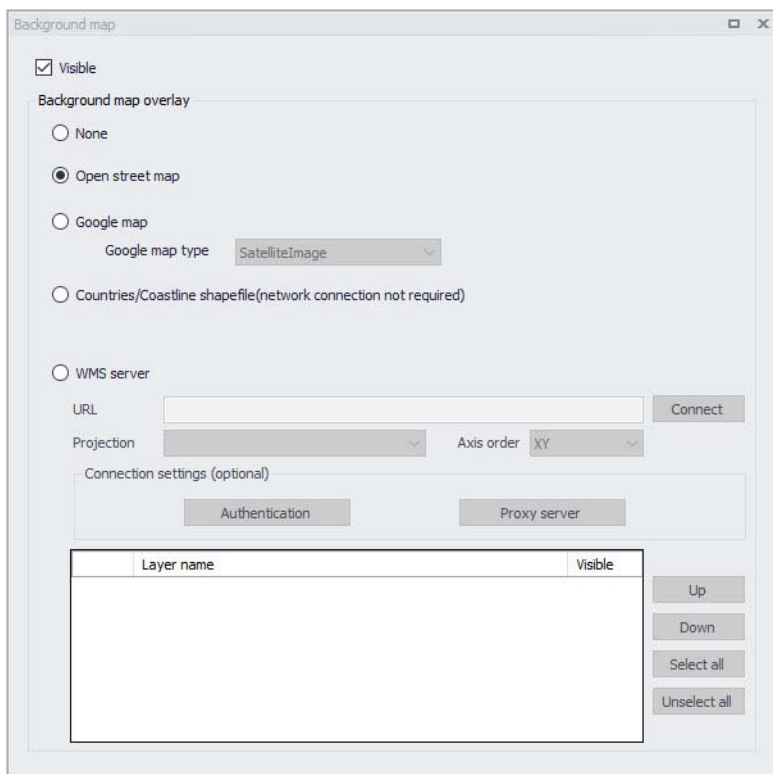
Figure 2.3   The Background Map Editor

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 feature showing coastlines and demarcating oceans and inland areas.

- **WMS server**. Background maps obtained from a remote server. Enter the URL of the server and click 'Connect'. If the server is a private server, you will need to supply the user name and password by using the 'Authentication' button, and you may optionally tick 'Save password' in order not to enter it again the next time you open MIKE+. If your network uses a proxy server, you will need to press the 'Proxy server' button to provide the server address (which should include the port number, if any) as well as optional user name and password for this proxy server. When the connection is established, the table will provide the list of layers available on the WMS server, and it is possible to select which layers to display in MIKE+ using the 'Visible' box. The list of projections will show the map projection(s) supported by the WMS server, and the one used for the model data in MIKE+ will be selected if possible. Note that displaying layers from a WMS server requires that the MIKE+ project uses the same map projection as the WMS layers: if the projection used in MIKE+ doesn't match any of the projections supported by the WMS server, you will be asked to update the map projection in MIKE+. Also note that it is only possible to connect to WMS servers using projected map projections (geographical coordinate systems not supported). An axis order also needs to be specified, defining the format of the coordinates on the WMS server: most of the servers provide coordinates in the XY order, but some servers provide coordinates in the opposite order and in this case the option must be changed to 'YX' otherwise the layers won't be displayed on the map.

An internet connection must be available for Open Street Map, Google Map and WMS server overlays (Figure 2.4).
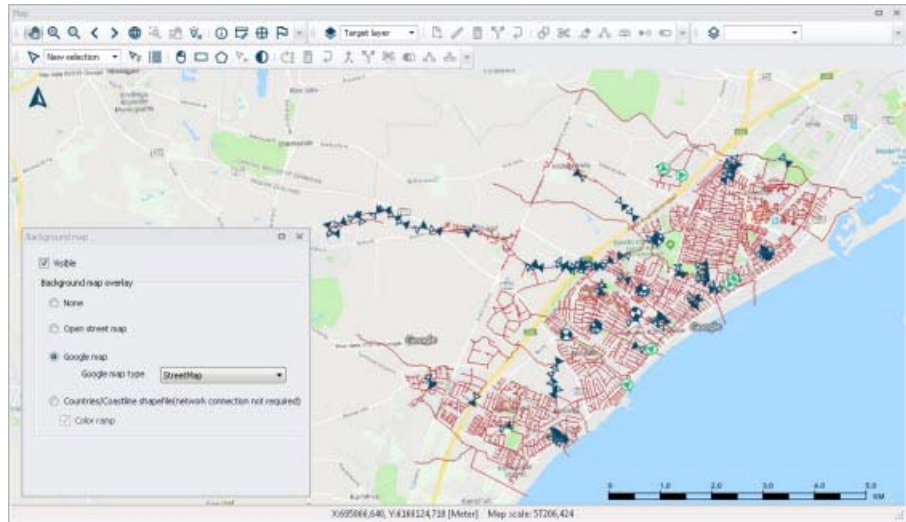
Figure 2.4   An example Google Map background on the Map View in MIKE+

Note: Because the proxy server settings are related to your local network and not to the model database, these settings are stored on the local machine instead of the MIKE+ project files. That means that the specified settings will then apply to other MIKE+ projects opened on the same computer. On the opposite, the proxy settings will have to be supplied again if the same model database is later opened from another computer connected to the same network.

3   2D Overland Hydraulic Modelling

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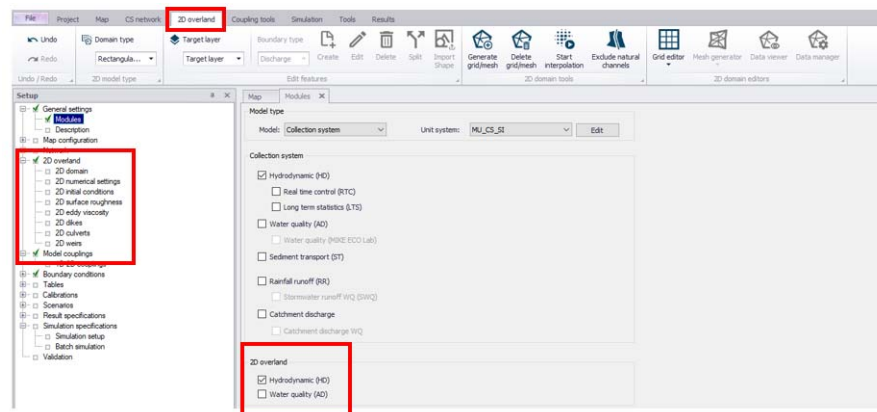
Figure 3.1   Activate the Hydrodynamic (HD) 2D Overland module to access editors for building 2D Overland models in the MIKE+) Setup Tree View.

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 Fluid Properties
- 2D Infrastructures
- 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+, 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.
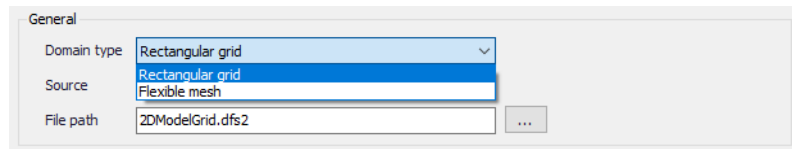
Figure 3.3   2D domain type options

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+. The Source dropdown menu offers options for using a:

- Domain file created from MIKE+ definition: With this option, it is required to specify the extent 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.
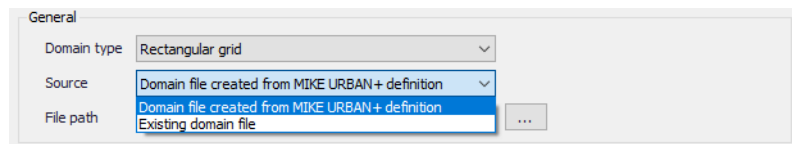
Figure 3.4   2D domain source options

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+: This is relevant for when the Source is 'Domain file created from MIKE+ definition'.

- Path to the existing domain file: This is relevant for when Source is 'Existing domain file'.
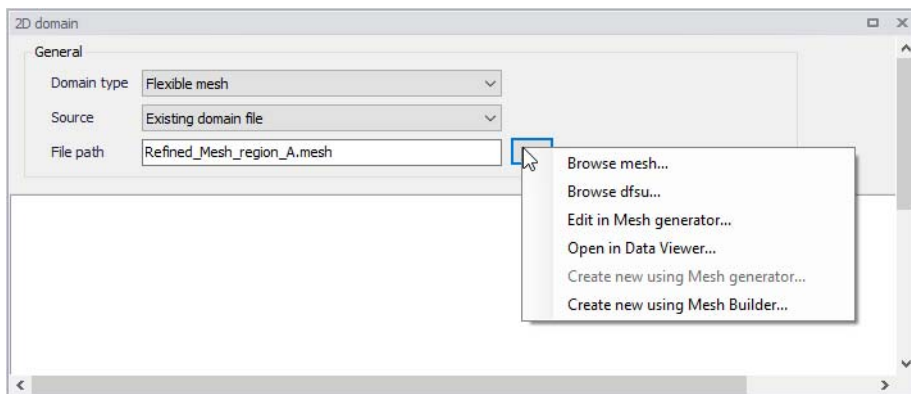
Figure 3.5   File path options

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.

- Create new using Mesh Generator: Launches the Mesh Generator to create a new flexible mesh file.

- Create new using Mesh Builder: Launches the Mesh Builder to create a new flexible mesh file. You must be signed in to MIKE Cloud in order to access the Mesh Builder. Read Chapter 2.13 'Working with MIKE Cloud' in the MIKE+ Model Manager User Guide for more information.

3.1.2 Domain

When creating new computational grids from MIKE+, 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+

- 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 'Topography interpolation' tab to be properly detected as 'Topographical Data'.
  – Collection system network: The extent of the network of pipes and canals.
  – River network: The extent of the whole network of rivers
  – 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+.

  - 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 within the area with indented cells. The adjusted level in this lowest level is assigned to all other cells in 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).

Lowered cells (open boundary)

5.4	5.3	4.9	4.5	3.4	3.5	2.7	2.4	2.4	2.5	2.8	3.3	3.5	3.6	3.6	4.2	4.4	3.9
82.5	82.5	82.5	82.5	1.7	1.7	1.7	1.7	1.7	1.7	1.7	1.7	1.7	1.7	82.5	82.5	82.5	82.5
5.6	5.4	5	4.5	3.5	3.4	2.7	2.2	2.7	3.8	3.9	4	3.8	3.6	3.5	3.8	3.8	3.9
5.6	5.4	5	4.5	1.7	1.7	1.7	1.7	1.7	1.7	1.7	1.7	1.7	1.7	3.5	3.8	3.8	3.9
5.4	5.4	4.9	4.6	3.8	3.2	3	2.5	2.2	3	3.3	3.8	3.8	3.8	3.6	3.9	4.1	4.3
5.4	5.4	4.9	4.6	3.8	3.2	3	2.5	2.2	3	3.3	3.8	3.8	3.8	3.6	3.9	4.1	4.3
5.3	5.3	4.8	4.6	3.7	3.5	3	2.8	2.6	3.1	3.3	3.9	4	3.9	2.8	4	4.2	4.7
5.3	5.3	4.8	4.6	3.7	3.5	3	2.8	2.6	3.1	3.3	3.9	4	3.9	2.8	4	4.2	4.7

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 in the area (i.e. 2.2 m encircled) minus the specified indentation (here 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.
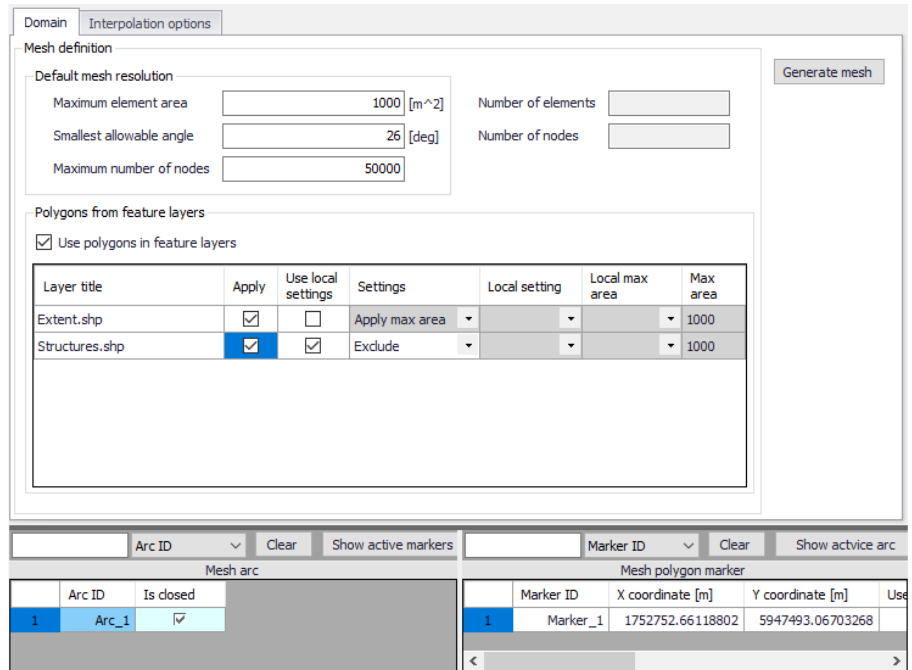
2D Domain

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

The first step to build a flexible mesh is to specify the 'Default mesh resolu-tion', which will apply in locations where no local mesh resolution is defined. Under this 'Default mesh resolution' 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.

The second step is to define Mesh polygons and arcs, to apply local mesh resolutions. Polygons are used to define 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: When the option 'Use polygons in feature layers' is active, polygons from feature layers may be used for the mesh genera-tion. The layers must be added to the Map before use in mesh genera-tion. Usable feature layers are automatically listed. Activate the 'Apply' option for layers to use and adjust the settings below:

– Use local settings: If this option is not selected, the default mesh resolution will apply within the polygon: the effect of the polygons is limited to the fact that mesh elements will be somehow aligned to the polygons. If this option is selected, then local settings controlled by the following parameters will be applied in the polygons.

– Settings. Three settings definitions are available:

• Apply max area: For this setting, all polygons in the feature layer will use the same local mesh resolution, controlled by the value specified for the 'Max area'.

• Exclude: For this setting, all polygons in the feature layer will be excluded from the mesh. Not other input is required.

• Layer attributes: For this setting, each polygon in the feature layer may have its own local mesh resolution, which is controlled by the attributes of the layer. In the 'Local setting' column, it is required to select an attribute from the feature layer, containing the type of setting for each polygon: this must be a text attribute, and for each polygon the text must be either 'local area' or 'exclude' (case insensitive). In the 'Local max area' column, it is required to select an attribute containing the maximum element area for each polygon, to be used for polygons included in the mesh.

Polygons from feature layers

☑ Use polygons in feature layers

Layer title	Apply	Use local settings	Settings	Local setting	Local max area	Max area
Extent.shp	☑	☐	Apply max area ▾	▾	▾	1000
Structures.shp	☑	☑	Exclude ▾	▾	▾	1000

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

• Mesh arcs and polygon markers: Mesh arcs and polygon markers are 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). A polygon without marker will be meshed with the default mesh resolution.

Define arcs and polygons on the Map using the Flooding Layer Editing tool-bar (Figure 3.10) or the Edit Features toolbox on the 2D Overland menu rib-bon (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 pol-ygon. 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). The 'Show active marker' button, in the table of arcs, may be used to find the marker located in a given closed arc: select the relevant arc in the table, press the button, and the number of the corresponding marker in the table will be highlighted in dark blue. Similarly, the 'Show active arc' button, in the table of markers, may be used to find the encapsulating arcs: select the relevant marker in the table, press the button, and the numbers of the corresponding arcs in the table will be highlighted.
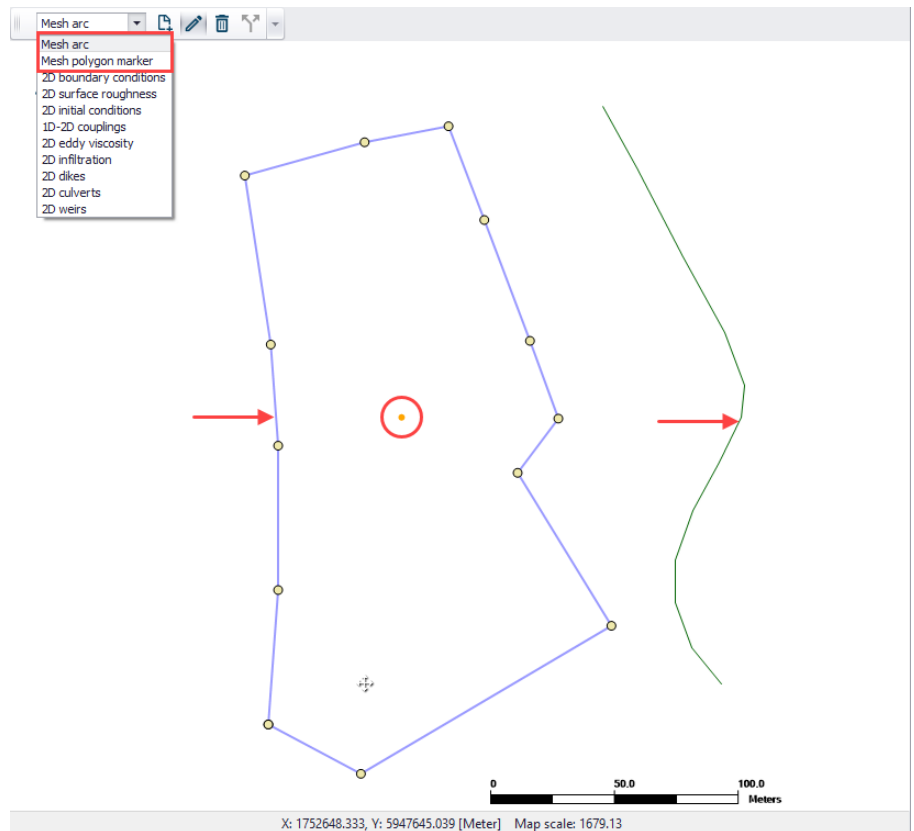
Figure 3.10   Define arcs and polygons on the Map via the Flooding Layer Editing toolbar. The figure shows a closed and open arc indicated by the arrows, and a polygon marker (encircled).
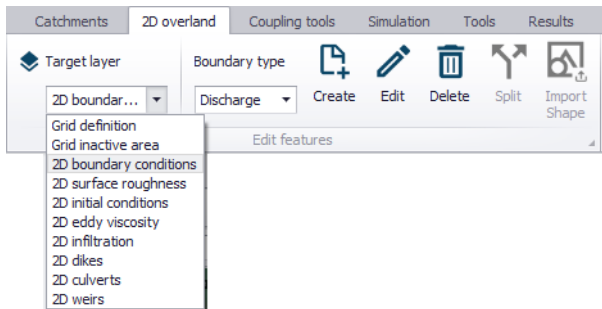
Figure 3.11   The Edit Features toolbox on the 2D Overland menu ribbon

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

It may sometimes happen that a polygon from feature layers and a polygon defined by closed mesh arcs overlap (one being eventually fully included in the other one). In this case, the following rules apply:

- If one polygon is requested to be meshed and the other polygon is requested to be excluded from the mesh: then the priority is given to the polygon excluded from the mesh, and all the area covered by this polygon is excluded (even the parts covered by the other polygon).

- If the two polygons are requested to be meshed: priority is given to the polygon from the feature layer. In this polygon, the mesh will get the resolution requested for this polygon, even in the parts covered by the polygon defined with mesh arcs.

- If the two polygons are requested to be excluded from the mesh: the total area from the two polygons is excluded from the mesh.

Note that building areas can be excluded from the simulation, in order to take the effect of the obstacles on the flow. However, if rainfall is to be applied on the 2D domain, the volume of rain falling on the buildings will not be accounted for. The use of 2D Infrastructures is an alternative solution to describe buildings, which can keep taking into account the rainfall volume falling on buildings.

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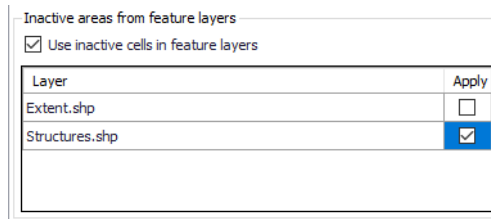
Figure 3.13   List of polygon features that may be used to define inactive cells for rectangular grids

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.
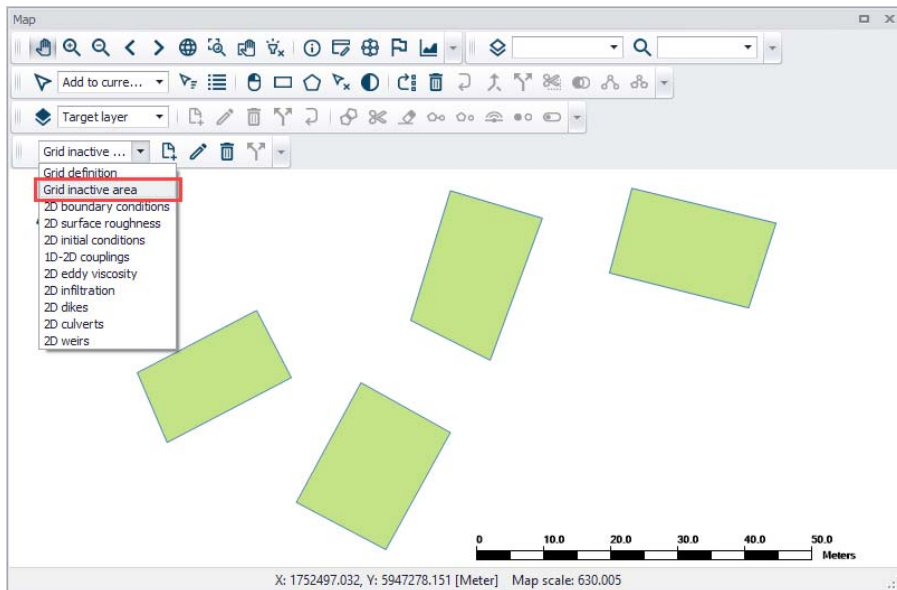
Figure 3.14   Use the Flooding Layer Editing toolbar to create Grid Inactive Area polygons on the map.
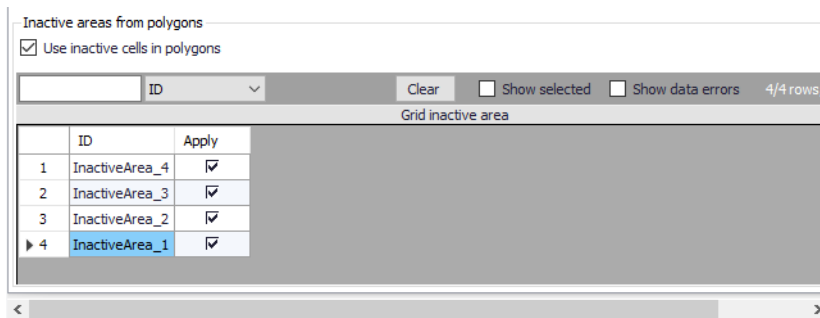
Figure 3.15   Grid inactive area secondary table at the bottom of the Inactive Areas tab page

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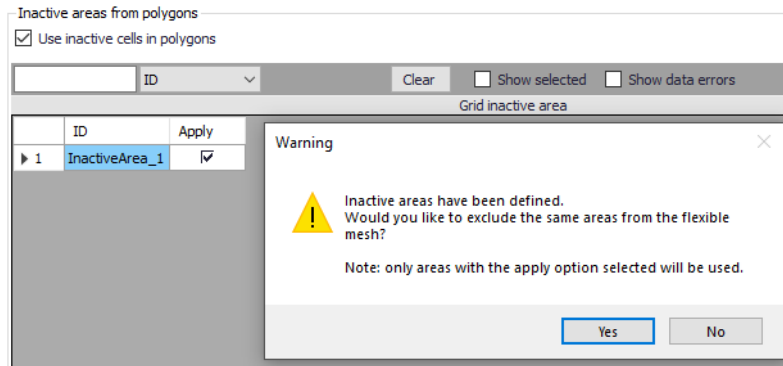
Figure 3.16   Warning message offering to option to use grid inactive areas a mesh domain polygons.

Note that building areas can be excluded from the simulation, in order to take the effect of the obstacles on the flow. However, if rainfall is to be applied on the 2D domain, the volume of rain falling on the buildings will not be accounted for. The use of 2D Infrastructures is an alternative solution to describe buildings, which can keep taking into account the rainfall volume falling on buildings.

3.1.4   Topography interpolation

The 'Topography interpolation' 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.
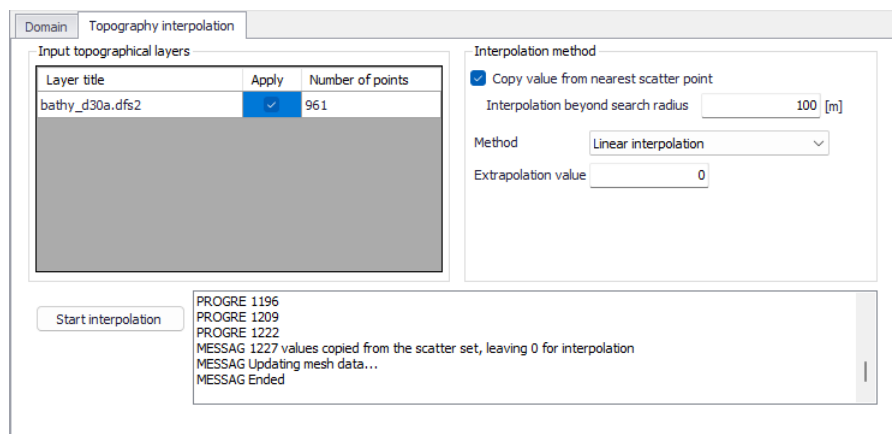
Figure 3.17   The Topography interpolation tab page

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+:

- Raster Layers *.TIF and *.TIFF

- Raster Layer *.DFS2

- Feature Layer *.XYZ

- Mesh Layers *.MESH and *.DFSU

Load these data layers into MIKE+ 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' column 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 *TIFF, *.DFS2 or *.DFSU files, respectively.

Figure 3.18   The Input Topographical Layers table on the Topography interpolation 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.
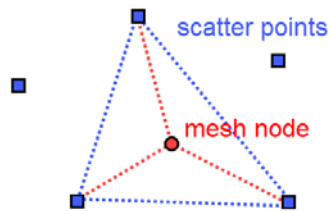
Figure 3.19   Illustration of the linear interpolation method for deriving topography values for mesh nodes from input scatter points

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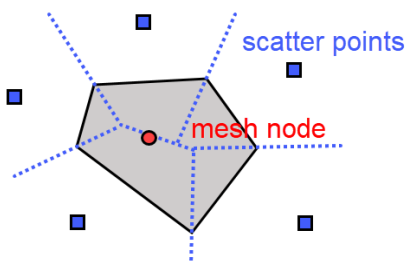
Figure 3.21   Illustration of the natural neighbour interpolation method for deriving
topography values for mesh nodes from input scatter points

- 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.
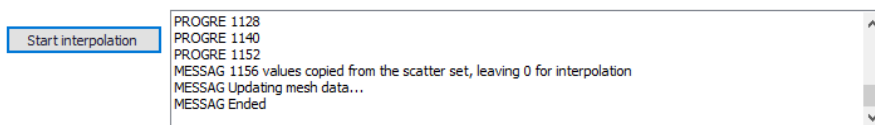
Figure 3.22   The Start Interpolation button and progress log window

3.1.5   Creating a Rectangular Grid Workflow Example

1. Load desired input topographical layers for 2D grid interpolation into the MIKE+ project.

2. On the 2D Domain editor, choose a 'Rectangular grid' domain type and 'Domain file created from MIKE+ 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 'Topography interpolation' 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.
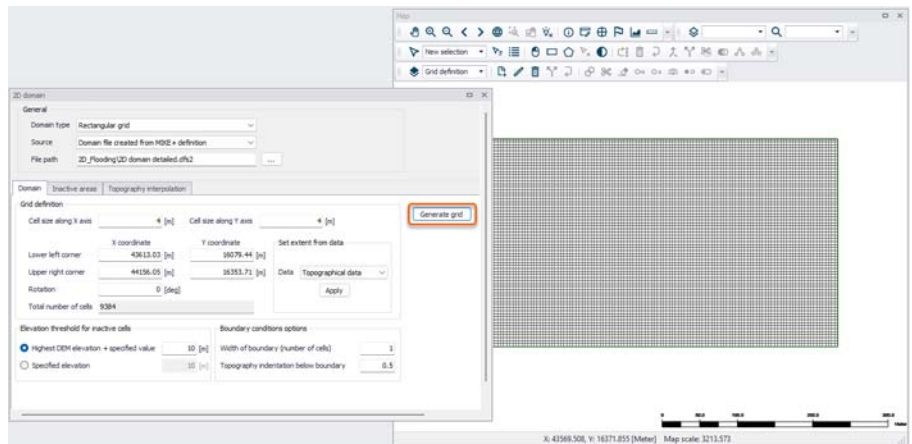
6. Examine the generated grid. Adjust grid sizes and extent, as needed. Re-generate the grid after changes to grid parameters.

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 'Topography interpolation' 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+ project.

2. On the 2D Domain editor, choose a 'Flexible mesh' domain type and 'Domain file created from MIKE+ 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. 20).
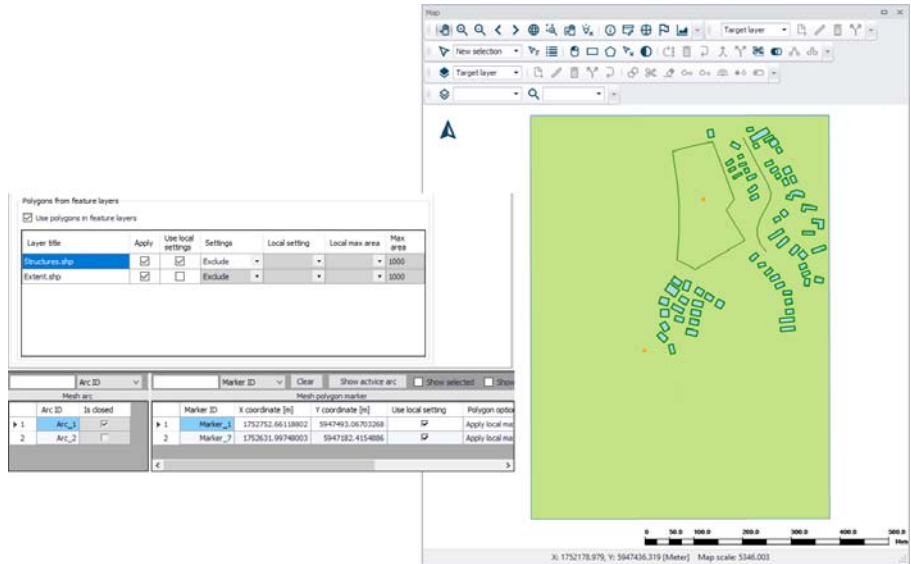
4. Customize other meshing parameters on the Domain tab, such as 'Maximum element area' and 'Smallest allowable angle', if so desired. More details on these options are described in Chapter 3.1.2 - Flexible Mesh *(p. 20)*.

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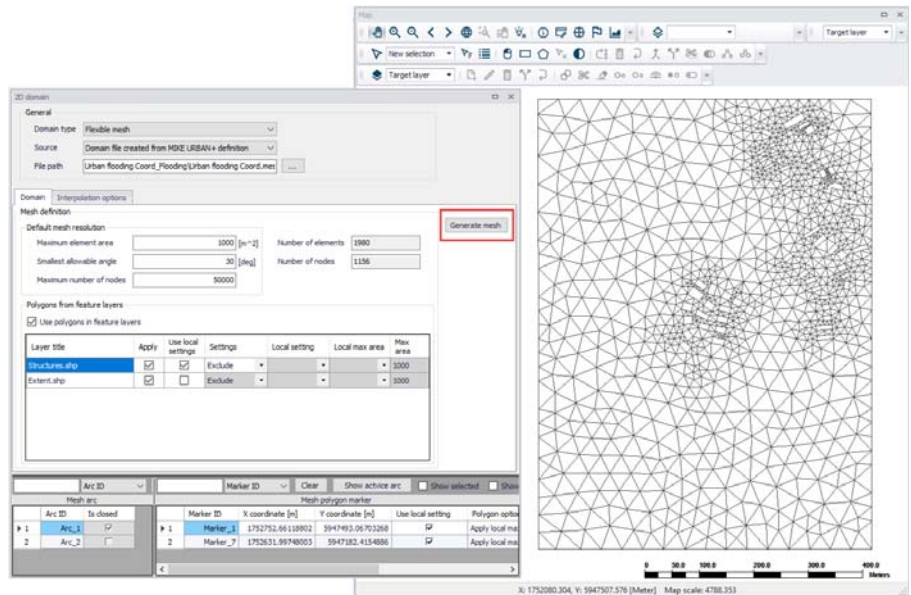
6. Examine the generated computational mesh. Adjust element sizes, the extent, or mesh areas using arcs and polygons, as needed. Re-generate the mesh after changes to meshing boundaries and parameters.

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

Input topographical layers

Layer title	Apply	Number of points
DTM_XYZ	☑	224326
DTM_DFSU	☐	67562
DTM_MESH	☐	67562
DTM_DFS2	☑	933530

8. Choose the preferred interpolation method. More details on the methods are described in Chapter 3.1.4 - Interpolation Method *(p. 28).*

Interpolation method

☑ Copy value from nearest scatter point

Interpolation beyond search radius   100 [m]

Method   Linear interpolation

Extrapolation value   0

9. Click on the 'Start interpolation' button. A progress bar appears, and the status of the process is shown on the log window.
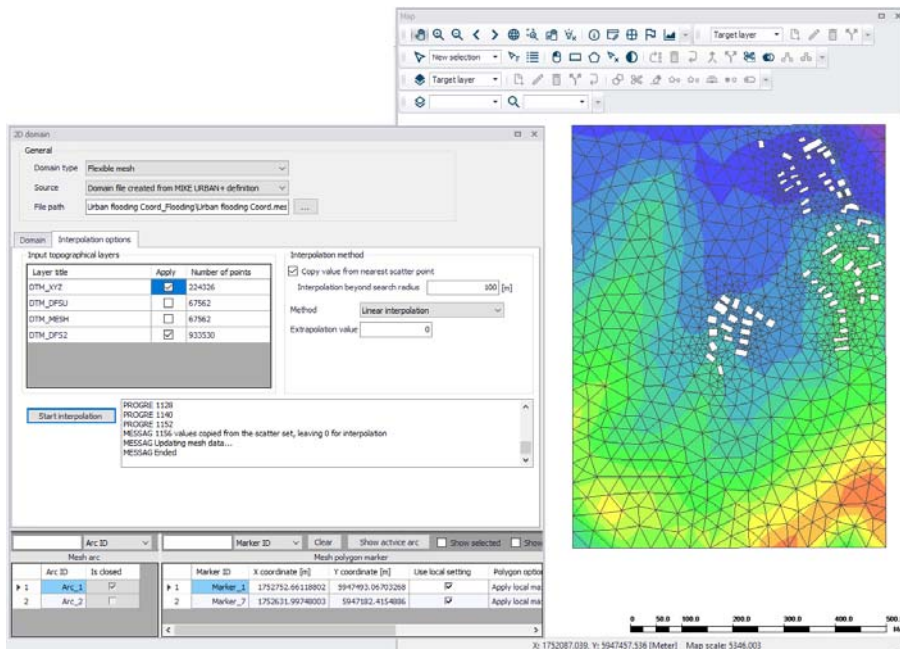
3.2   2D Numerical Settings

Configure 2D model numerical solution settings in the 2D Numerical Settings editor.
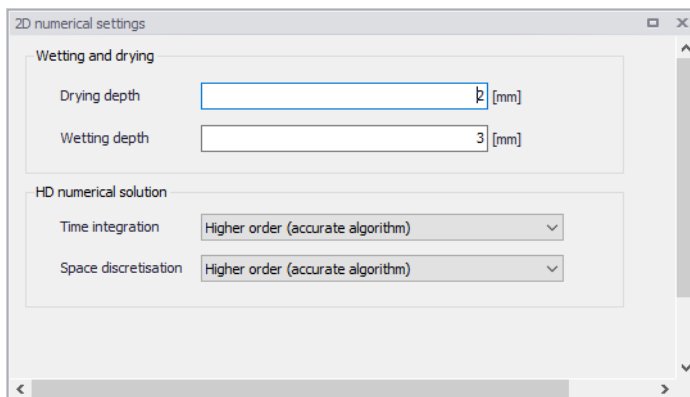
Figure 3.23   The 2D Numerical Settings editor

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 (hdry)
- Wetting depth (hwet)

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 hdry and the water depth at the other side of the face is larger than hwet.

- An element is dry if the water depth is less than hdry 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 hdry and less than hwet, or when the depth is less than hdry 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 hwet. Both the mass fluxes and the momentum fluxes are calculated.
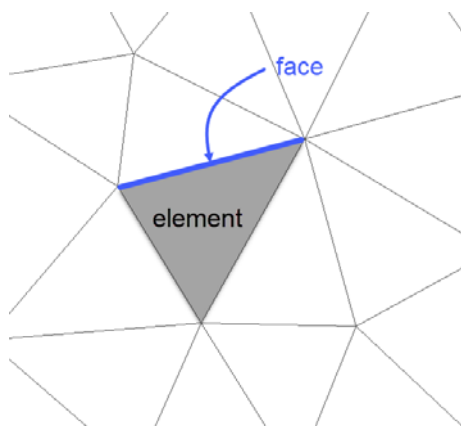
Figure 3.24   Illustration of elements and element faces

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 hu and hv 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 hdry = 5 mm and wetting depth hwet = 10 mm.

The wetting depth, hwet, must be larger than the drying depth, hdry.

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.

3.2.3 AD and MIKE ECO Lab Numerical Solution

Define the numerical solution technique to use in the Transport modules of the 2D overland simulations. The corresponding parameters become visible when the Advection-Dispersion or the MIKE ECO Lab modules are active in the project.

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

- Lower order: Faster but less accurate scheme.
- Higher order: Slower but more accurate scheme.

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

The time integration of the transport (advection-dispersion) equations is performed using an explicit scheme. Due to the stability restriction using an explicit scheme the time step interval must be selected so that the CFL number is less than 1. A variable time step interval is used in the calculation and it is determined so that the CFL number is less than a critical CFL number in all computational nodes. To control the time step it is also possible for the user to specify a minimum time step and a maximum time step. The time step interval for the transport equations is synchronized to match the overall time step specified for the hydrodynamic simulation in the 'Simulation setup' editor.
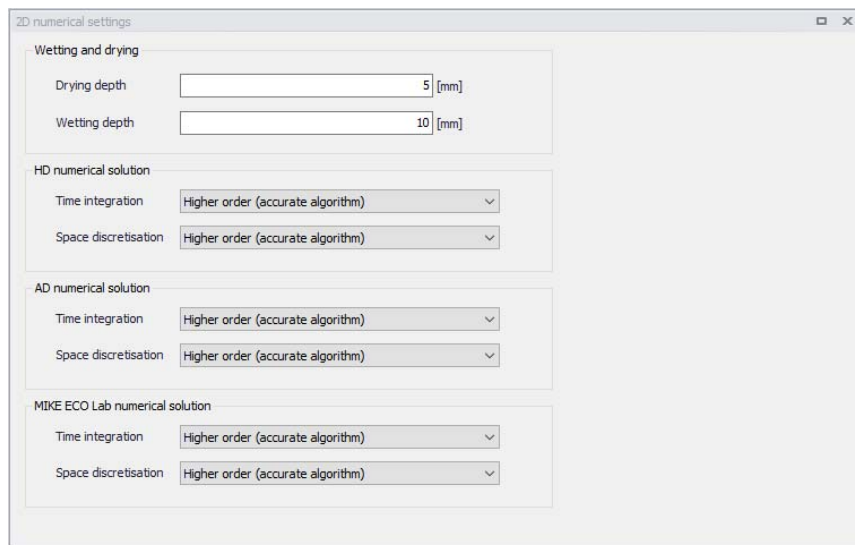
Figure 3.25   The numerical settings with Transport modules active

3.3   2D Fluid Properties

The hydrodynamic solver uses per default fluid properties describing clear water. It is however possible to simulate one-phase flows with different flow characteristics, e.g. oil or water with high concentrations of debris or mud. To enable this, select the 'Water loaded with mud or debris, or oil' type.

A flow resistance relationship for a flow regime equivalent to a Bingham or Non-Newtonian flow approach is calculated under the assumption that the debris-flow material behaves like a viscoplastic fluid. Fluids considered for this solution would typically be oil or debris flow with high concentrated mixtures of flowing sediments and water.

Fluid property parameters to be defined include: Fluid Density, Yield stress and Bingham fluid viscosity.

Please note 2D Precipitation and evaporation, as well as 2D infiltration, are not available in connection with this alternative fluid type and the respective data pages are therefore automatically hidden when activating this option.

3.3.1   General Description

The 'Water loaded with mud or debris, or oil' type enables an alternative hydrodynamic solution in which a fluid dependent flow resistance is added to the standard flow momentum equations.

Naef et al (2006) defines a number of formulations for calculating flow resistance relations and the flow resistance term; $\tau0/\rho$. The current implementation

includes a 'Full Bingham' relation, which in addition allows for applying simpler resistance formulations which excludes the Bingham viscosity term (e.g. the 'Turbulent and Yield' relation as defined in Naef et al (2006)).

The full Bingham flow resistance relation determine the flow resistance term ($\tau 0/\rho gh$) from the following third order equation (see Naef et al, 2006):

$$2\tau_0^3 - 3\left(\tau_y + 2\frac{\mu_B q}{h^2}\right)\tau_0^2 + \tau_y^3 = 0 \tag{3.1}$$

where q is the flux (discharge per unit width), h is the fluid depth, τy is the yield stress and μB is the Bingham fluid viscosity.

The third order equation is solved numerically during the simulation to give $\tau 0$ as function of the yield stress (τy), Bingham viscosity (τB), water depth (h) and flux (q).

Note, that it is possible to activate e.g. the Turbulent and Yield resistance formulation by simply setting the Bingham Viscosity parameter, μB, equal to zero.

3.3.2 Recommended Values

The rheological properties of non-Newtonian fluid are driven by the complex interaction of a fluid's chemical and material composition. Key composition properties include the particle size distribution (e.g. percent fines), solids concentration, water content, chemical composition, and mineralogy such as the presence of clay minerals.

The Bingham rheological model is well suited for homogenous fluid mixtures with high concentrations of fine particles (e.g. mudflows, hyper-concentrations of fine sand, silt, and clay-size sediment) and other material types such as oils.

The key parameters for the Mud/Debris/Oil model are the following (default unit shown in brackets):

Fluid density: Density of the fluid mixture [kg/m^3]

Yield stress: Shear stress threshold that needs to be exceeded for the fluid to flow [Pa]

Dynamic viscosity: Dynamic viscosity of the Bingham fluid mixture [PS s]

Fluid density
The fluid density can be determined from either measurements of the fluid to be modelled or calculated using the solids concentrations of the fluid mixture.

Yield stress

Ideally, the yield stress (i.e. yield strength) of the fluid to be modelled can be determined from rheograms developed from viscometric measurements in a laboratory. A rheogram relates the shear rate of the fluid to the applied shear stress. A commercially available concentric cylindrical viscometer is ideally suited for this type of analysis because it is capable of developing the rheogram for a wide shear rate range. However laboratory derived rheological analyses may not always be possible or practical.

Yield stress can also be determined empirically from both case studies involving similar fluid compositions and empirical relationships. For hyper-concentrations composed of fine sediment, yield stress is often formulated as a function of material type (e.g. clay mineralogy) and sediment concentration. Julien (2010) provides the following recommended empirical relationships for yield stress as a function of sediment concentration for a variety of material types using this exponential form:

$$\tau_y = a \cdot 10^{b \cdot C_v} \tag{3.2}$$

where τy is the yield stress [Pa], a and b are coefficients (see Table 3.1) and C_v is the volumetric sediment concentration.

Table 3.1   Coefficients for yield stress empirical relationships from Julien (2010)

Material	a	b
Bentonite (montmorillonite)	0.002	100
Sensitive clays	0.3	10
Kaolinite	0.05	9
Typical soils	0.005	7.5

Oils are a special application where the yield stress is typically set to zero and the dynamic viscosity dictates the laminar flow nature represented by the Bingham rheological model. For zero yield stress the Bingham fluid model is valid for laminar depth-integrated flow.

Note that the typical exponential relationship between yield stress and sediment concentration indicates that at some point small changes in concentrations can dramatically change yield stress. This is an important dynamic sensitivity to consider when evaluating Bingham fluids.

Dynamic viscosity

Once the fluid is in motion, the dynamic viscosity (i.e. plastic viscosity) represents how the fluid flows under applied shear stresses. Similar to yield stress, the dynamic viscosity can be determined from rheograms developed from viscometric measurements in a laboratory. However laboratory derived rheological analyses may not always be possible or practical.

Dynamic viscosity can also be determined empirically from both case studies involving similar fluid compositions and empirical relationships. Julien (2010) provides the following recommended empirical relationships for yield stress as a function of sediment concentration for a variety of material types using this exponential form:

$$\mu_m = 0{,}001 \cdot 10^{c \cdot C_v} \tag{3.3}$$

where μm is the dynamic viscosity [Pa s], c is a coefficient (see Table 3.2) and Cv is the volumetric sediment concentration.

Table 3.2   Coefficients for dynamic viscosity empirical relationships from Julien (2010)

Material	c
Bentonite (montmorillonite)	100
Sensitive clays	10
Kaolinite	9
Typical soils	7.5

For modelling viscous, low-strength fluids such as oils, the dynamic viscosity is the key parameter for the Bingham model as the yield stress is often set to zero. The dynamic viscosity for such materials is best determined from rheograms developed from viscometric measurements in a laboratory, e.g. commercially available concentric cylindrical viscometer. Available literature (e.g. product descriptions) and case studies for commercially derived materials are other appropriate sources for choosing the value for the dynamic viscosity parameter.

3.3.3   Remarks and Hints

The Bingham fluid viscosity must be defined as a Dynamic viscosity.

When a Bingham formulation is activated and a resistance relation is calculated from this, the turbulent resistance (Bed resistance) defined by Manning or Chezy values should ideally be deactivated. However, in situations where Yield/Bingham stresses are not dominating, turbulent resistance may be crucial to maintain model stability, and hence, it is important to realize that turbulent resistance and the resistance relation from the Bingham solution are both active and included as individual resistance contributions in the hydrodynamic solution.

3.4   2D Infrastructures

A special numerical treatment can be applied to two types of infrastructures in the 2D domain: buildings and roads. The spatial extent of buildings and roads is defined by zones. A zone is a series of polygons from a GIS layer, all being applied the same properties. Zones are therefore used as categories of buildings or roads.

The topography in the 2D domain should be given as natural ground level and for buildings and roads it is possible to specify a correction of the topography. A special treatment of the flux across faces on the borders of buildings and roads is also applied in the simulation.

Three types of file format can be used for defining the zones:

- Shape file
- DFS2 or DFSU file
- XYZ file

For XYZ files and shape files, the zones are defined as the closed region within a number of polygons. For dfs files, the zones are defined by a map identifying the location of the different zones. Each zone is identified by an integer number larger than zero. If there are overlapping zones for the two types of zones and/or with land cover zones (e.g. specified for 2D surface roughness definition), the priority is: Building, road and land cover. If an element from the 2D domain belongs to more than one polygon in the XYZ file or the shape file, the information from the last polygon read from the file is applied.

For dfs files, the value should be zero in areas with no building or road.

For XYZ files, the data in the file must be formatted as ASCII text in five columns with the two first columns giving the x-and y-coordinates of the points. The third column represents connectivity. The connectivity column is used to define arcs. All points along an arc - except the last point - shall have a connectivity value of 1 and the last point shall have a connectivity value of 0. The fourth column contains the z-value at the point. Finally, the fifth column contains the zone number. The z-coordinate is only used when the XYZ file is used to define building zones. Here the Z-value must contain the height of the buildings or the level of the top of the buildings.

Note that the 2D domain mesh / grid elements don't have to be aligned with the zones polygons: the zones will automatically be mapped to the nearest element faces. The use of 2D infrastructures can therefore avoid meshing buildings with very small mesh elements, which improves model stability; but the mesh should not be too coarse compared to the buildings size in order to correctly take these buildings into account.

The use of 2D infrastructures is particularly suited when applying rainfall on the 2D domain because the volume of rain falling on buildings remains accounted for (as opposed to when buildings are excluded from the mesh / grid), and the effect of the buildings on the flow is also included.

General description

The flux across an infrastructure face - a building face or a road face - is calculated using an empirical formula or the flow equations depending on the surface elevation. A face in the 2D mesh is defined as building face, if the element to one side of the face belongs to a building zone and the element to the other side belongs to a road zone or to none of the zones. Also, a face where the elements on both sides of the face belong to a building zone and where there is a difference in the level of the buildings, is defined as a building face. A face in the 2D mesh is defined as road face, if the element to one side of the face belongs to a road zone and the element to the other side belongs to none of the zones.

The structure level for an infrastructure face is defined as the maximum of the level for the two elements on each side of the face. If the water level in both elements on each side of the infrastructure face is above the structure level, the flow equations are applied and otherwise the empirical formula is applied. If a building zone is excluded from the calculation the flux across the corresponding building faces are always zero and the elements in the zone are treated as dry elements.

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

$$Q = WC(H_{us} - H_w)^k \left[1 - \left(\frac{H_{ds} - H_w}{H_{us} - H_w} \right) \right]^{0,385} \tag{3.4}$$

where Q is discharge through the structure, W is width, C is weir coefficient, k is the weir exponential coefficient, Hus is upstream water level, Hds is downstream water level and Hw is the structure level. The value of the weir exponent is 1.5 and the default value of the weir coefficient is 1.838.

Building zones

When 'Include building zones' has been selected, the following items must be specified:

- Layer: The source layer defining the building zones

- Zones attribute / item: For a shape file or dfs file, this is the attribute or item in the file containing the integer number, used to identify the zone each polygon belongs to.

- Map projection: For a XYZ file, this is the map projection in which the XY coordinates in the file are defined.

- Height type: Using a XYZ file or a shape file, buildings heights can either be defined as a relative height above ground level or as an absolute elevation. Using a dfs file, the height of a building is specified by the level of the top of the building.

- Height definition: Buildings heights, controlling the correction of the topography, can either be uniform (same height for all zones) or read for each zone from the source layer.

- Height attribute / item: This is the attribute / item in the source file containing the height value for each zone, when the height is read from the source layer.

- Zones table: For each of the zones, it is required to specify:

  – Zone ID: Zones are initially named using the corresponding integer number in the source layer, but it is possible to rename the zone to more appropriate names.
  – Include: It is possible to specify if the building should be included or excluded from the calculation. If the buildings are excluded, the building elements will be treated as dry elements.
  – Uniform height / elevation: The relative height or absolute elevation of the buildings in the zone. When using relative heights, you must specify the height of the buildings as the distance from the natural ground level to the highest point of the building. The top level of the building is then calculated from the maximum ground level in the area of the building. When using absolute elevations, the height of the building is specified as the top level of the building.
  – Discharge coefficient: For each building zone you also have to specify a discharge / runoff coefficient. This coefficient must be larger or equal to zero and lower or equal to one. The coefficient is multiplied to the flux across a building face.

- 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. This depth is a critical water level difference below which the discharge gradients are suppressed. The default value is 0.01 meter. If a structure shows oscillatory behaviour it is recommended to increase this value slightly.

- Weir coefficient: The weir coefficient used to compute the discharge through the building face.

Figure 3.26   Defining building zones from a shape file layer

Road zones

When 'Include road zones' has been selected, the following items must be specified:

- Layer: The source layer defining the road zones

- Zones attribute / item: For a shape file or dfs file, this is the attribute or item in the file containing the integer number, used to identify the zone each polygon belongs to.

- Map projection: For a XYZ file, this is the map projection in which the XY coordinates in the file are defined.

- Offset definition: Using dfs files, roads' offset - controlling the correction of the topography - can either be uniform (same offset for all zones) or read for each zone from the source layer. Using a XYZ file or a shape file, the offset can only be specified as a constant value for each road zone.

- Zones table: For each of the zones, it is required to specify:

  – Zone ID: Zones are initially named using the corresponding integer number in the source layer, but it is possible to rename the zone to more appropriate names.
  – Uniform offset: The offset for each road zone, when using a uniform offset.

- 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. This depth is a critical water level difference below which the discharge gradients are suppressed. The default value is 0.01 meter. If a structure shows oscillatory behaviour it is recommended to increase this value slightly.

- Weir coefficient: The weir coefficent used to compute the discharge through the road face.

Description
Use the Description tab to add free text descriptions for buildings and roads.

3.5 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.

3.5.1 Dry Domain

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

3.5.2 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.

3.5.3 Varying in Domain

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

- MIKE+ 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+ 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+ 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.

3.6   2D Surface Roughness

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

- Uniform
- Varying in Domain
- Varying in Domain and Time
- Varying in domain and flow dependent

3.6.1   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.

3.6.2 Varying in Domain

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

- MIKE+ 2D Surface Roughness Layer
- Background Layer of Individual Polygons
- 2D File

MIKE+ 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 polygons:

- 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 of Individual Polygons

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+ 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.

Background Layer of Land Cover

The 'Background layer of land cover' option allows defining the surface roughness as a function of land cover, i.e. per land cover type. A polygon layer must be selected to define the land cover, in which the polygons are grouped by category (land cover type). Three types of file format can be used for defining the land cover types:

- Shape file

- DFS2 or DFSU file

- XYZ file

For XYZ files and shape files, the land cover zones are defined as the closed region within a number of polygons. For dfs files, the zones are defined by a map identifying the location of the different zones. Each land cover type is identified by an integer number larger than zero. If an element from the 2D domain belongs to more than one polygon in the XYZ file or the shape file, the information from the last polygon read from the file is applied. If there are

overlapping zones with the buildings and road zones (from the 2D infrastructures editor), the priority is first given to the building zones, then to the road zones and finally to the land cover zones.

For dfs files, the value should be zero in areas with no local infiltration zone in a dfs file.

For XYZ files, the data in the file must be formatted as ASCII text in five columns with the two first columns giving the x-and y-coordinates of the points. The third column represents connectivity. The connectivity column is used to define arcs. All points along an arc - except the last point - shall have a connectivity value of 1 and the last point shall have a connectivity value of 0. The fourth column contains the z-value at the point, which is unused for land cover layers. Finally, the fifth column contains the zone number, corresponding to the land cover type.

In the 'Land cover layer' section , the following needs to be specified:

- Layer: The source layer defining the land cover zones
- Item: For a shape file or dfs file, this is the attribute or item in the file containing the integer number, used to identify the land cover type each polygon belongs to.
- Map projection: For a XYZ file, this is the map projection in which the XY coordinates in the file are defined.

In the 'Roughness values' section, the roughness formula must be selected (Manning M, Manning n or Chezy C), and the roughness values must be specified for each zone in the table. The list of zones is made of the list of land cover types plus the list of road zones and / or buildings zones eventually selected in the 2D infrastructures editor. The land cover ID for these zones is editable. A default roughness value must also be supplied, which is applied in areas not covered by any of the previous zones.
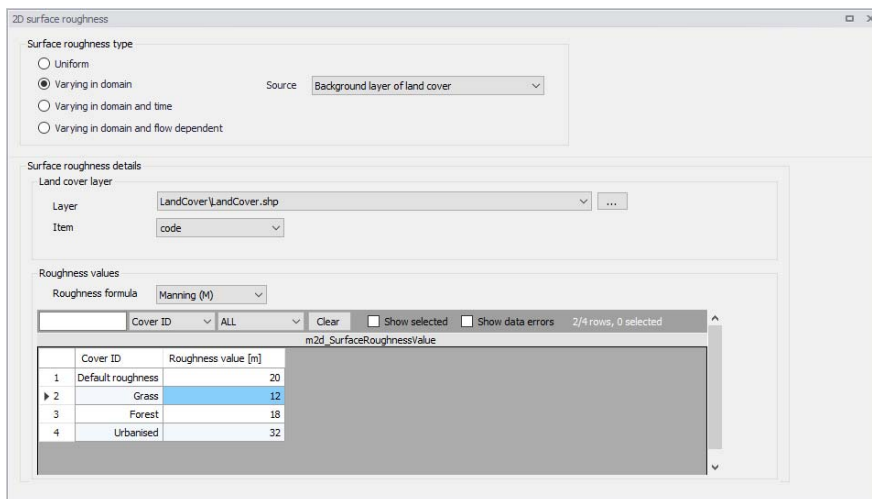
Figure 3.27   Defining roughness values for land cover zones

3.6.3   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+ 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.

3.6.4   Varying in Domain and Flow Dependent

The 'Varying in domain and flow dependent' option allows defining the surface roughness as a function of the flow on the surface (water depth or flux) and as a function of land cover, i.e. per land cover type. A polygon layer must be selected to define the land cover, in which the polygons are grouped by

category (land cover type). Three types of file format can be used for defining the land cover types:

- Shape file
- DFS2 or *.DFSU file
- XYZ file

For XYZ files and shape files, the land cover zones are defined as the closed region within a number of polygons. For dfs files, the zones are defined by a map identifying the location of the different zones. Each land cover type is identified by an integer number larger than zero. If an element from the 2D domain belongs to more than one polygon in the XYZ file or the shape file, the information from the last polygon read from the file is applied. If there are overlapping zones with the buildings and road zones (from the 2D infrastructures editor), the priority is first given to the building zones, then to the road zones and finally to the land cover zones.

For dfs files, the value should be zero in areas with no local infiltration zone in a dfs file.

For XYZ files, the data in the file must be formatted as ASCII text in five columns with the two first columns giving the x-and y-coordinates of the points. The third column represents connectivity. The connectivity column is used to define arcs. All points along an arc - except the last point - shall have a connectivity value of 1 and the last point shall have a connectivity value of 0. The fourth column contains the z-value at the point, which is unused for land cover layers. Finally, the fifth column contains the zone number, corresponding to the land cover type.

In the 'Land cover layer' section , the following needs to be specified:

- Layer: The source layer defining the land cover zones
- Item: For a shape file or dfs file, this is the attribute or item in the file containing the integer number, used to identify the land cover type each polygon belongs to.
- Map projection: For a XYZ file, this is the map projection in which the XY coordinates in the file are defined.

In the 'Roughness values' section, the roughness curves must be specified for each zone in the table. The list of zones is made of the list of land cover types plus the list of road zones and / or buildings zones eventually selected in the 2D infrastructures editor. The land cover ID for these zones is editable. A default roughness curve must also be supplied, which is applied in areas not covered by any of the previous zones. For each zone, the data to be specified are:

- Function of: Controls whether the Manning number is specified as a function of the water depth or as a function of the flux.

- Roughness curve: The curve defining the Manning number as a function of the depth or flux. When the roughness is defined as a function of water depth, the valid curve types are either 'Depth dependent Manning (M)' or 'Depth dependent Manning (n)'. When the roughness is defined as a function of flux, the valid curve types are either 'Flux dependent Manning (M)' or 'Flux dependent Manning (n)'.

The 'Create curves' button can be used to create curves with default values for each of the zones. The curve data are accessed via the Curves and Relations editor.
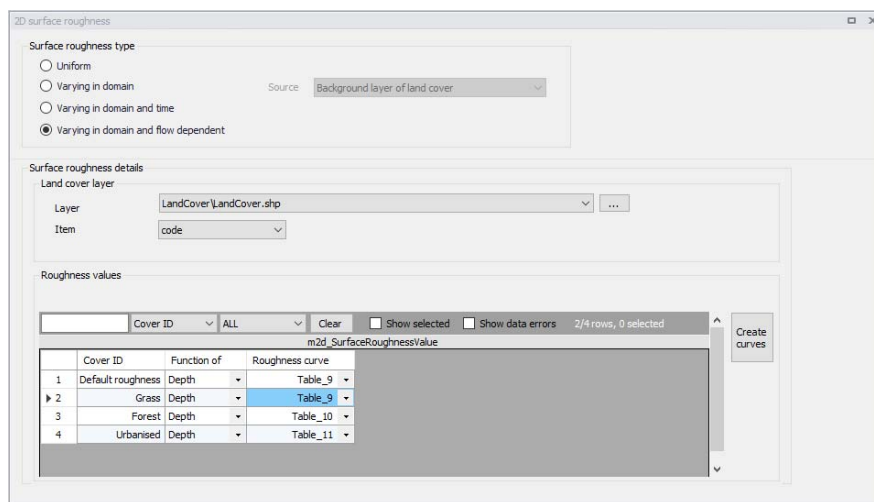
Figure 3.28   Defining flow-dependent roughness values for land cover zones

3.6.5   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

$$\frac{\tau}{\rho_0} = c_f\, u|u| \tag{3.5}$$

where cf is the drag coefficient, u is the velocity and $\rho 0$ is the density of the water.

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

$$c_f = \frac{g}{C^2} \tag{3.6}$$

$$c_f = \frac{g}{(Mh^{1/6})^2} \tag{3.7}$$

where h is the total water depth and g is the gravitational acceleration. The units of Chezy numbers and Manning numbers are m1/2/s and m1/3/s respectively.

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

$$M = \frac{25.4}{k_s^{1/6}} \tag{3.8}$$

3.6.6   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 m1/3/s are normally used.

If Chezy numbers are applied, a model calibration can normally be achieved with values in the range 30 - 50 m1/2/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

3.6.7   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 m1/3/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 h1/6 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.

3.7   2D Eddy Viscosity

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

- None
- Uniform
- Varying in Domain
- Smagorinsky Formulation

3.7.1   None

The eddy viscosity terms are omitted from the calculations.

3.7.2   Uniform

A constant eddy viscosity value in m2/s is specified that applies to the entire 2D domain.

3.7.3   Varying in Domain

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

- MIKE+ 2D Eddy Viscosity Layer
- 2D File

MIKE+ 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.

3.7.4 Smagorinsky Formulation

Dynamically calculate eddy viscosity by means of the Smagorinsky formula.

A minimum and maximum value (m2/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.

3.7.5 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:

$$\frac{E \cdot \Delta t}{\Delta l^2} \leq \frac{1}{2} \tag{3.9}$$

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

$$\nu_t = c_s^2 \, l^2 \sqrt{2 S_{ij} S_{ij}} \tag{3.10}$$

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

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

(3.11)

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

3.7.6 Recommended Values

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

3.7.7 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.

3.8 2D Structures

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

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

3.9 2D Dikes

In MIKE+, 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 flood-plain or along low-lying coastlines.

3.9.1 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.

3.9.2 Insert from File

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

The format for XUYZ files 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+ project projection.

For shape files, if the input file is a 3D shape file (containing crest levels in the lines geometry), then all the lines from the file are imported simultaneously as separate dikes, and the crest levels are also imported from the file. For regular shape files, each file should contain a single line corresponding to a single dike.

3.9.3 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 3.42).

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.
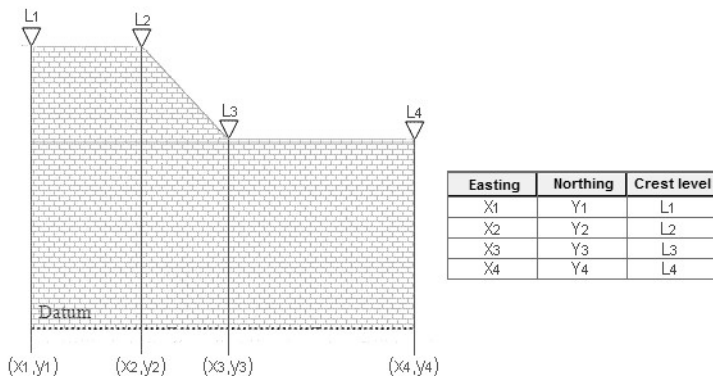
Figure 3.29   Definition sketch of spatial varying dike geometry

3.9.4   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 :

$$Q = wC(H_{us} - H_w)^{3/2}\left(1 - \left(\frac{H_{ds} - H_w}{H_{us} - H_w}\right)^{3/2}\right)^{0,385} \quad \text{for } H_{us} > H_{ds} > H_w$$

$$Q = wC(H_{us} - H_w)^{3/2} \qquad\qquad\qquad\qquad \text{for } H_{us} > H_w \geq H_{ds}$$

$$Q = 0 \qquad\qquad\qquad\qquad\qquad\qquad\qquad\quad \text{for } H_w \geq H_{us} > H_{ds}$$

(3.12)

where Q is discharge through the structure, w is the local width (cell face width), C is discharge coefficient, Hus is upstream water level, Hds is downstream water level and Hw is the crest level taken with respect to the global datum.

The default value of the weir coefficient is 1.838.
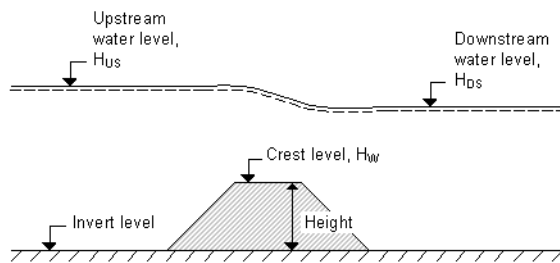
Figure 3.30   Definition sketch for Dike Flow

3.9.5   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 hydro-dynamic simulation. A linear interpolation will be applied if the time steps differ.

3.10 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+. 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.

3.10.1 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 represent a cross section of the channel. Element faces from the mesh or grid are then selected based on this cross section: an element face is included in the culvert's section when the line between its two neighboring elements' centres crosses one of the cross section segments (see 3.13 Structures General Description *(p. 81)* for more details). To identify the exact faces where the culvert's calculation applies, the 2D log file can be interrogated as it will list all element faces selected along the cross section.

During the model initialisation period, the engine will look at the ground level in the cells / mesh elements on both sides of the selected element faces, and check if they are greater than the culvert invert. If these ground levels are higher than the culvert invert level, then an error message will be thrown. Therefore, it is important to make sure that the ground levels (defined in the 2D domain file) 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 cell / element size of the 2D domain. Thus when using a culvert in a 2D domain with a fine resolution, it is important to take care with schematization, as described in the example below. Culverts that are much longer than the cell/element size can instead be modelled as a long culvert.

Example of a short culvert setup:

There are two images in Figure 3.31 below illustrating a case with a stream running south to north and a road bisecting the stream. The image on the left shows the raw data, where the road is described in the topography and where the black line represents the culvert's cross section location. This setup does not work for a short culvert, because the topography in the cells downstream the culvert's cross section is higher than the culvert invert level.

The right image illustrates the grid modified in order to fit the requirements for using a short culvert. The water level upstream of the culvert is calculated from cells 1 and 2 and the downstream water level calculated from cells 3 and 4. The flow through the culvert will be computed using the resulting upstream and downstream water levels, and the culvert's geometry.
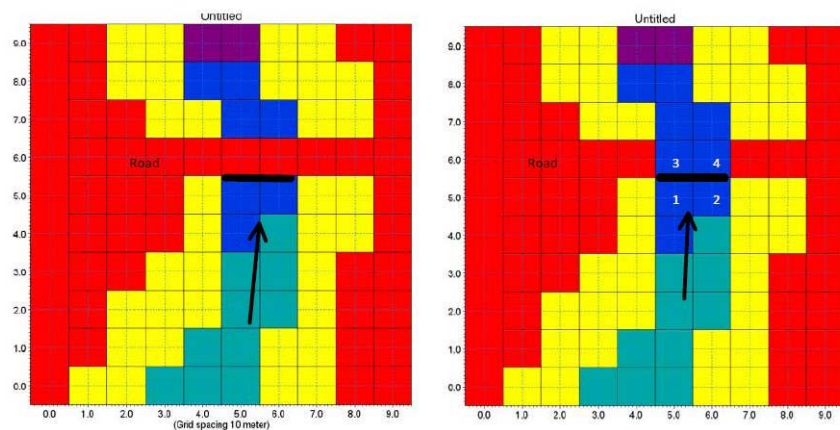
Figure 3.31   Setup of a short culvert

Note: if the overflow above the road needs to be represented, it can be computed using a weir placed at the same location as the 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 3.32 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.
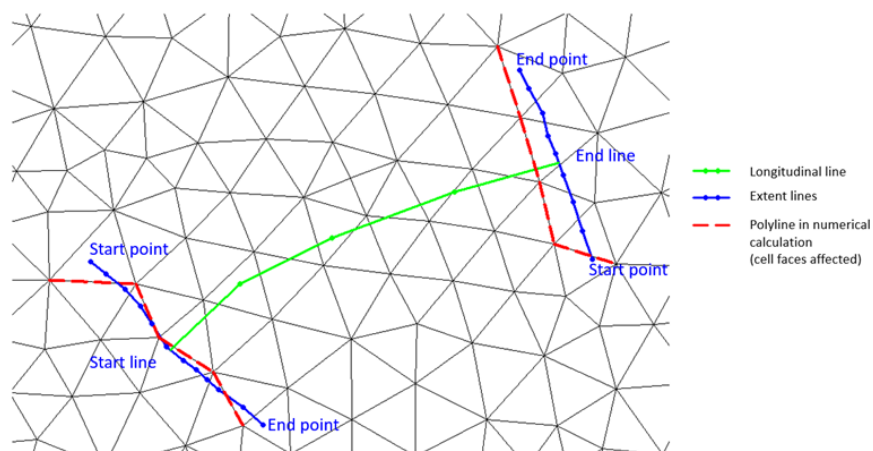
Figure 3.32   Definition sketch for a long culvert

3.10.2 Insert

From the MIKE+ 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 3.32).

3.10.3 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 3.32).

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

3.10.4 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.
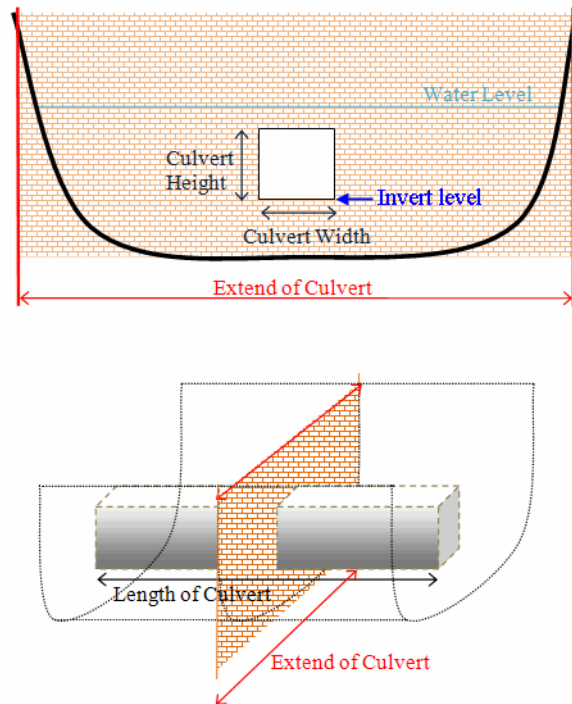
Figure 3.33   Setup definition of culvert

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
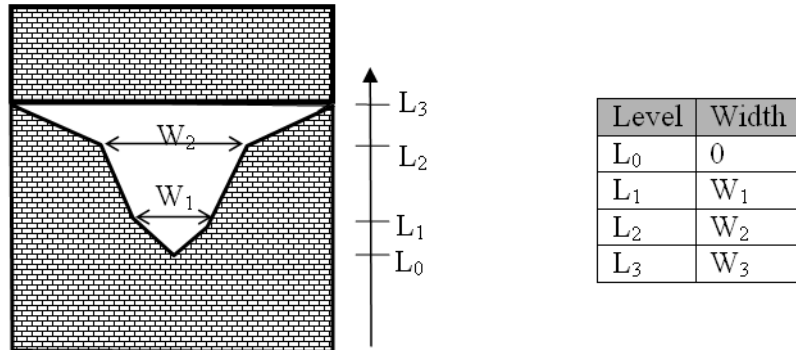
Figure 3.34   Definition sketch for irregular culvert

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)

3.10.5 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 3.35   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 h5/3, 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 3.35.

For a long culvert the flow direction is positive when flow is from the start line to the end line (see Figure 3.32).

Calibration factors can also be applied to the head losses.

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

$$\Delta H_{loss} = \frac{q^2}{2g}\left(\frac{\zeta_1}{A_{s_1}^2} + \frac{\zeta_f}{A_{s_A}^2} + \frac{\zeta_2}{A_{s_2}^2} \right)$$
(3.13)

where AS1, AS1 and AS1 are the mean cross section areas along the length of the culvert and q is the discharge, ζ1 is the entrance or contraction loss

coefficient, $\zeta2$ is the outlet or expansion loss coefficient and ζf is the friction loss. $\zeta1$ and $\zeta2$ are calculated by

$$\zeta_1 = \zeta_{in}\left(1 - \frac{A_s}{A_1}\right)$$
(3.14)

$$\zeta_2 = \zeta_{out}\left(1 - \frac{A_s}{A_2}\right)^2$$
(3.15)

The upstream and downstream cross section areas, A1 and A2 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, ζin and ζout. The friction loss coefficient is calculated using the Manning formula

$$\zeta_f = \frac{2gLn^2}{R^{4/3}}$$
(3.16)

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.

3.11   2D Weirs

In MIKE+, 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).

3.11.1   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.

3.11.2 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.

3.11.3 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 3.42).

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.

3.11.4 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.
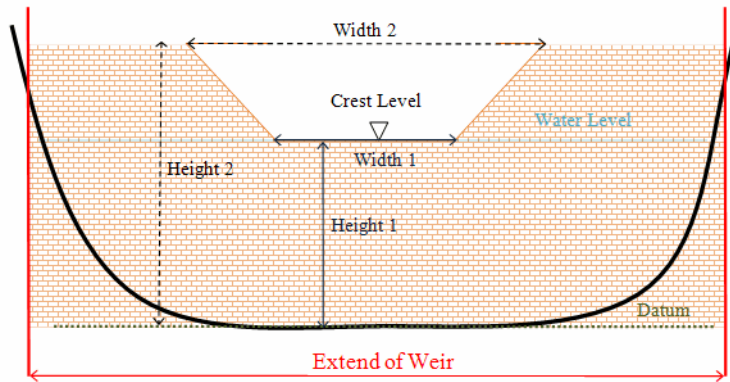
Figure 3.36   Setup definition of contracted weir
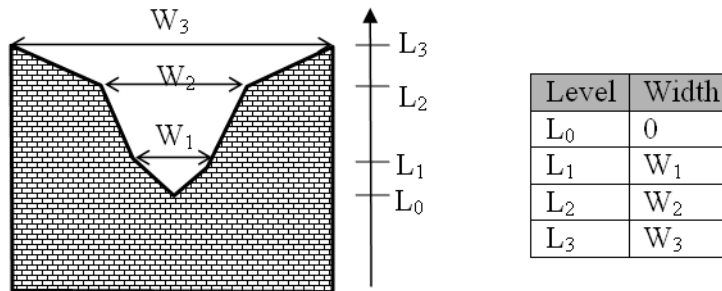
Figure 3.37   Definition sketch of broad crested weir geometry

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 79). 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:

$$q = \zeta_t \frac{V_s}{2g} \tag{3.17}$$

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.

$$\zeta_t = \zeta_1 + \zeta_2 = \zeta_{in}\left(\frac{V_1}{V_s}\right) + \zeta_{out}\left(\frac{A_s}{A_2}\right)^2 \qquad (3.18)$$

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

$$\zeta_t = \zeta_1 + \zeta_2 = \zeta_{in} + \zeta_{out} \qquad (3.19)$$

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.
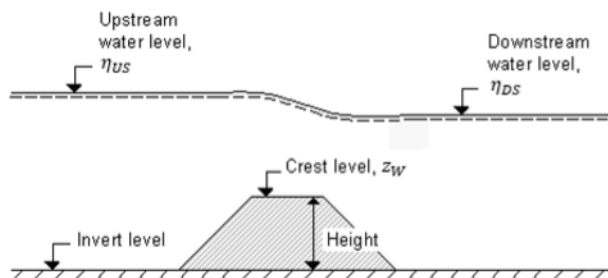
Villemonte Formula

Figure 3.38   Definition sketch for Weir Flow

For this type of weir the width, height and invert level for the weirs must be specified (Figure 3.38). 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.

$$q = C(\eta_{US} - z_w)^k \left(1 - \frac{\eta_{DS} - z_w}{\eta_{US} - z_w}\right)^{-0,385} \cdot W \tag{3.20}$$

where q is the discharge through the structure, w is the width, C is the weir coefficient, k is the weir exponential coefficient, ηUS is the upstream water level, ηDS is the downstream water level and zw 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 3.38). 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:

$$q = \begin{cases} C_1(\eta_{US} - z_w)\sqrt{\eta_{US} - z_w}\,W & \frac{(\eta_{DS} - z_w)}{\eta_{US}} < \frac{2}{3} \\[2ex] C_1(\eta_{DS} - z_w)\sqrt{\eta_{US} - \eta_{DS}}\,W & \frac{(\eta_{DS} - z_w)}{\eta_{US}} \geq \frac{2}{3} \end{cases} \tag{3.21}$$

where q is the discharge through the structure, W is the width, C1 and $C_2 = 1,5\sqrt{3}\,C_1$ are the two weir coefficients, ηUS is the upstream water level, ηDS is the downstream water level and zw is the weir level.

3.11.5 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 3.39   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 h5/3, 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.

3.12 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 coordi-

nates 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 3.40.
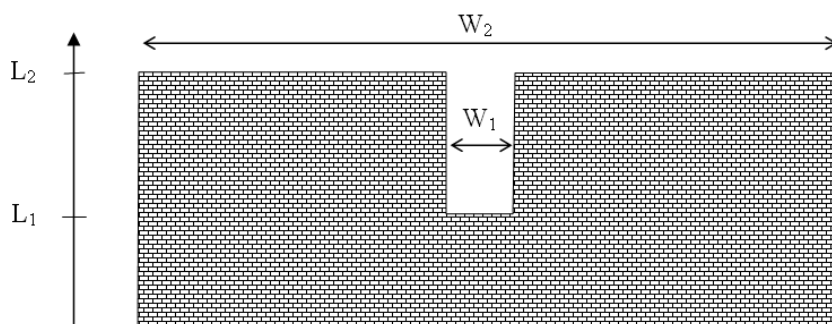
Figure 3.40   Wide weir with a small opening

There are a number of possibilities when modeling this:

1. One weir

2. Two weirs with width W1 and width (W2-W1), respectively

3. Three weirs with widths W1 and two with width (W2-W1)/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 (L2) 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 3.41.
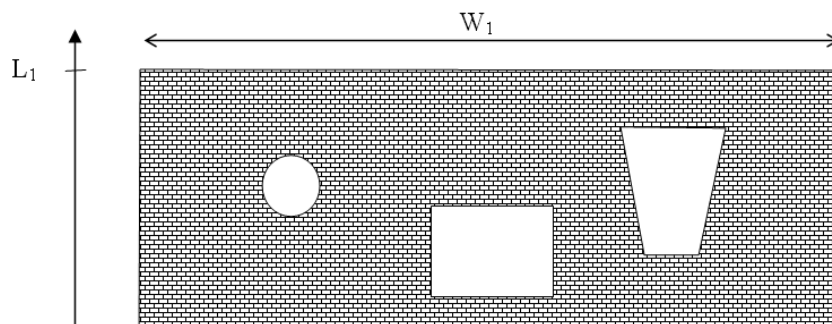
Figure 3.41   Wide weir with multiple culvert openings

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.

3.13   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 3.42. 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.
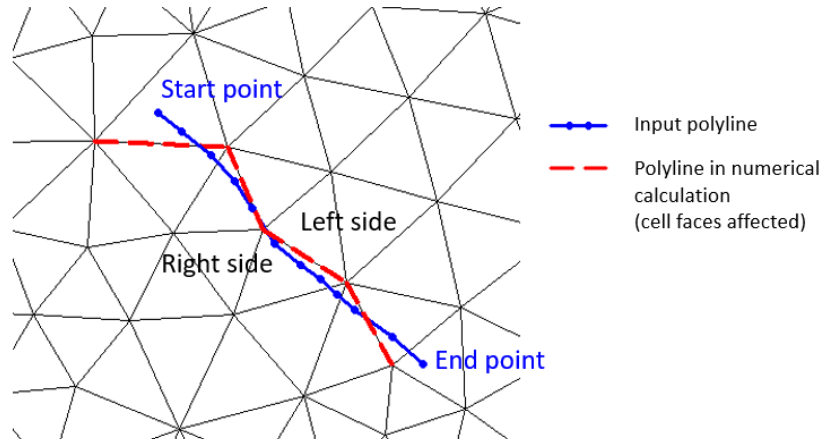
Figure 3.42   The location of a line section

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
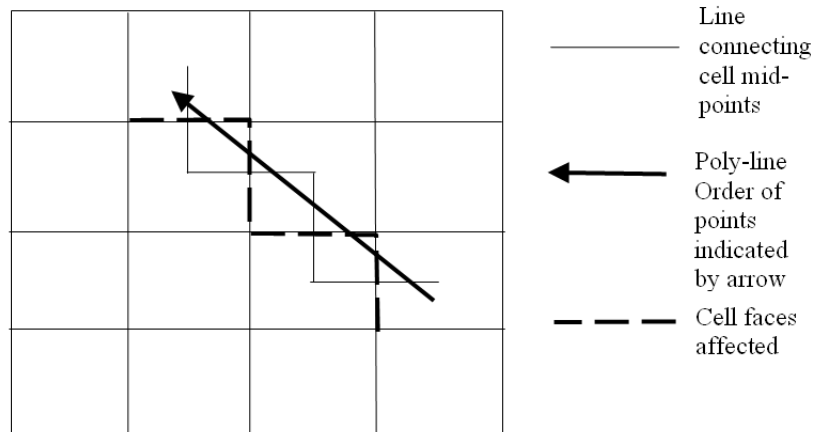
Figure 3.43   Definition of cell mid-points

3.14   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+.

Water quality (AD) computations are possible for 2D overland models and couplings in models of type 'Rivers, collection system and overland flows.'However, at present, water quality calculations for 2D overland models or couplings are unavailable for models of type 'SWMM5 collection system and overland flows,' and thus, sections and editors related to data inputs for 2D overland WQ modelling are irrelevant for such model types.

4.1 2D Boundary Conditions

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

If the 2D domain source is 'Domain file created from MIKE+ 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+.

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+:

- 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

Note: The location of source boundary conditions is defined or edited using the coordinates defined in the '2D boundary conditions' editor. It is different for the other types of boundary conditions which must always be drawn or edited on the map. Therefore, it is not possible to change a source boundary to another type of boundary condition, from the 'Type' list.

4.1.1   Defining 2D Boundary Conditions

Example workflows for setting-up 2D boundary conditions in MIKE+ 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+ (i.e. Source = Domain file from MIKE+ definition).

1. Ensure the new domain grid/mesh has been generated e.g. following instructions in Chapter 3.1 2D Domain *(p. 16)*.

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.
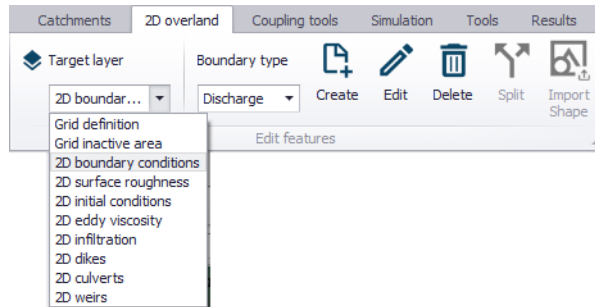
Figure 4.2   The Edit features toolbox on the 2D Overland menu ribbon
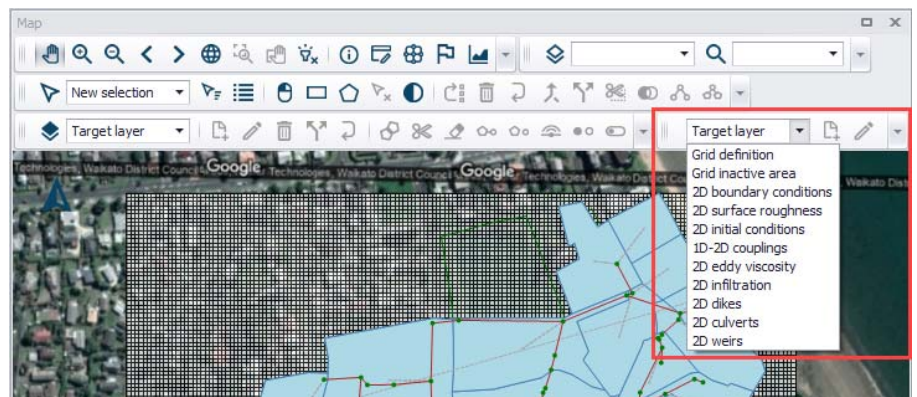
Figure 4.3   The Flooding Layer Editing Tools toolbar on the Map

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

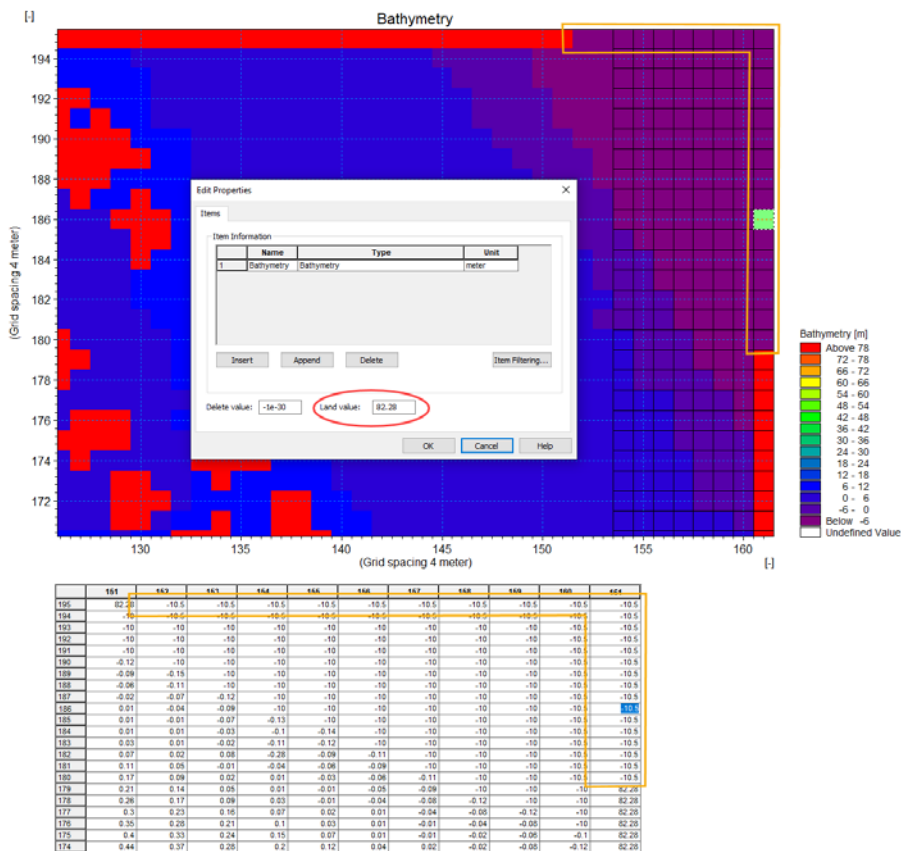|	151	152	153	154	155	156	157	158	159	160	161
195	82.28	-10.5	-10.5	-10.5	-10.5	-10.5	-10.5	-10.5	-10.5	-10.5	-10.5
194	-10	-10.5	-10.5	-10.5	-10.5	-10.5	-10.5	-10.5	-10.5	-10.5	-10.5
193	-10	-10	-10	-10	-10	-10	-10	-10	-10	-10.5	-10.5
192	-10	-10	-10	-10	-10	-10	-10	-10	-10	-10.5	-10.5
191	-10	-10	-10	-10	-10	-10	-10	-10	-10	-10.5	-10.5
190	-0.12	-10	-10	-10	-10	-10	-10	-10	-10	-10.5	-10.5
189	-0.09	-0.15	-10	-10	-10	-10	-10	-10	-10	-10.5	-10.5
188	-0.06	-0.11	-10	-10	-10	-10	-10	-10	-10	-10.5	-10.5
187	-0.02	-0.07	-0.12	-10	-10	-10	-10	-10	-10	-10.5	-10.5
186	0.01	-0.04	-0.09	-10	-10	-10	-10	-10	-10	-10.5	-10.5
185	0.01	-0.01	-0.07	-0.13	-10	-10	-10	-10	-10	-10.5	-10.5
184	0.01	0.01	-0.03	-0.1	-0.14	-10	-10	-10	-10	-10.5	-10.5
183	0.03	0.01	-0.02	-0.11	-0.12	-10	-10	-10	-10	-10.5	-10.5
182	0.07	0.02	0.08	-0.28	-0.09	-0.11	-10	-10	-10	-10.5	-10.5
181	0.11	0.05	-0.01	-0.04	-0.06	-0.09	-10	-10	-10	-10.5	-10.5
180	0.17	0.09	0.02	0.01	-0.03	-0.06	-0.11	-10	-10	-10.5	-10.5
179	0.21	0.14	0.05	0.01	-0.01	-0.05	-0.09	-10	-10	-10	82.28
178	0.28	0.17	0.09	0.03	-0.01	-0.04	-0.08	-0.12	-10	-10	82.28
177	0.3	0.23	0.16	0.07	0.02	0.01	-0.04	-0.08	-0.12	-10	82.28
176	0.35	0.28	0.21	0.1	0.03	0.01	-0.01	-0.04	-0.08	-10	82.28
175	0.4	0.33	0.24	0.15	0.07	0.01	-0.01	-0.02	-0.06	-0.1	82.28
174	0.44	0.37	0.28	0.2	0.12	0.04	0.02	-0.02	-0.08	-0.12	82.28

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+ 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.
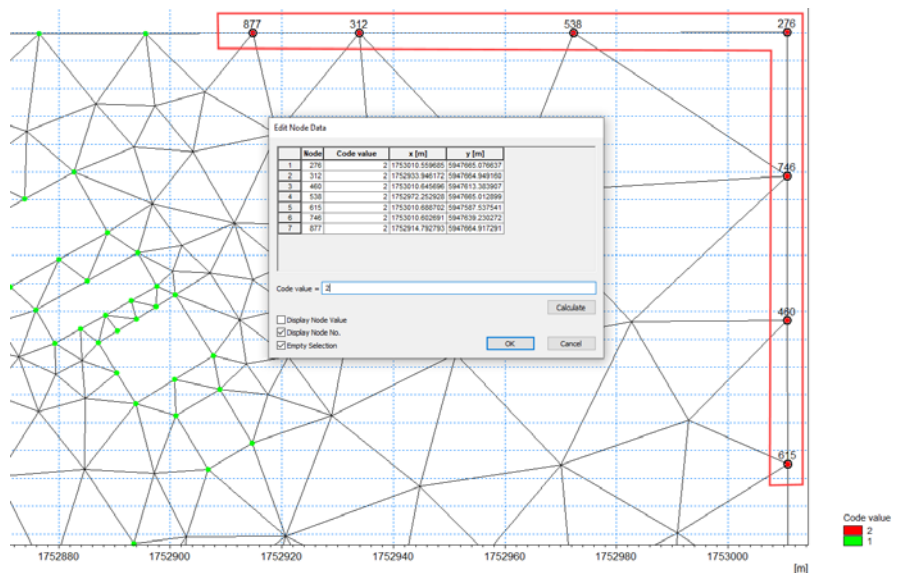

Figure 4.5   Mesh file viewed in the Data Viewer showing the 'Code value' properties for mesh perimeter nodes (outlined in red). Code values may be shown on the map by activating the corresponding data layer from the toolbar, or by selecting node points from the map.

3.   Configure all the defined boundary conditions in the 2D Boundary Conditions Editor (Boundary Conditions|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+. 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. 100)* 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. 100)*.

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.
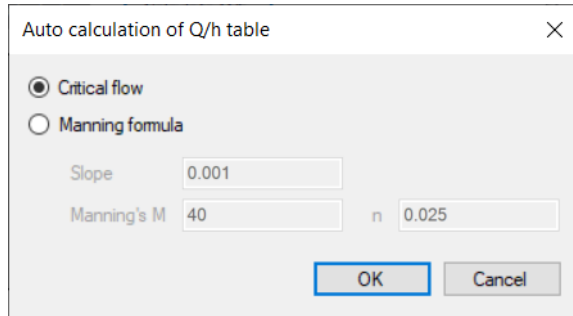

Figure 4.9   The dialog for configuring the auto-calculation of Q/h table values

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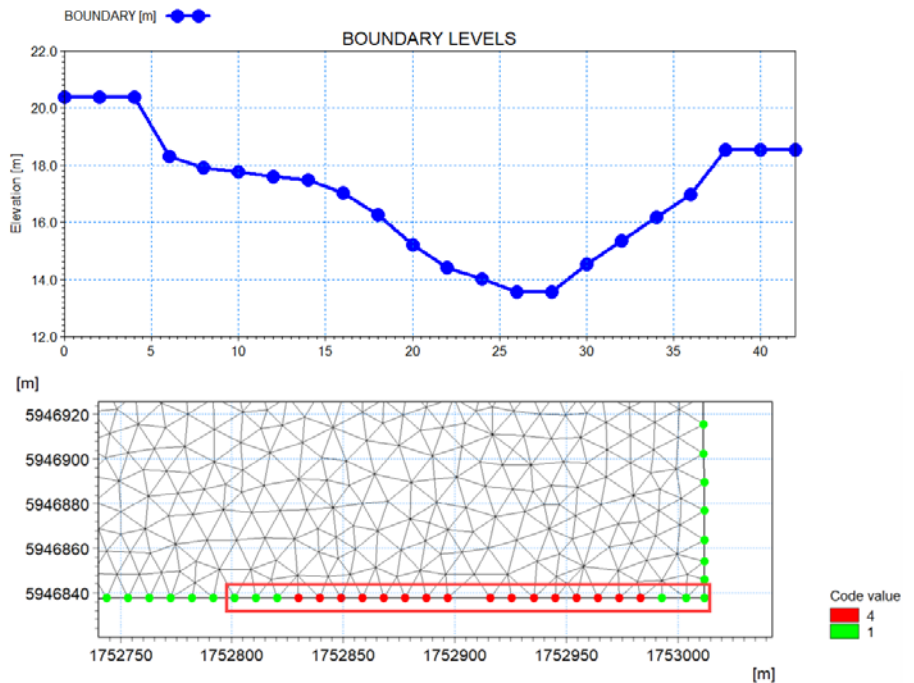
Figure 4.10   Illustration of topographical cross section (top) along an e.g. mesh boundary (bottom).

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 specified.

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:

$M = \dfrac{1}{n}$ = 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
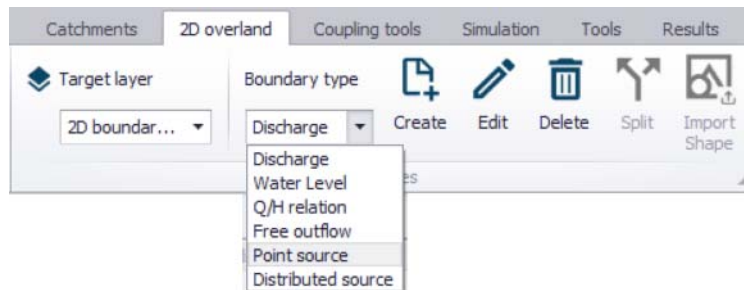
Figure 4.11   Adding Source boundaries on the Map using the Edit features toolbox on the 2D Overland menu ribbon

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 but-
             ton

Single distributed source

A distributed source is characterized by a polygon area in MIKE+. For this
type of source, the specified discharge value is distributed (i.e. weighed distri-
bution 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

If source velocity is applied (in the 'Temporal variation' tab), the specified
velocity values are not distributed but remain identical in all mesh/grid ele-
ments within the polygon.

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 conti-
nuity 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
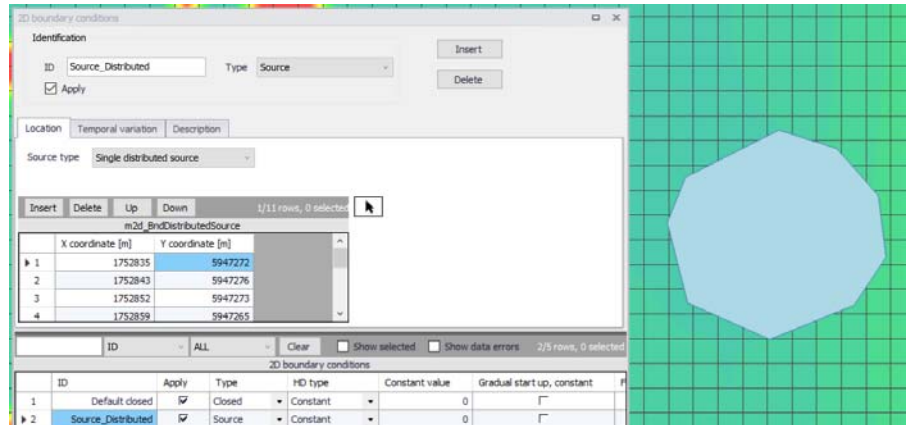point source segments on the Map.
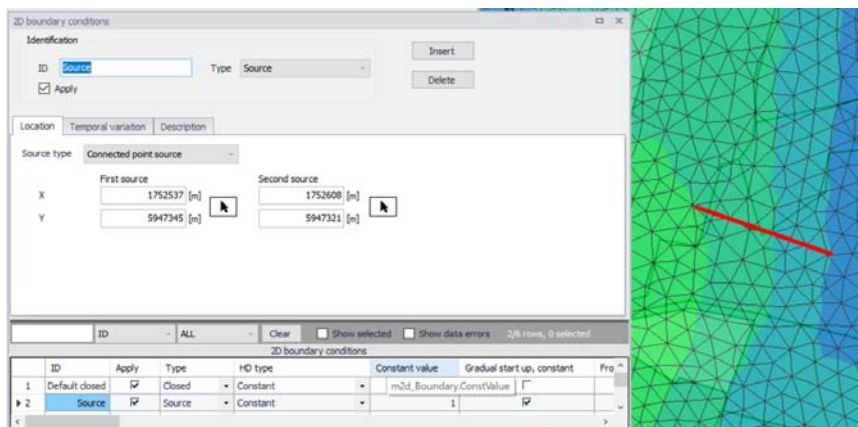
Figure 4.14   Define locations for the First Source and Second Source points for Connected Point Sources on the 2D Boundary Conditions editor and the Map

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 source is defined as:

- Constant
- Varying in time
- Q/h relation

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.

For the Q/h relation type (not available for distributed sources), specify a time series files (*.dfs0) containing the relationship between discharge and water level. The file must use a relative item axis describing the water level (where the unit must be meter) and must contain a dynamic item describing the discharge.
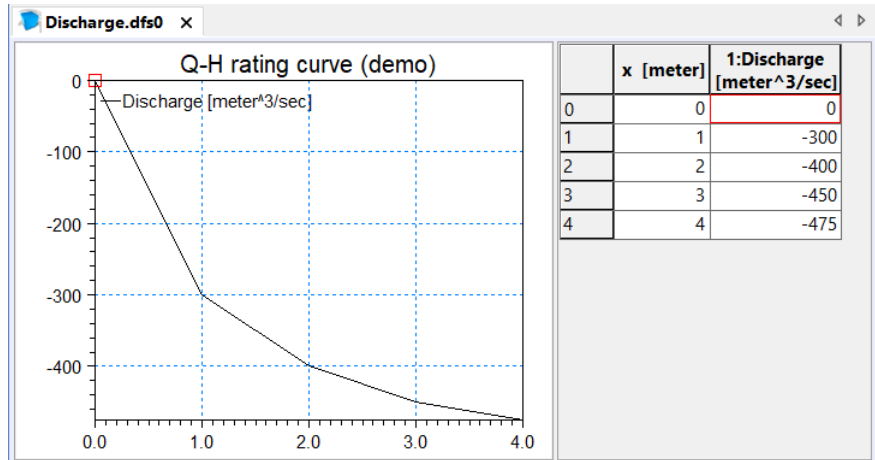
Figure 4.15   dfs0 file containing a Q/h relation

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 (for a Q/h relation type, these velocity components must be specified as a time series, using a regular time axis).

See the section on Source Types *(p. 100)* for more details.
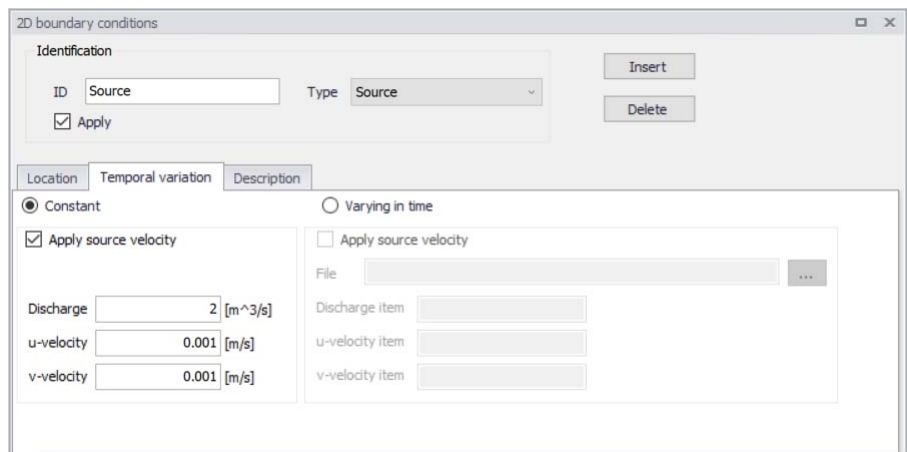
Figure 4.16   Temporal Variation tab for Source 2D boundary conditions

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 4.17).
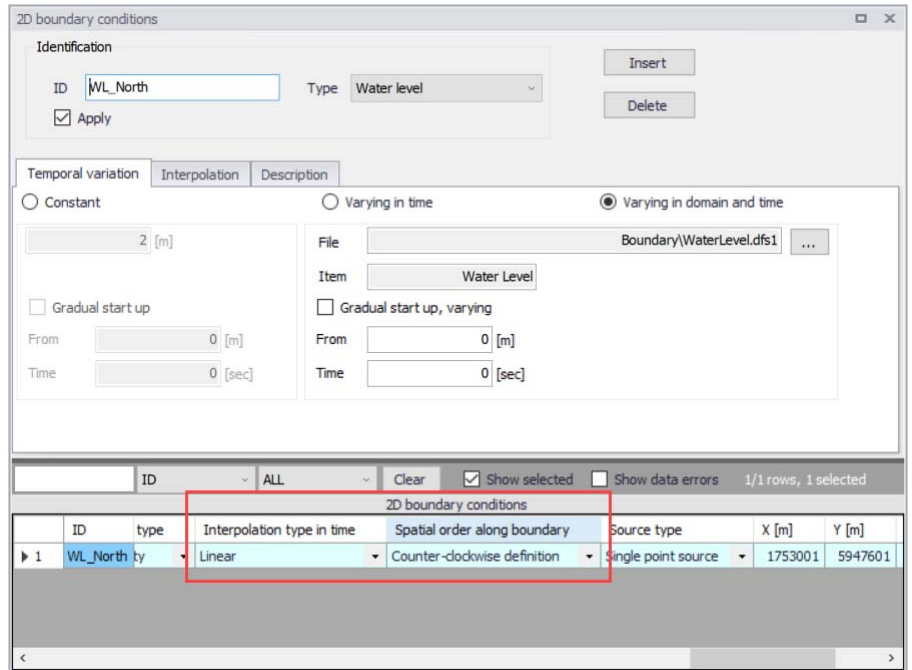
Figure 4.17   The 'Interpolation type in time' and 'Spatial order along boundary' attributes in the 2D boundary conditions overview table

4.2   2D WQ Boundary Conditions

MIKE+ 2D Overland offers options for simulating transport of materials (e.g. pollutants) via the 'Transport (AD, SWQ)' module.

At present, 2D WQ computations are not available for models of type 'SWMM5 collection system and overland flows', and so options for defining 2D WQ boundary conditions are not offered for these types of models.

Options for setting-up 2D WQ boundary conditions (Figure 4.18) are made available when both the 'Hydrodynamic (HD)' and 'Transport (AD, SWQ)' 2D Overland modules are activated for models of type 'River, collection system and overland flows.'

Figure 4.18   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. 85)* 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 4.19   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 103).

- 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 4.20). 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 4.20   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 106).

- 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 106).

  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 4.21).

Figure 4.21   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 4.22).
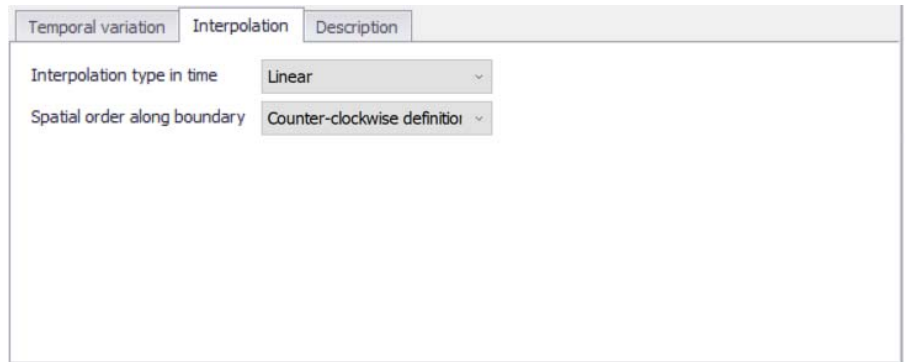
Figure 4.22   The Interpolation tab page on the 2D WQ Boundaries editor

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.
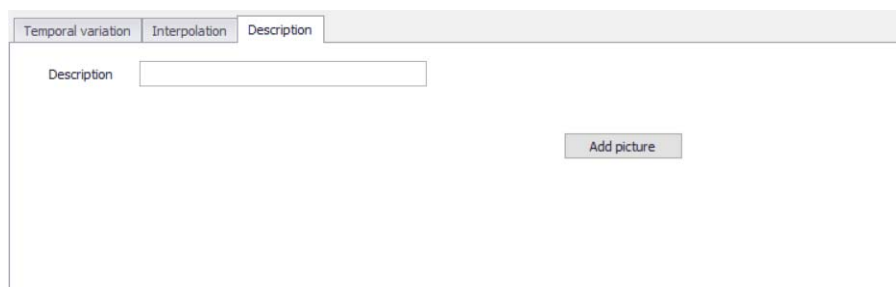
Figure 4.23   The Description tab page in the 2D WQ Boundaries editor

4.3   2D Precipitation and Evaporation

MIKE+ 2D Overland offers options for specifying ambient conditions for pre-
cipitation 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 impor-
tant.

These options are available on the 2D Precipitation and Evaporation editor
when the 2D Overland Hydrodynamic (HD) module is active (Figure 4.24).
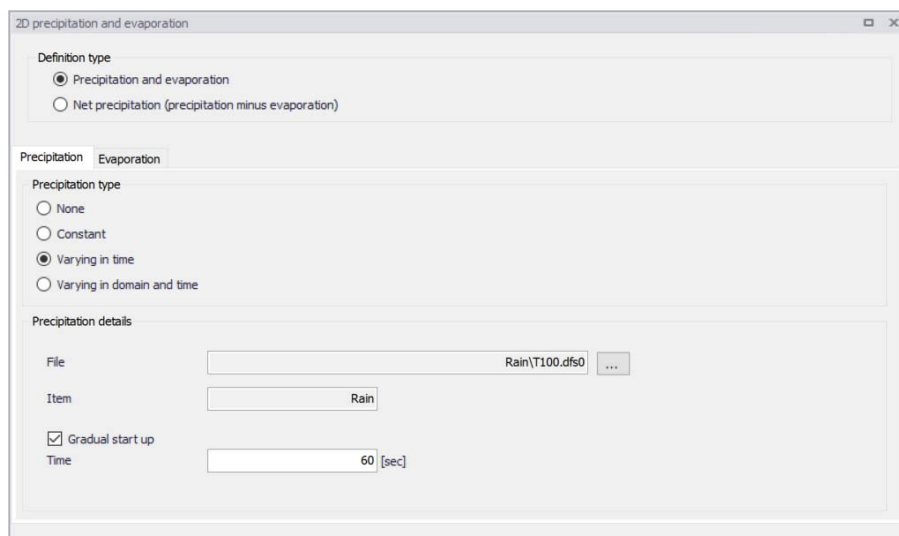
Figure 4.24   The 2D Precipitation and Evaporation editor

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 pre-
  cipitation and evaporation conditions may be specified. Defined precipi-
  tation 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 4.25   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 4.26).

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 4.27). 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.
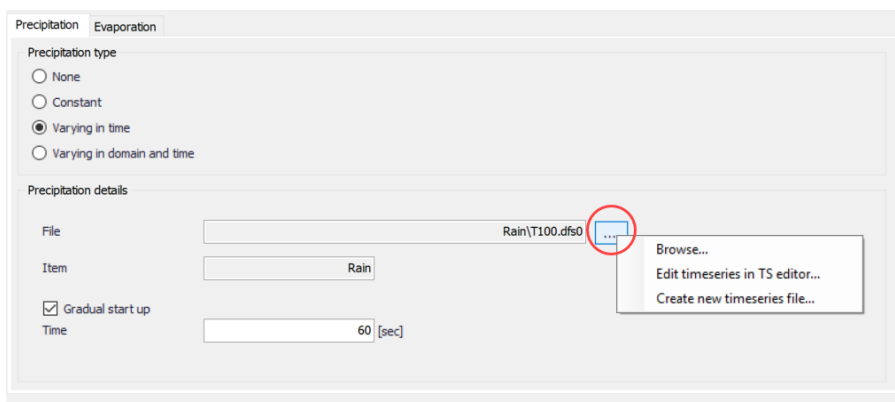
Figure 4.26   Options offered via the ellipsis button in relation to specification of pre-cipitation time series data file

Figure 4.27   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 4.28).

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 4.29). 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 4.28   2D grid or mesh files containing time- and domain-varying precipitation rates data must be specified
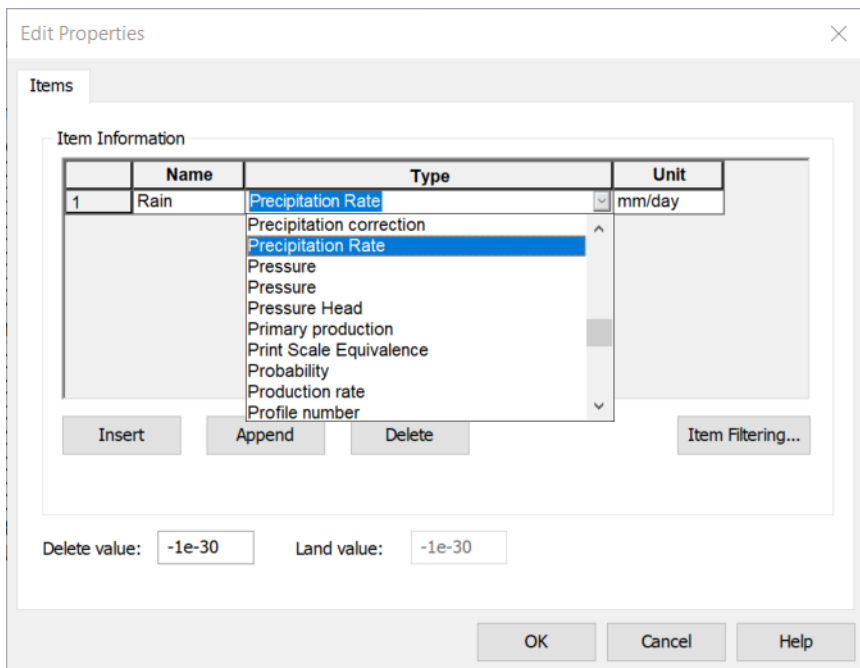
Figure 4.29   Edit the grid item type in the Grid Series editor via the Edit Properties dialog (Edit| Items).

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 4.30).

Figure 4.30   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 4.31   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 4.32).

Figure 4.32   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 4.33).
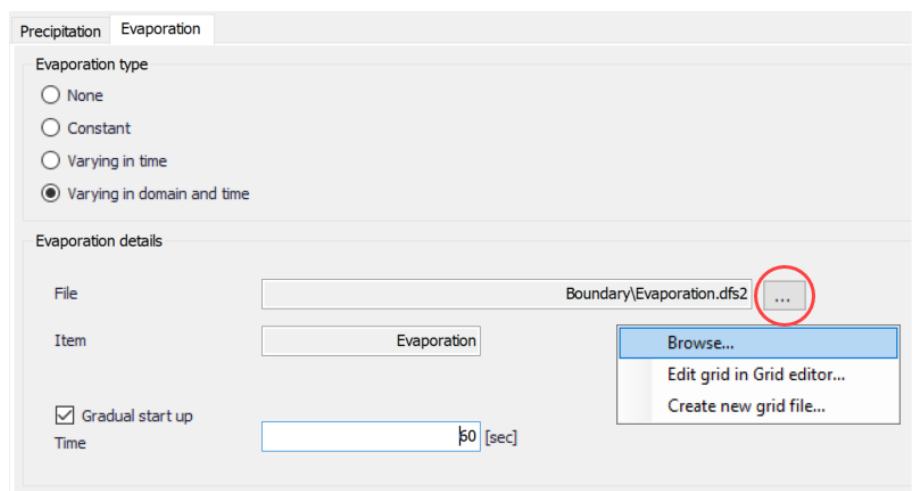
Figure 4.33   2D grid or mesh files containing time- and domain-varying evaporation rates data must be specified

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.
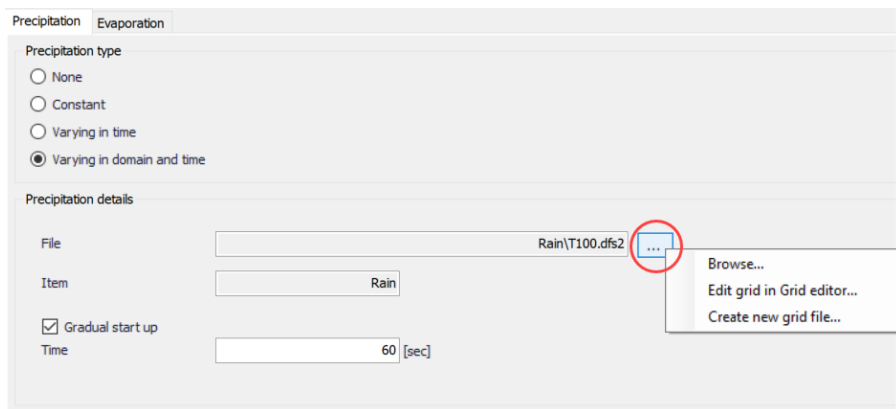
Gradual start up

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+ 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+ 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. 108)*) 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+.

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 4.34) 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 4.34   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 concentra-
tions (in component unit) (Figure 4.35).

Figure 4.35   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+ 2D AD Precipitation layer
- Background layer

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

MIKE+ 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 4.36). 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.
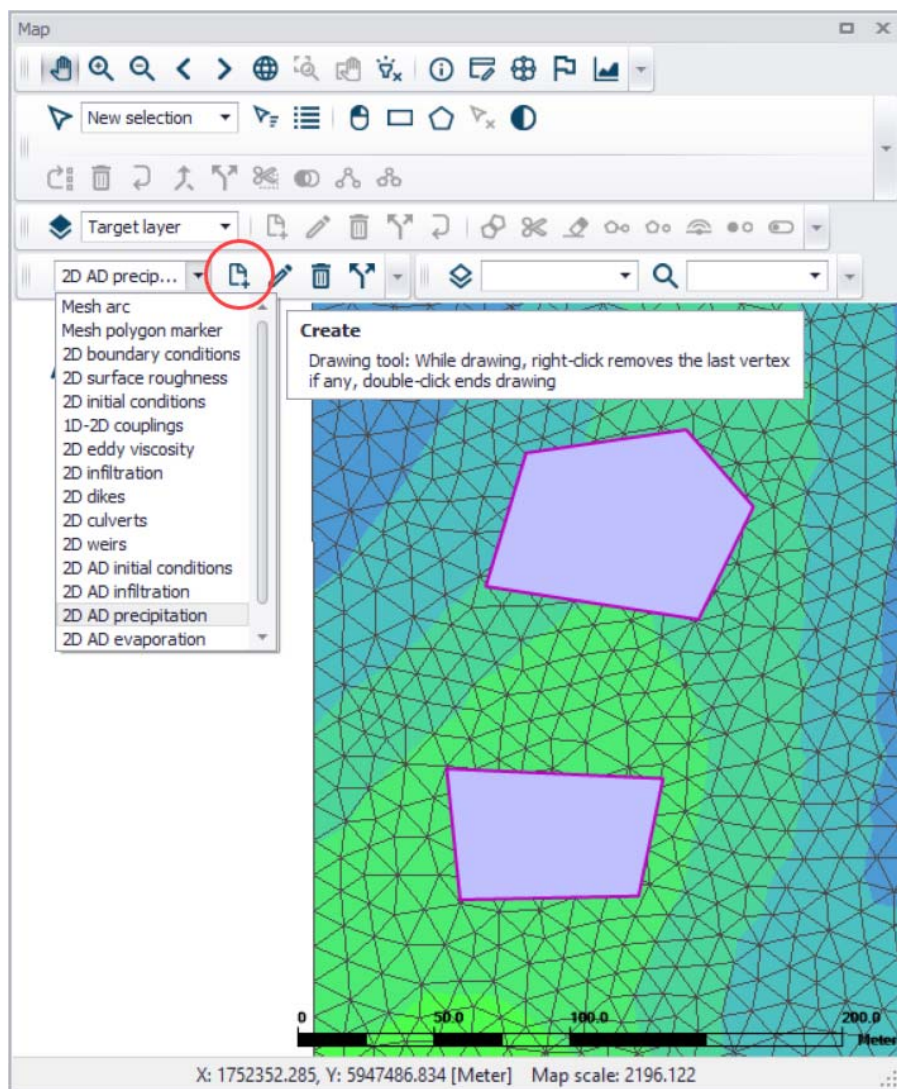
Figure 4.36   Draw 2D AD Precipitation layer polygons on the Map using the Create tool

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 4.37). 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.
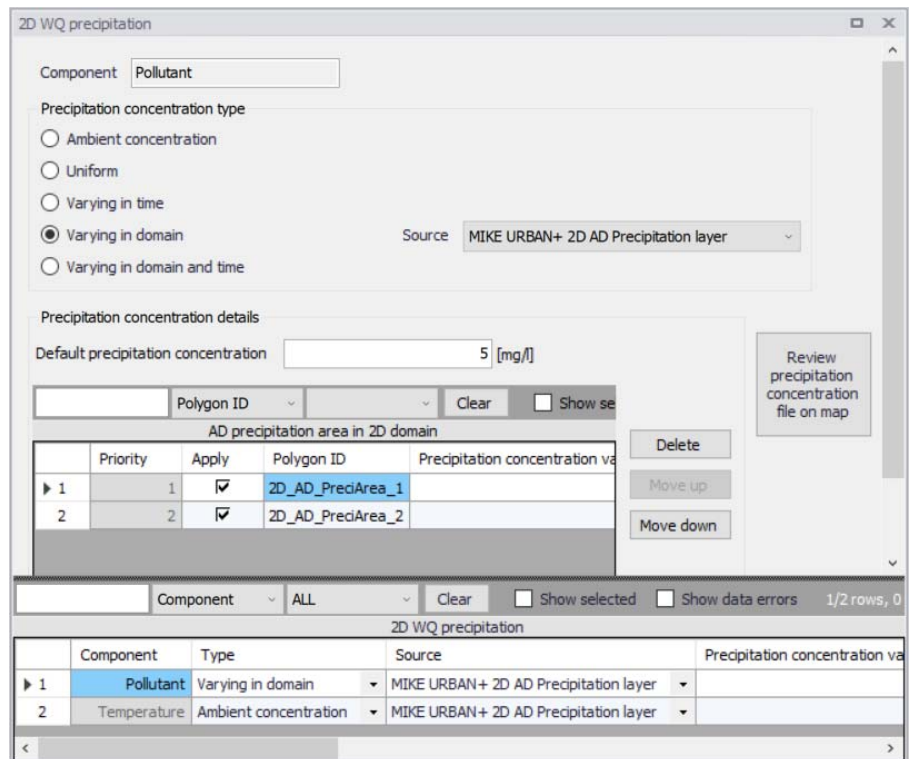
Figure 4.37   2D AD Precipitation layer polygons on the Map are listed in a table on the editor. Specify the related WQ component concentration for each polygon feature on the table.

Use the 'Review precipitation concentration file on map' button to generate a 2D file from the precipitation concentration configuration and view the gener-

ated 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 4.38   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 4.39 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+.

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. 117)*).

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 4.40).

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 4.40   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+ 2D AD Evaporation layer
- Background layer

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

MIKE+ 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 4.41). New records cannot be added from the tabular view. They are only added when drawing a new polygon on the map.

Figure 4.41   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 view-ing/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+ 2D Overland module.

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

- None
- Varying in Domain: Net infiltration rate varying spatially over the domain.
- Varying in Domain and Flow Dependent: Net infiltration rate varying spa-tially over the domain, calculated as a function of the water depth and the land cover.
- Varying in Domain and Time: Net infiltration rates that vary both spatially and temporally.
- Varying in Domain Using Capacity: Spatially-varying infiltration consider-ing the capacity of the infiltration zone. Uses simplified infiltration model that calculates net infiltration using specified information on surface zone layer properties.
- Varying in Domain and Flow Dependent, Using Capacity: Varying infiltra-tion considering the capacity of the infiltration zone, defined as a function of the water depth and the land cover. 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.
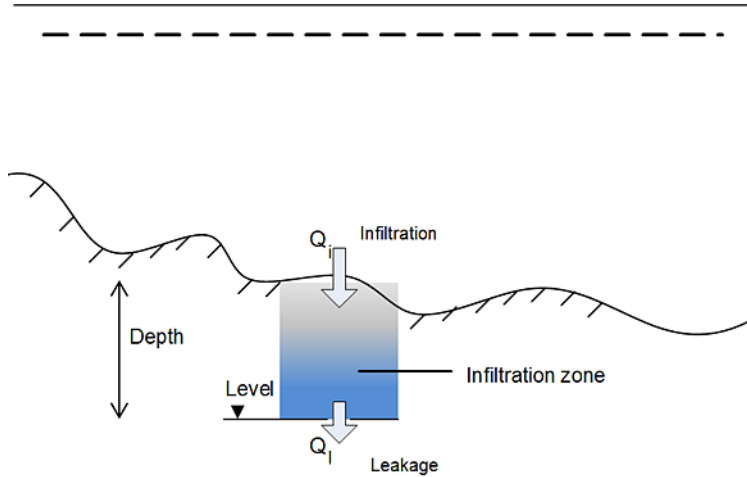
Figure 4.42   Definition of infiltration with storage and leakage: In the case of net infiltration, $Q_{infiltration} = Q_{leakage}$, the model assumes an infinite storage volume in the infiltration zone.

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+ 2D infiltration layer
- Background layer of individual polygons
- 2D file
- Background layer of land Cover
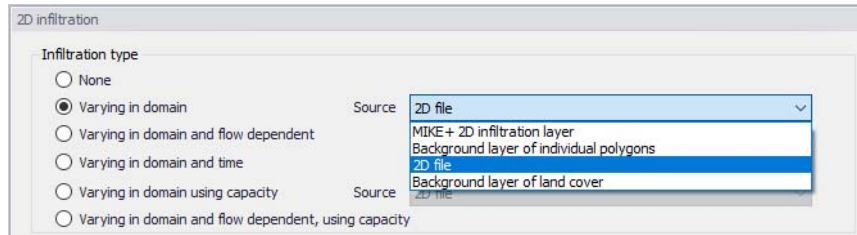
Figure 4.43   Options for specifying spatially-varying infiltration for the 2D overland model under the Information Type section

MIKE+ 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 4.44), or the Flooding Layer Editing Tools toolbar on the Map (Figure 4.45).
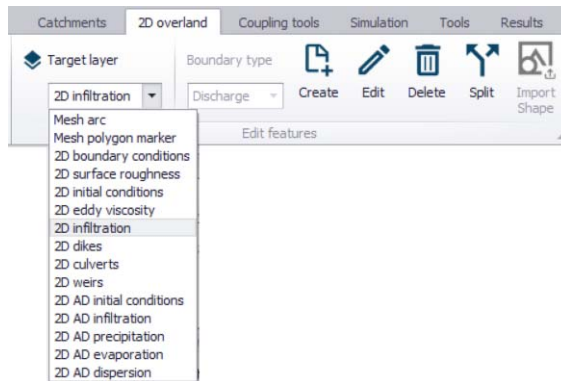
Figure 4.44   Edit 2D model layers on the Map using the Edit Features toolbar on the 2D Overland menu ribbon.
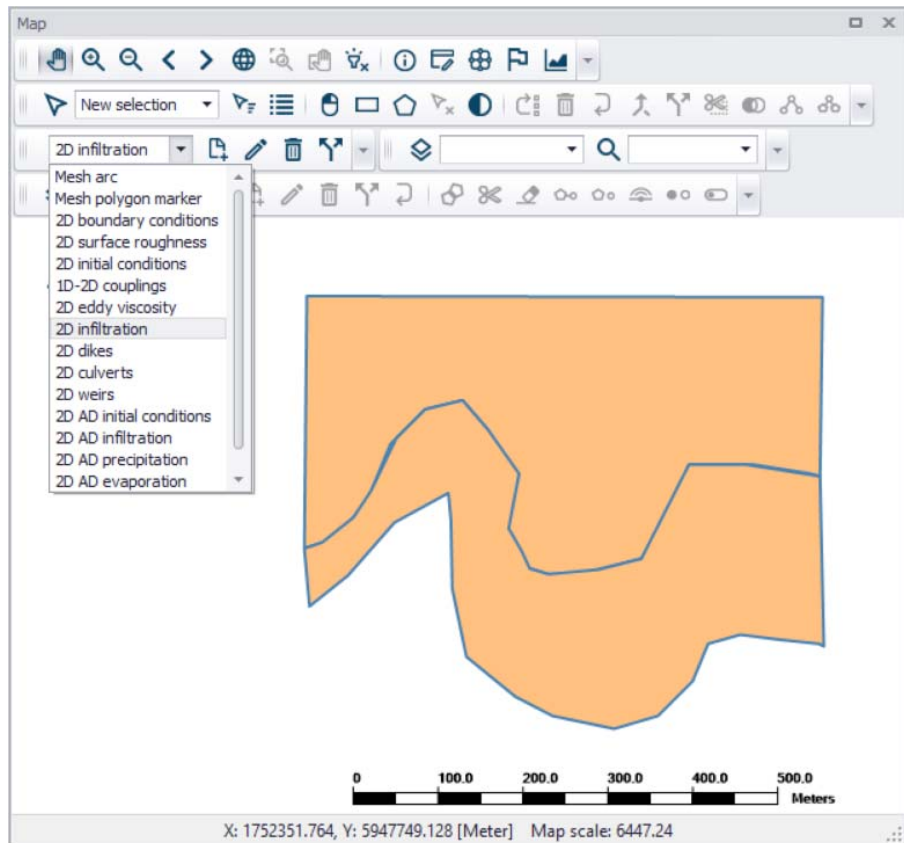
Figure 4.45   Create 2D infiltration polygon features on the Map using tools from the Flooding Layer Editing Tools toolbar

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 4.46). 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 4.46   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 of individual polygons

When the space-varying infiltration input source is from a 'Background Layer of individual polygons', 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.
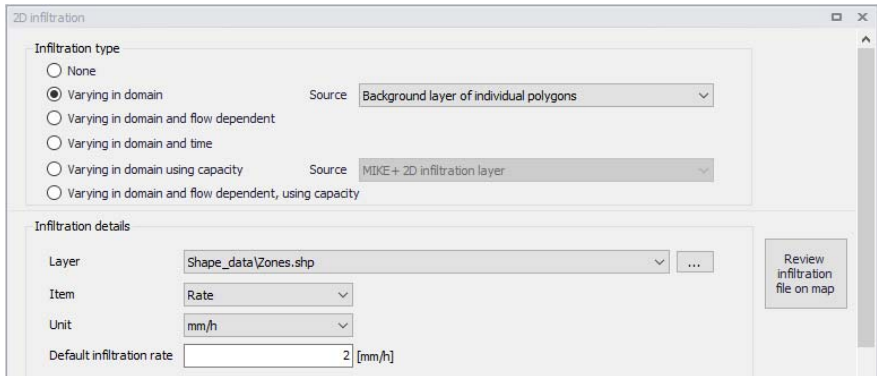
Figure 4.47   The program detects and offers valid Background Layers on the Map as sources of infiltration data. Use the 'Review infiltration file on map' button to view the configured 2D infiltration data layer on the Map.

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 4.48).
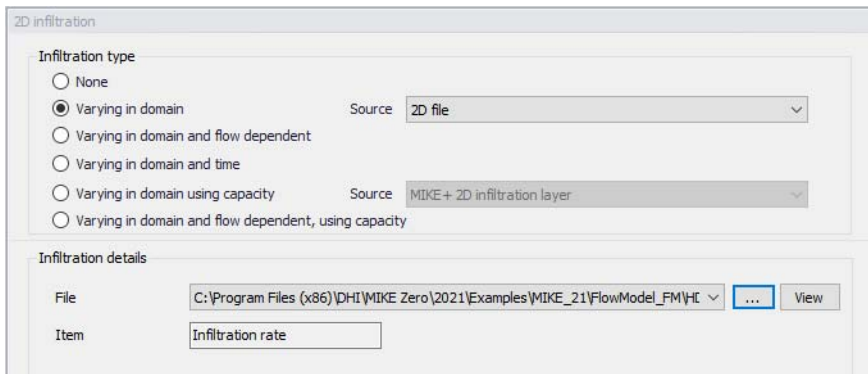
Figure 4.48   Point to a 2D infiltration file under the Infiltration details section on the editor

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 4.49). 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.
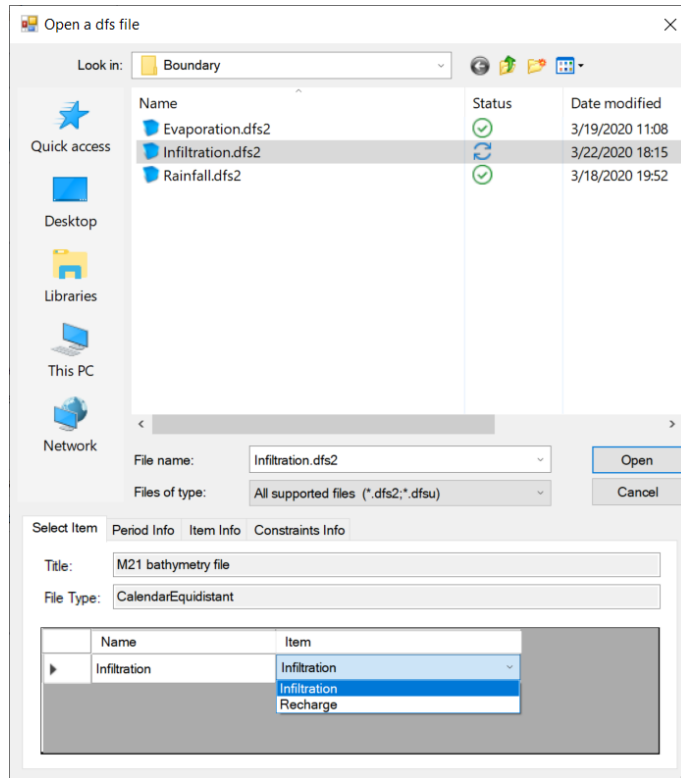
Figure 4.49   Browse to the valid 2D file via the 'Open a dfs file' dialog.

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.

Background layer of land Cover

The 'Background layer of land cover' option allows defining the net infiltration rate as a function of land cover, i.e. per land cover type. A polygon layer must be selected to define the land cover, in which the polygons are grouped by category (land cover type). Three types of file format can be used for defining the land cover types:

- Shape file
- DFS2 or DFSU file
- XYZ file

For XYZ files and shape files, the land cover zones are defined as the closed region within a number of polygons. For dfs files, the zones are defined by a map identifying the location of the different zones. Each land cover type is identified by an integer number larger than zero. If an element from the 2D domain belongs to more than one polygon in the XYZ file or the shape file, the information from the last polygon read from the file is applied. If there are overlapping zones with the road zones (from the 2D infrastructures editor), the priority is first given to the road zones.

For dfs files, the value should be zero in areas with no local infiltration zone in a dfs file.

For XYZ files, the data in the file must be formatted as ASCII text in five columns with the two first columns giving the x-and y-coordinates of the points. The third column represents connectivity. The connectivity column is used to define arcs. All points along an arc - except the last point - shall have a connectivity value of 1 and the last point shall have a connectivity value of 0. The fourth column contains the z-value at the point, which is unused for land cover layers. Finally, the fifth column contains the zone number, corresponding to the land cover type.

In the 'Land cover layer' section, the following needs to be specified:

- Layer: The source layer defining the land cover zones
- Item: For a shape file or dfs file, this is the attribute or item in the file containing the integer number, used to identify the land cover type each polygon belongs to.
- Map projection: For a XYZ file, this is the map projection in which the XY coordinates in the file are defined.

In the 'Infiltration rate' table, the net infiltration rate for each of the zones. The list of zones is made of the list of land cover types plus the list of road zones eventually selected in the 2D infrastructures editor. The land cover ID for these zones is editable. A default rate must also be supplied, which is applied in areas not covered by any of the previous zones.
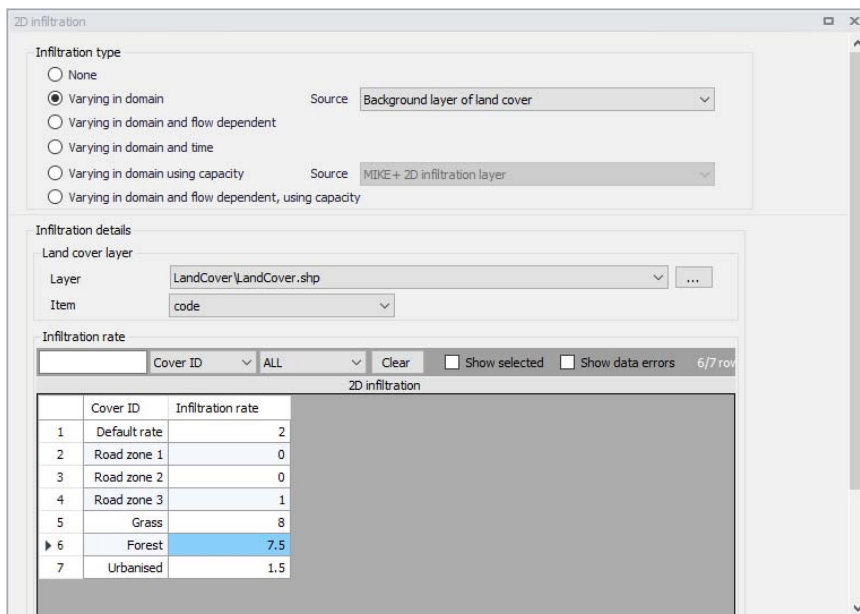
Figure 4.50   Defining infiltration rates for land cover types.

4.5.4   Varying in Domain and Flow Dependent

The 'Varying in domain and flow dependent' option allows defining the net infiltration rate as a function of the water depth on the surface, and also as a function of land cover, i.e. per land cover type. A polygon layer must be selected to define the land cover, in which the polygons are grouped by category (land cover type). A curve defining the relationship between the water depth and the infiltration rate must then be selected for each land cover type. Three types of file format can be used for defining the land cover types:

- Shape file

- DFS2 or DFSU file

- XYZ file

For XYZ files and shape files, the land cover zones are defined as the closed region within a number of polygons. For dfs files, the zones are defined by a map identifying the location of the different zones. Each land cover type is identified by an integer number larger than zero. If an element from the 2D domain belongs to more than one polygon in the XYZ file or the shape file, the information from the last polygon read from the file is applied. If there are overlapping zones with the road zones (from the 2D infrastructures editor), the priority is first given to the road zones.

For dfs files, the value should be zero in areas with no local infiltration zone in a dfs file.

For XYZ files, the data in the file must be formatted as ASCII text in five columns with the two first columns giving the x-and y-coordinates of the points. The third column represents connectivity. The connectivity column is used to define arcs. All points along an arc - except the last point - shall have a connectivity value of 1 and the last point shall have a connectivity value of 0. The fourth column contains the z-value at the point, which is unused for land cover layers. Finally, the fifth column contains the zone number, corresponding to the land cover type.

In the 'Land cover layer' section, the following needs to be specified:

- Layer: The source layer defining the land cover zones

- Item: For a shape file or dfs file, this is the attribute or item in the file containing the integer number, used to identify the land cover type each polygon belongs to.

- Map projection: For a XYZ file, this is the map projection in which the XY coordinates in the file are defined.

In the 'Infiltration rate' table, the Infiltration Curve defining the relationship between the water depth and the net infiltration rate must be selected for each zone. The list of zones is made of the list of land cover types plus the list of road zones eventually selected in the 2D infrastructures editor. The land cover ID for these zones is editable. A default curve must also be supplied, which is applied in areas not covered by any of the previous zones.

Valid curves are curves with the type 'Depth dependent infiltration'. The 'Create curves' button can be used to create a curve (in the Curves and Relations editor) with some default values for each of the zones.
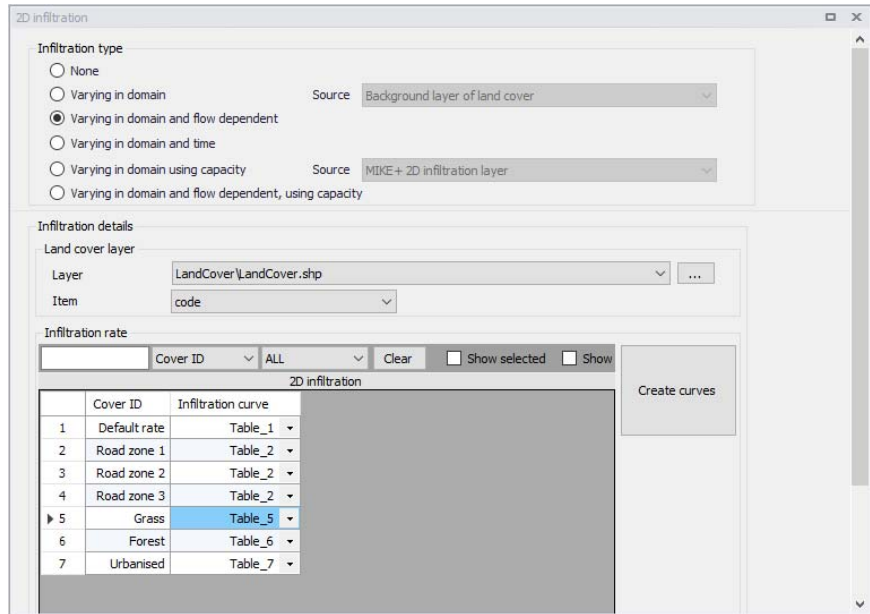
Figure 4.51 Defining depth-dependent infiltration curves for land cover types.

4.5.5 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 4.52).
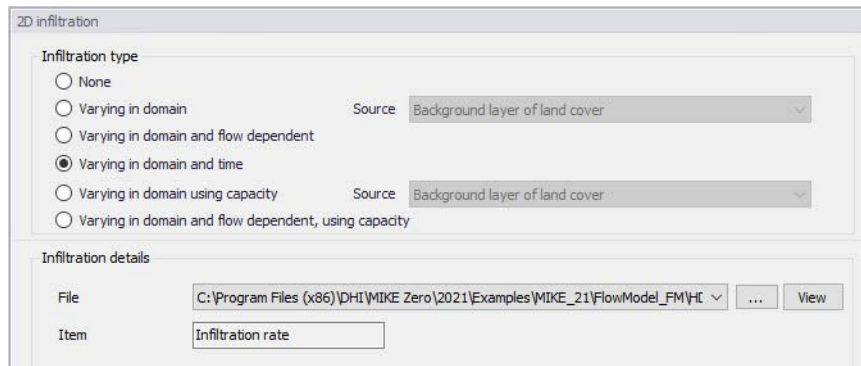
Figure 4.52 Browse to a time-varying 2D file for input infiltration data that varies in domain and time

- 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.

4.5.6   Varying in Domain Using Capacity
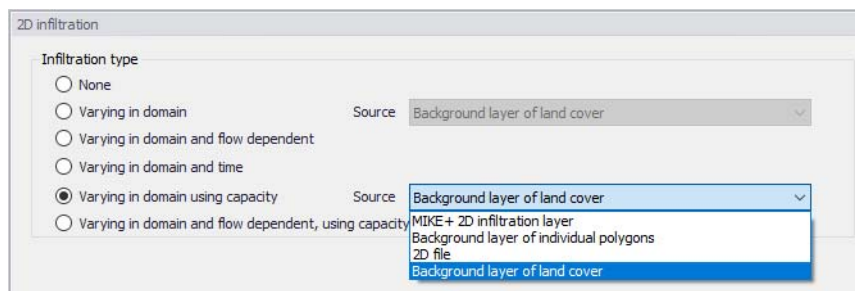
Data for infiltration that is 'Varying in domain using capacity' may be defined in the 2D model with the following sources (Figure 4.53):

- MIKE+ 2D infiltration layer: 2D infiltration polygons defined on the Map in MIKE+.

- Background layer of individual polygons: Pre-loaded polygon background layer on the Map.

- 2D file: Existing infiltration 2D file (*.DFS2 or *.DFSU types)

- Background layer of land cover: Pre-loaded polygon background layer defining the type of land cover

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

MIKE+ 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 4.44), or the Flooding Layer Editing Tools toolbar on the Map (Figure 4.45) to create 2D infiltration polygons. Records for each polygon feature are listed in a table on the 2D Infiltration editor (Figure 4.54).

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   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 of individual polygons

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 4.55   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.
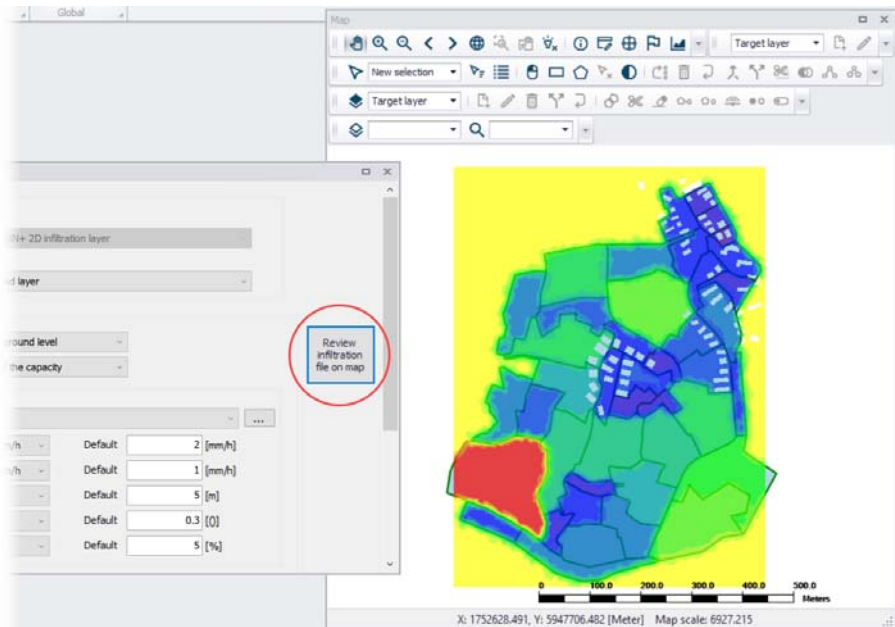
Figure 4.56   2D infiltration data layer (*.DFSU) generated from configuration using the 'Review infiltration file on map' functionality

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 4.57   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 4.49).

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[()])

Background layer of land cover

The 'Background layer of land cover' option allows defining the infiltration capacity as a function of land cover, i.e. per land cover type. A polygon layer must be selected to define the land cover, in which the polygons are grouped by category (land cover type). Three types of file format can be used for defining the land cover types:

- Shape file

- DFS2 or DFSU file

- XYZ file

For XYZ files and shape files, the land cover zones are defined as the closed region within a number of polygons. For dfs files, the zones are defined by a map identifying the location of the different zones. Each land cover type is identified by an integer number larger than zero. If an element from the 2D domain belongs to more than one polygon in the XYZ file or the shape file, the information from the last polygon read from the file is applied. If there are overlapping zones with the road zones (from the 2D infrastructures editor), the priority is first given to the road zones.

For dfs files, the value should be zero in areas with no local infiltration zone in a dfs file.

For XYZ files, the data in the file must be formatted as ASCII text in five columns with the two first columns giving the x-and y-coordinates of the points. The third column represents connectivity. The connectivity column is used to define arcs. All points along an arc - except the last point - shall have a connectivity value of 1 and the last point shall have a connectivity value of 0. The fourth column contains the z-value at the point, which is unused for land cover layers. Finally, the fifth column contains the zone number, corresponding to the land cover type.

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

Under the 'Land cover layer' section the following needs to be specified:

- Layer: The source layer defining the land cover zones

- Item: For a shape file or dfs file, this is the attribute or item in the file containing the integer number, used to identify the land cover type each polygon belongs to.

- Map projection: For a XYZ file, this is the map projection in which the XY coordinates in the file are defined.

In the 'Infiltration rate' table, the infiltration capacity is specified for each of the zones. The list of zones is made of the list of land cover types plus the list of road zones eventually selected in the 2D infrastructures editor. The land cover ID for these zones is editable. A default rate must also be supplied, which is applied in areas not covered by any of the previous zones. For each zone, the following data need to be provided:

- 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 dropdown 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[()]).

Figure 4.58   Defining infiltration capacity for land cover zones

4.5.7   Varying in Domain and Flow Dependent, Using Capacity

The 'Varying in domain and flow dependent, using capacity' option allows defining the infiltration capacity as a function of the water depth on the surface, and also as a function of land cover, i.e. per land cover type. A polygon layer must be selected to define the land cover, in which the polygons are grouped by category (land cover type). A curve defining the relationship between the water depth and the infiltration rate must then be selected for each land cover type. Three types of file format can be used for defining the land cover types:

- Shape file
- DFS2 or DFSU file
- XYZ file

For XYZ files and shape files, the land cover zones are defined as the closed region within a number of polygons. For dfs files, the zones are defined by a map identifying the location of the different zones. Each land cover type is identified by an integer number larger than zero. If an element from the 2D domain belongs to more than one polygon in the XYZ file or the shape file, the information from the last polygon read from the file is applied. If there are overlapping zones with the road zones (from the 2D infrastructures editor), the priority is first given to the road zones.

For dfs files, the value should be zero in areas with no local infiltration zone in a dfs file.

For XYZ files, the data in the file must be formatted as ASCII text in five columns with the two first columns giving the x-and y-coordinates of the points. The third column represents connectivity. The connectivity column is used to define arcs. All points along an arc - except the last point - shall have a connectivity value of 1 and the last point shall have a connectivity value of 0. The fourth column contains the z-value at the point, which is unused for land cover layers. Finally, the fifth column contains the zone number, corresponding to the land cover type.

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

In the 'Land cover layer' section, the following needs to be specified:

- Layer: The source layer defining the land cover zones

- Item: For a shape file or dfs file, this is the attribute or item in the file containing the integer number, used to identify the land cover type each polygon belongs to.

- Map projection: For a XYZ file, this is the map projection in which the XY coordinates in the file are defined.

In the 'Infiltration rate' table, the infiltration capacity is specified for each of the zones. The list of zones is made of the list of land cover types plus the list of road zones eventually selected in the 2D infrastructures editor. The land cover ID for these zones is editable. A default curve must also be supplied, which is applied in areas not covered by any of the previous zones. For each zone, the following data need to be provided:

- Infiltration curve: The curve defining the relationship between the water depth and the infiltration rate defined in the Curves and Relations editor.

- 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 dropdown 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[()]).

Valid curves are curves with the type 'Depth dependent infiltration'. The 'Create curves' button can be used to create a curve with some default values for each of the zones.

4.6   2D WQ Infiltration

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

These options are available in MIKE+ 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 4.59) to configure water quality properties for Infiltration boundaries in the model.

Figure 4.59   The 2D WQ Infiltration editor

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

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+ 2D AD Infiltration Layer
- Background Layer

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

MIKE+ 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 4.60). 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.
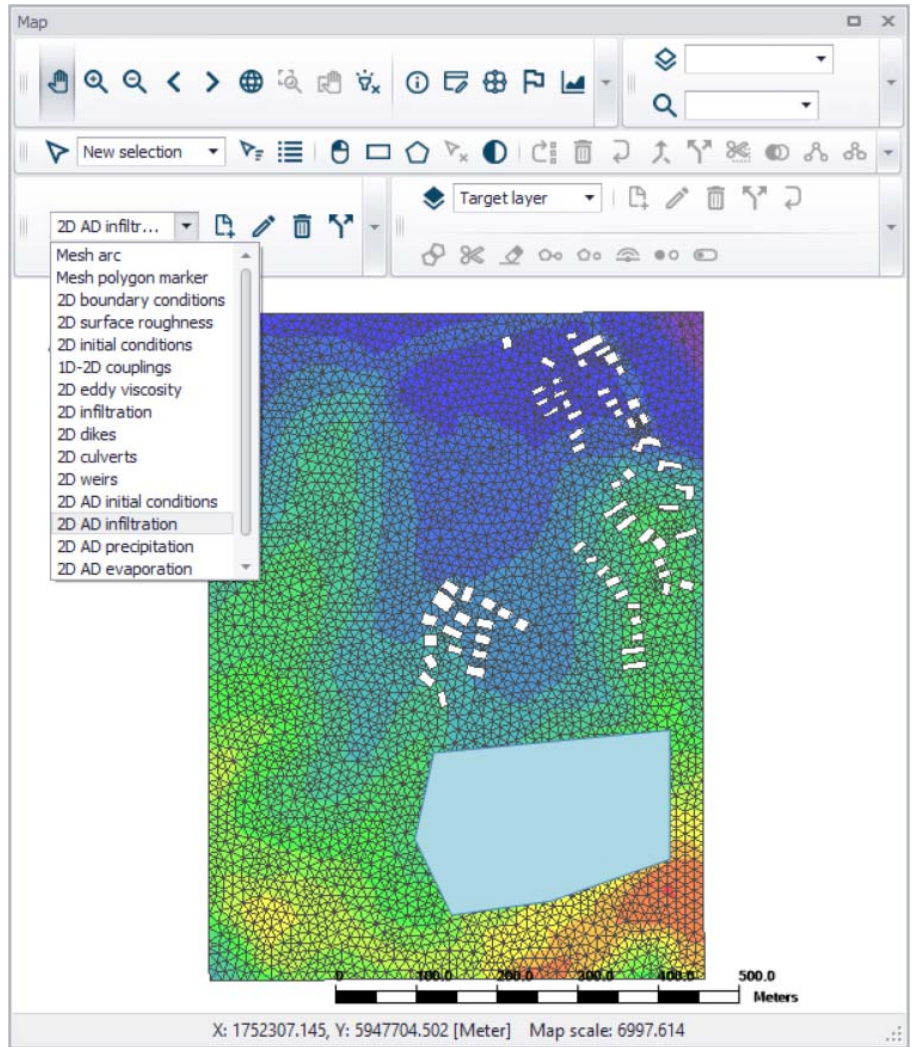
Figure 4.60   Define '2D AD infiltration' layer polygons on the Map

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 4.61   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 compo-
nent) 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 concentra-
  tion 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 4.62   Specify domain-varying infiltration concentration parameters based on Background layer data

The 'Infiltration concentration details' section on the 2D WQ Evaporation editor displays (Figure 4.62):

- 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 4.63). 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 4.63   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+, the benefits of 2D and 1D modelling can be utilized in an integrated model where complex networks and channels represented in 1D are coupled to an overland surface representation in 2D. Models of collection systems and river networks may be coupled to models of the overland surface.

SWMM5 collection system models may also now be coupled to 2D overland models in MIKE+ to build integrated 1D-2D models.

5.1 River Model

This section appears in the Setup tree if coupling to an external MIKE HYDRO river model has been enabled for the MIKE+ project via the Model Type editor.

When 'Coupling to MIKE HYDRO River' is active in the Model Type editor, the desired MIKE HYDRO model setup file must be defined in the River Model editor under the Model Couplings section in the Setup tree.

The ellipsis button (...) allows for browsing to the required MIKE HYDRO model file. The coupled file should preferably be saved with the latest version of MIKE HYDRO River, which is MIKE HYDRO River 2023.

Once selected, the main features from the MIKE HYDRO River model are displayed on the Map in MIKE+. However, they cannot be edited from within MIKE+ but must be accessed through MIKE HYDRO.

5.2 2D/3D Overland Model

When 'Coupling to MIKE 21 or MIKE 3 Flow Model FM' is active in the Model Type editor, the desired 2D or 3D overland model setup file must be defined in the 2D/3D Overland Model editor under the Model Couplings section in the Setup tree.

The ellipsis button (...) allows for browsing to the required MIKE 21 FM or MIKE 3 FM model file.

Once selected, the 2D domain file from the 2D or 3D model is displayed on the Map in MIKE+. However, it cannot be edited from within MIKE+ but should be accessed through MIKE 21 FM or MIKE 3 FM, as appropriate.

5.3 1D-2D Couplings

Couplings between network and overland models are defined in the 1D-2D Couplings editor.

5.3.1 Coupling Types

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

- Manhole
- Basin
- Outlet
- Soakaway
- Pump
- Weir
- Natural channel
- Junction (SWMM5)
- Storage unit (SWMM5)
- Flow divider (SWMM5)
- Outfall (SWMM5)
- River bank
- River end

Note that currently-available options for coupling SWMM5 to 2D overland models are primarily at node elements (e.g. junctions, outfalls), and coupling along open SWMM5 conduits banks are as yet absent.

Coupling definitions may be individually specified in the 1D-2D Couplings editor or batch-generated using coupling tools available in MIKE+.

5.3.2 Coupling Tools

In the Coupling Tools menu ribbon (activated when the 2D Overland Hydrodynamic module is turned on), tools are available for defining and editing 1D-2D couplings.
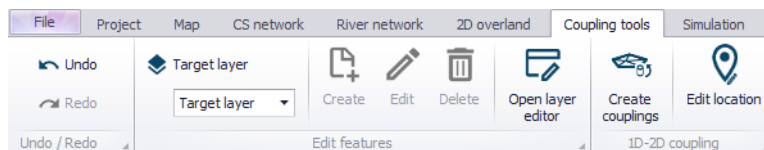
Figure 5.1   The Coupling Tools menu ribbon

Create Couplings

The Create Couplings tool is used to configure couplings between 1D and 2D model elements. New coupling definitions are created using this tool, with new rows with unique IDs automatically generated in the 1D-2D Couplings editor table.

Tip: If you intend to couple your model in several steps, you can dock the coupling configuration dialog within your workspace for easier access.
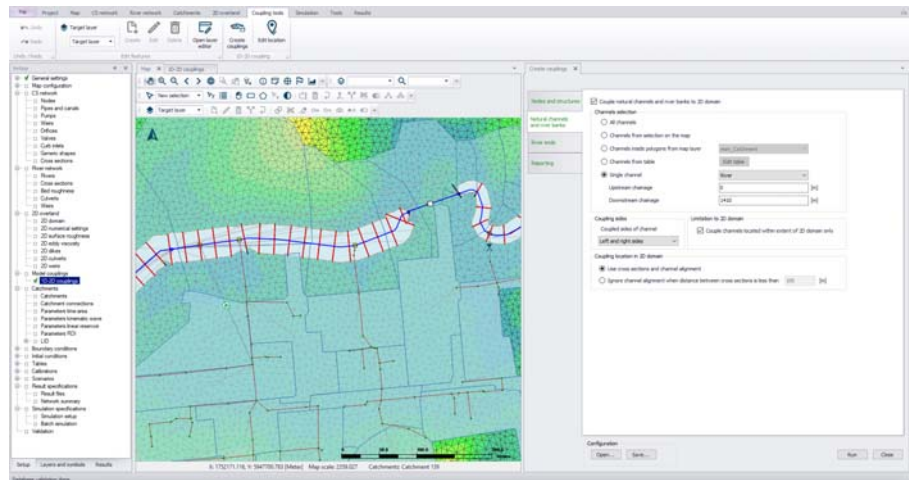
Figure 5.2   Docking the Create Couplings tool window in the workspace

Nodes and Structures

The first tab in the Create Couplings tool dialog enables coupling of nodes and structures to 2D models. Tick on the option 'Couple nodes and structures to 2D domain."

For 'Rivers, collection systems and overland flows' model types, the tool offers tab pages for coupling other model components to the 2D domain (e.g. Natural channels and river banks). If only couplings to item types other than nodes and structures are needed, leave this option inactive.

When working with 'SWMM5 collection system and overland flows' model types, only options for coupling nodes and structures to the 2D domain are available.
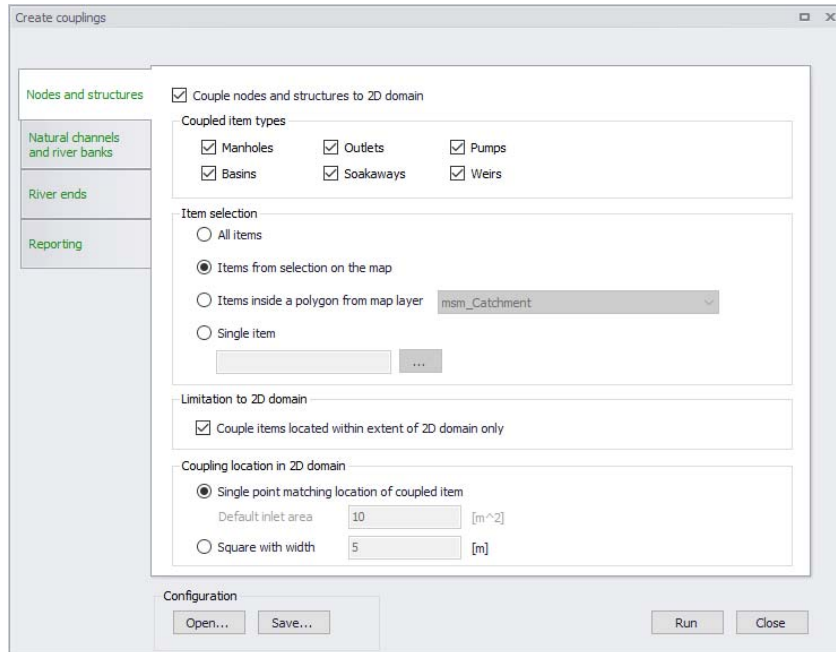
Figure 5.3   Couple nodes and structures using the Create Couplings tool

Coupled Item Types

Select the node or structure components to be coupled. The item types offered vary depending on the type of collection system model being used in the project (e.g. SWMM5).

Tip: Every time the Create Couplings tool is executed, coupling definitions are added to the 1D-2D Couplings editor table, so one may define couplings incrementally.

Item Selection

A number of options are available for defining which individual 1D items are to be coupled:

- All items: Creates coupling definitions 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 item editor overview table which will be selected on the Map. The currently active selection will be coupled.

- Items inside a polygon from map layer: Limit the coupling to items that are located within an area (e.g. within a catchment). This area defined by a polygon feature must be part of the model (e.g. msm_Catchment data layer) or loaded into the model as a background layer and selected.

- Single item: Only couple one node or structure. In this case, click on the ellipsis button (…) to specify the ID of the element to be coupled.

Limitation to 2D Domain

Tick on the 'Couple items located within extent of 2D domain only' option to ensure that only nodes and structures within the 2D domain are considered when coupling (recommended).

Coupling Location in 2D Domain

Define how couplings are to be spatially defined:

- 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 they are defined at the location of the upstream node (i.e. 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 dimensions around the node/structure location. In this case, more than one 2D cell/element may be coupled to a1D item, and the flow will be distributed among the cells when flowing from the 1D item. This feature is useful if one expects high flow exchanges that may cause computational instabilities.
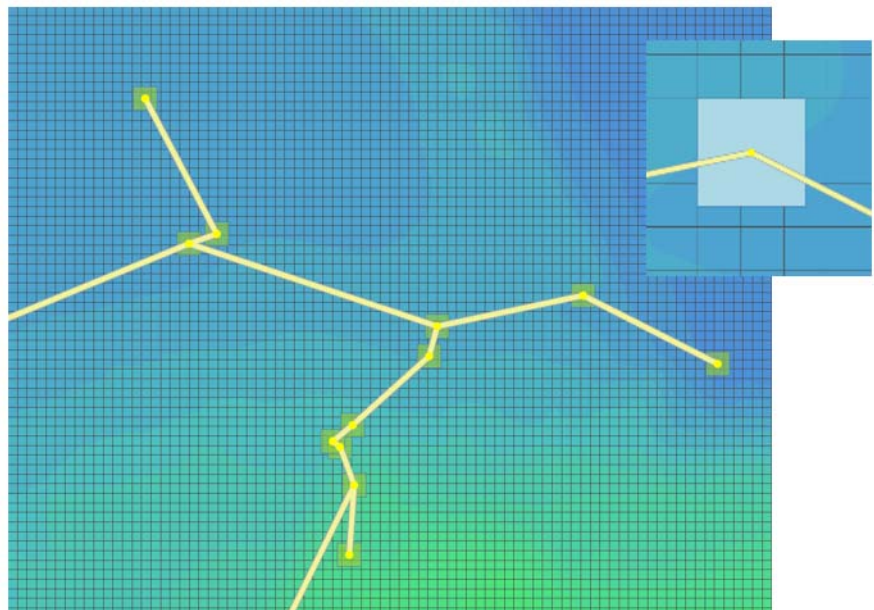
Figure 5.4   Coupling 2D cells by square area with defined width

A default inlet area is assigned to the created couplings. In 'Rivers, collection system and overland flows' mode, the default area for couplings to manholes is the area of the manhole computed using its diameter, and for couplings to basins or soakaways it is obtained from the node's geometry curve. In 'SWMM5 collection system and overland flows' mode:

- With the option 'Single point matching location of coupled item', the default inlet area of the inlet is manually specified in the 'Default inlet area' field.

- With the option 'Square with width', the default inlet area is computed using the specified square's width.

It is always possible to manually edit the inlet area of individual couplings afterwards, from the '1D-2D couplings' editor.

Natural Channels and River Banks

Another tab in the Create Couplings tool enables natural channels and river banks to be coupled to 2D domains. Tick on the 'Couple natural channels to 2D domain' to activate this option.

If only couplings to item types other than natural channels and rivers are needed, leave this option unticked.

Note that there are currently no options for coupling SWMM5 open conduits to 2D overland domains.
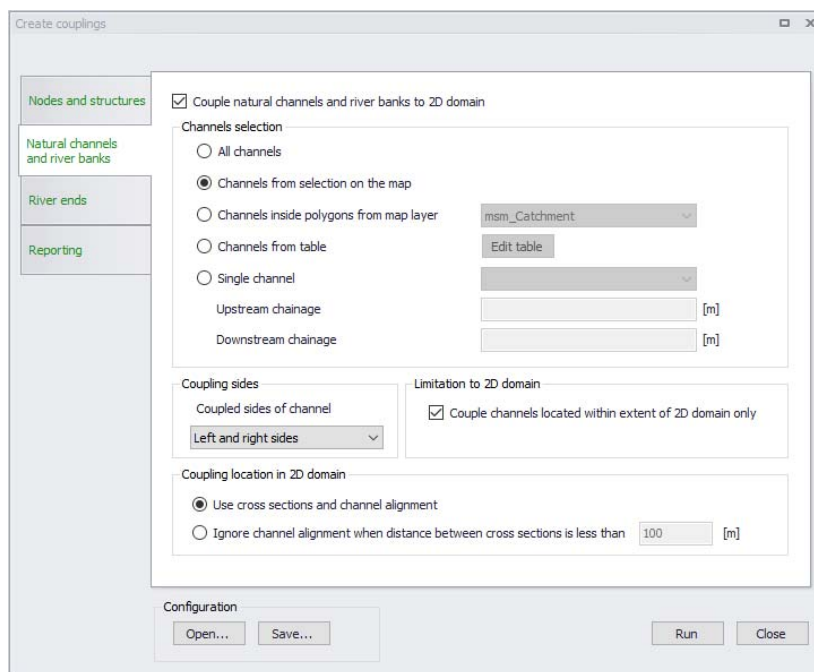
Figure 5.5   Couple natural channels and river banks using the Create Couplings tool

Channels Selection

A number of selection methods are available for defining channels to be coupled:

- All channels: Creates a coupling definition for every natural channel (specified in the CS network | Pipes and Canals editor) or river (specified in River network | Rivers or in MIKE HYDRO).

- Channels from selection on the map: Select channels via the Map or network feature overview tables. Selecting items in editor overview tables will select/highlight the corresponding items on the Map. The currently active selections will be coupled.

- Channels inside polygons from map layer: Limit the coupling to items within an area (for example a catchment) defined by a polygon. This polygon feature 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 branches (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 the 'Import from file' option. 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 section of the channel to couple (upstream and downstream chainage of the channel).

Coupling Sides

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 the option '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

Determination of which 2D element faces are to be coupled is made based on the extents of cross sections along channels. 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.
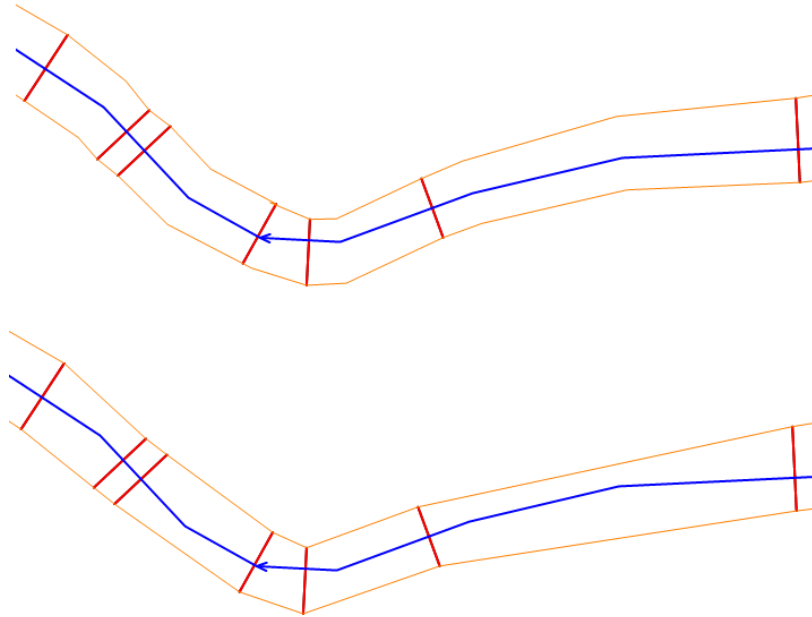
Figure 5.6   Bank coupling location in 2D domain using (at the top) and ignoring (at the bottom) channel alignment

River Ends

The third tab in the Create Couplings tool enables ends of rivers to be coupled to the 2D overland model by ticking on the 'Couple river ends to 2D domain' option. If only coupling to other item types than river ends are needed, leave this option unticked.
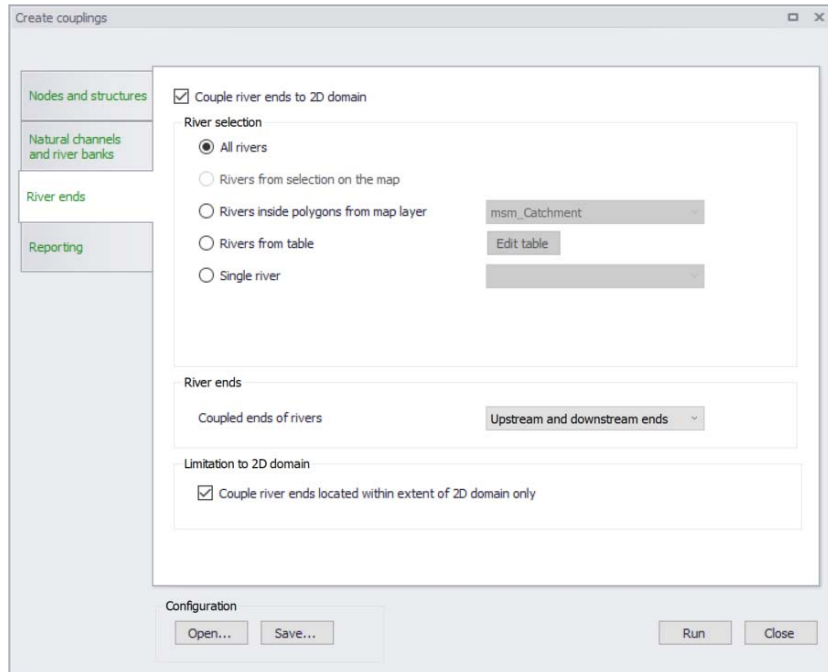
Figure 5.7   Couple river ends using the Create Couplings tool

River Selection

A number of selection methods are available for defining which rivers to couple:

- All rivers: Creates a coupling definition for all rivers (specified in River network Rivers or in MIKE HYDRO).

- Rivers from selection on the map: Select rivers via the Map or Rivers editor 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 area defined by a polygon must be part of the model (e.g. msm_Catchment data layer) 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 couple. Alternatively import a list of channels in text format via the 'Import from file' option. The text file must contain two space-delimited columns with a value representing the coupled end (1 for Upstream, 2 for Downstream) and the river ID.

- 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

Use the 'Save' button to save configuration settings in *.XML format for reuse later or in another model.

The 'Open' button loads a previously-saved configuration file.

Once your configuration is complete, run the tool using the 'Run' button 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

The Edit Location tool is used for editing existing couplings between 1D and 2D model elements. The tool replicates the functionality of the Create Couplings tool (see Create Couplings, page 163) but unlike the Create Couplings tool, this tool will update the location (i.e. coordinates) of existing coupling definitions in the 1D-2D Couplings editor.
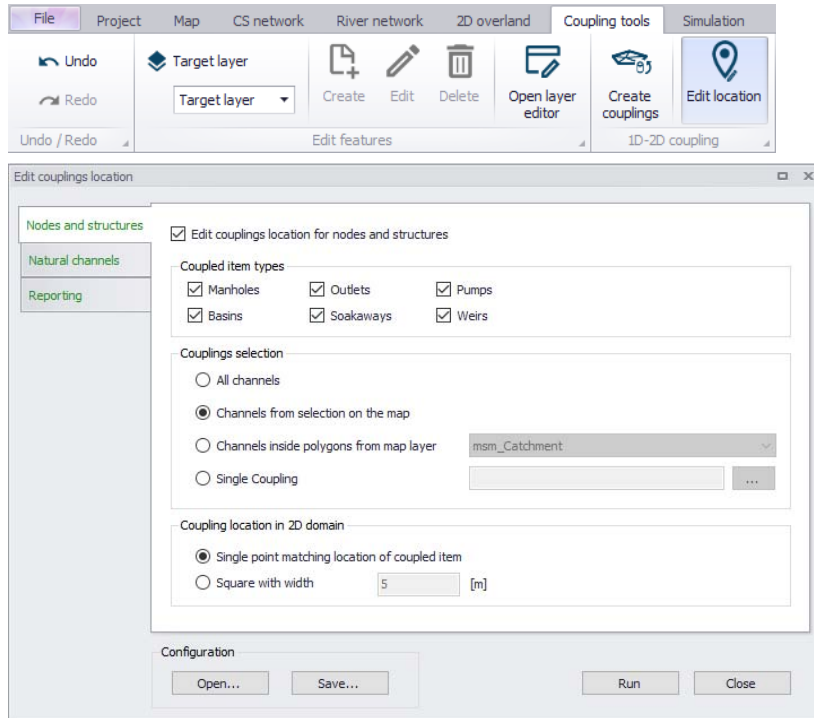
Figure 5.8   Edit Location coupling tool

This tool enables fast updating of locations for multiple coupling items at once. Batch editing of couplings may be especially useful when computation instabilities are encountered in the integrated model due to high flow exchanges through couplings. The tool may be used to distribute flow from 1D model elements to more 2D cells/elements and reduce flow velocities over the coupled cells/elements.

Edit, Delete
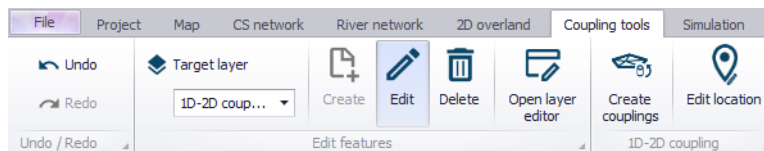
Figure 5.9   Edit and Delete tools on the Coupling Tools menu ribbon

The locations of couplings can be manually edited on the Map. Set '1D-2D couplings' as the target layer in the Edit Features toolbox on the Coupling Tools menu ribbon and activate the 'Edit' button.

Click on a coupling feature 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 using the 'Delete' button and clicking on the desired coupling feature.

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

5.3.3 1D-2D Couplings Editor

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

Identification

Each coupling definition has a unique ID (editable) and information on the 1D network item type coupled (e.g. manhole, basin, river bank, pump, etc.).

There is also a switch to control which couplings are activate during a simulation via the 'Apply' tick box.

When coupling to an external MIKE HYDRO River model setup, some coupling types can be coupled to either the 2D overland model or to a 1D network model. A 'Couple to' option allows 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 coupling types using the 'Inlet area' parameter in the 'Attributes' tab, the area is recomputed to a default value whenever the node ID is re-selected from the list. The default value is based on node geometry. For SWMM5 models where nodes have no diameters, the Inlet Area parameter is initially assigned a value of 0.

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' secondary 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' secondary 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 computational instabilities. Points defining the polygon can be added, removed, or reordered 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 computational stability.

The 'Square width' option automatically updates the coupling to a polygon with a square shape and with the specified width when the 'Update' button 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, 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 specifying 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
The coupling types 'Natural Channel', 'River bank' and 'MIKE HYDRO River bank' all describe lateral coupling of a channel that describes 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 information differs.

Note that at present, there are no options for laterally coupling SWMM5 open conduit banks to 2D overland models.

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 secondary table is automatically populated with the default location when the channel ID, chainages, and coupled sides are specified.

The following methods are also available for controlling 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 menu 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 using the Create Couplings tool (see Create Couplings page 163).

Tip: If cross sections of channels in a model setup are generally wider than the width of the cells/elements in the 2D domain, it may be appropriate to exclude the cells in the 2D domain lying within the channel bed and hence avoid flow calculation 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. Use the Exclude Rivers tool available via the 2D Overland menu ribbon to exclude the river extent from the 2D computational mesh/grid.

River End, MIKE HYDRO River End

River ID
The river's name (specified in River network | Rivers or in MIKE HYDRO River) 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 is coupled to the 2D overland model, the river end is coupled to a string of element faces from the 2D domain. The coupled element faces are the closest faces from the coupling line, defined in the 'Location in 2D domain' secondary 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 ends are specified.

The following methods are also available for controlling 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 menu 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 using the Create Couplings tool (see Create Couplings page 163).

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 defined. The node can be selected from the ellipsis button (…), or using the 'Get closest' function, which will find the node on the collection system network closest to the river end.

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 structures of 'Side structure' type along the selected river will show up in the list.

Side

The choice between left and right sides controls along 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' secondary 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 element faces from the 2D domain. The coupled element faces are the closest faces from the coupling line defined in the 'Location in 2D domain' secondary table.

The table is automatically populated with a default location when the structure ID and 'Length and distance from the river' values 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 for controlling and editing 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 menu 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 definitions and flow exchange parameters in the Attributes tab page of the 1D-2D Couplings editor.

Nodes

The coupling to nodes is designed to describe the interaction of water when a manhole, basin, junction, storage unit, slow divider, or soakaway node is overtopped, or when overland flow enters and exits a sewer/storm water network.
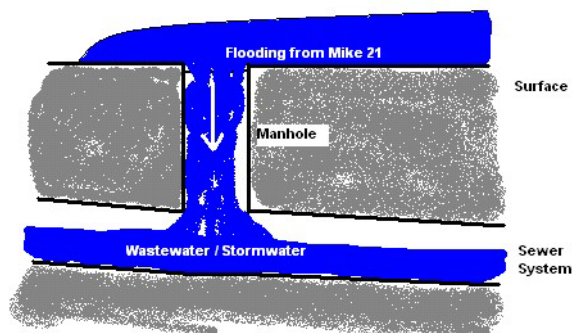
Figure 5.10   Flooding from overland into a non-surcharged sewer system
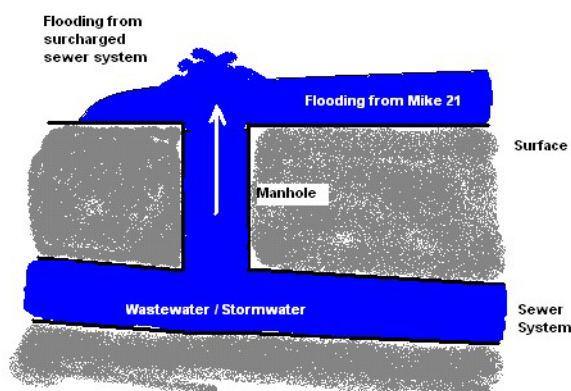
Figure 5.11   Flooding from a surcharged sewer system onto the 2D overland surface

The coupling to nodes may also be used for linking a sewer outlet/outfall 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 in the 1D model.

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 stronger smoothing in the model. Values are only valid in the range [0 to 1].

- For couplings to manholes, basins, junctions, storage units, flow dividers, 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/outfalls 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 (e.g oscillates wildly).

A high smoothing factor will cause a time lag in the transfer of the values from one model component to the other.

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 to stabilize setups where instability is caused by a small differences in the water levels in the coupled models.

If a value of 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:

$$\text{Suppression factor} = 1 - \left(\frac{DeltaDepth - dh}{DeltaDepth}\right)^2 \tag{5.1}$$

Max. Q

The coupling gives the option of placing an upper limit on the flow exchange. This is only available for couplings to manholes, basins, junctions, storage units, flow dividers, 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 flows 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, it is fully applied to the network (not to the coupling) as network load, 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. For SWMM5 models where nodes have no diameters, the Inlet Area parameter is initially assigned a value of 0. 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.
  - – Crest width: Describes the flow exchange between the urban and the 2D grid/mesh based on the weir formula. This parameter may correspond 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 (Ground 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 ellipsis 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 stronger smoothing in the model. Values are only valid in the range [0 to 1].

Natural Channel, 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 stronger 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 (i.e. oscillates wildly).

A high smoothing factor will cause a time lag in the transfer of the values from one model component to the other.

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 setups where the instability is caused by a small difference in the water level in the two coupled models.

If a value of 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 parameters for the weir flow calculation to be performed between the 1D and 2D components:

- 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 levels 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 levee 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 ellipsis button (...).

Please note: The crest level should not be below the 2D topography nor below the bed level in the channel. Thus, if the selection is such that this occurs, MIKE+ will use the limiting value.

River End, MIKE HYDRO River End, Side Structure

When coupling the end of a river 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 stronger smoothing in the model. Values are only valid in the range [0 to 1].

Flow Distribution

A non-uniform distribution indicates that flow from the river is distributed into the coupled cells/elements according to their respective water depths. 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 over the coupled cells/elements.

5.3.4   Ground Level at Couplings to Nodes

When coupling nodes from the 1D network to 2D overland, one ground level is specified for the network and another one is specified in the 2D domain file. These two values should normally be consistent, but due to differences of source data sets and accuracies, some discrepancies may arise.

When coupling manholes / junctions, their ground level is automatically adjusted during the simulation, and replaced by the ground level defined in the 2D domain, in order to ensure consistency and avoid volume errors. Modified ground levels will be reported in the simulation log file. But this can lead

to errors when starting the simulation, especially when the discrepancy is high: in that case, it can happen that the modified ground level in the node becomes lower than its invert level, which is not allowed and therefore the simulation will not start. For this reason, it is worth checking the consistency of ground levels in the network and in the 2D domain.

This can e.g. be achieved using user-defined columns, like this:

- Create a new user-defined column in the nodes editor to store the ground level from the 2D domain file

- Use the 'Interpolation and Assignment Tool' to assign the ground level from the 2D domain file to this new column

- Create another user-defined column in the nodes editor and use the 'Field calculator' to compute the difference between the ground level defined in the node and the ground level assigned from the 2D domain file

- You can finally use this column with differences to sort the nodes by differences, select or zoom in to troublesome nodes, show differences as labels on the map, etc.

Note: When the 'Location in 2D domain' of the coupling is defined by a polygon, the node is coupled to all elements / cells from the 2D domain which have their centroid within the polygon. The resulting ground level obtained from the 2D domain (and applied to the node) is then the average level from all the coupled elements / cells.

5.4 Groundwater Couplings

The 'Groundwater couplings' editor allows the coupling of all or parts of the river and/or collection system network to a MIKE SHE groundwater model. It also allows coupling the river network to a FEFLOW groundwater model.

To enable groundwater coupling, tick the 'Coupling to MIKE SHE or FEFLOW' option in the 'Model type' editor.

The network items (rivers, manholes or pipes) included in the coupling are those listed in the overview table at the bottom of the editor. Coupled network items are added or removed using the 'Insert' and 'Delete' buttons at the top.

The button 'Insert selected' will add all rivers, pipes and/or manholes which are selected on the map and not already present in the list of couplings.

The following parameters must be configured for each groundwater coupling.

5.4.1 Identification

ID

The descriptive name of the coupling.

Type

The type of item from the network to be coupled to the groundwater model. The available options are:

- River
- Manhole
- Pipe

Apply

This check box allows to toggle the Active status of the coupling on and off. The simulations will omit all couplings that are not active.

5.4.2 Location

River ID

Name of the river where the groundwater coupling should be applied. One river can be sub-divided into several reaches. A reason for doing so could be to allow different riverbed leakage coefficients for different reaches of the river.

From lowest chainage

Chainage of the start of the river reach to be coupled to the groundwater model.

To highest chainage

Chainage of the end of the river reach to be coupled to the groundwater model. If the highest chainage is set equal to the lowest chainage, the coupling will be performed in a single point on the river.

Node ID

Name of the manhole where the groundwater coupling should be applied.

Pipe ID

Name of the pipe where the groundwater coupling should be applied.

5.4.3 Exchange attributes for rivers

When the Weir formula option has been selected in MIKE SHE for Overland-River exchange calculation, the overbank spilling will be calculated using the

'Villemonte formula' weir equation. The parameters below, required for this equation, must be defined for each coupling reach.

Weir coefficient
This parameter refers to the weir coefficient in the Villemonte formula.

Weir exponential coefficient
This parameter refers to the weir exponent in the Villemonte formula.

Min. upstream height above bank for full weir width
In the Villemonte formula, when upstream water depth above the weir approaches zero, the flow over the weir becomes undefined. To prevent numerical problems, the flow is reduced linearly to zero when the water depth is below this minimum upstream height threshold.

Allow overbank spilling from river to overland domain
Activate this checkbox to allow overbank spilling from the river to the overland grid cells. If left unchecked, the exchange of overland flow is only one-directional. That is, from the overland model to the river.

Min. flow area for overbank spilling
The minimum flow area threshold prevents overbank spilling when the river is nearly dry. The flow area is calculated by dividing the volume of water in a coupling reach by the length of the reach. If the calculated flow area is less than the specified threshold value, overbank spilling from the river to the overland flow is temporarily disabled.

Conductance
The conductance can be defined through three different types of exchange between surface water and aquifer (described in the technical documentation of the MIKE SHE User Manual). The available options are:

- Aquifer only
- River bed only
- Aquifer + river bed

Leakage coefficient
Leakage coefficient for the river bed lining (see MIKE SHE documentation). The leakage coefficient is relevant only if the conductance type is either 'Aquifer + river bed' or 'River bed only'.

Linear reservoir exchange
When applying the Linear Reservoir mode for Saturated Flow in MIKE SHE, exchange between the river and the baseflow can be defined individually in coupling reaches as either a Gaining river or a Losing river. That is, only one directional exchange is possible for this option.

- **Gaining river**: with Gaining river, the river receives water from the base-flow reservoirs in MIKE SHE
- **Losing river**: with Losing river, the river loses water to the baseflow reservoirs in MIKE SHE as a function of the leakage coefficient, water depth in river, bank width and length of the coupling reach.

5.4.4   Exchange attributes for manholes and pipes

Couple to overland flow

Activate this check box to couple the manhole or pipe to the overland component of MIKE SHE.

Weir coefficient

This parameter refers to the weir coefficient in the Villemonte formula.

Weir exponent

This parameter refers to the weir exponent in the Villemonte formula.

Couple to overland drainage

Activate this check box to couple the manhole to the overland drainage component of MIKE SHE.

Overland drainage grid code

The overland drainage grid code is used for mapping MIKE SHE cells to the coupled manhole in MIKE+. It is important to use a unique code for each coupled manhole.

Couple to saturated zone drainage

Activate this check box to couple the manhole to the saturated zone drainage component of MIKE SHE.

Saturated zone drainage grid code

The saturated zone drainage grid code is used for mapping MIKE SHE cells to the coupled manhole in MIKE+. It is important to use a unique code for each coupled manhole.

Couple to saturated zone

Activate this check box to couple the pipe to the saturated zone component of MIKE SHE.

Conductance

The conductance can be defined through three different types of exchange between surface water and aquifer. The available options are:

- Link leakage + SZ conductivity
- Grouting

- Link leakage only

Leakage coefficient

Leakage coefficient for the pipe (see MIKE SHE documentation). The leakage coefficient is relevant only if the conductance type is either 'Link leakage + SZ conductivity ' or 'Link leakage only'.

5.4.5   Flooding

Flood area option

The Flood area option allows a number of cells from the MIKE SHE model to be flooded (being part of a river, lake, reservoir etc.) based on the water level from the river model. When this option is active, a river or a lake (with wide cross sections) may cause flooding of a number of cells in MIKE SHE. A correspondence / mapping is established between MIKE+ river h-points and individual cells in MIKE SHE. Subsequently a simple flood-mapping procedure is adopted to calculate water depths on the ground in MIKE SHE. The flood mapping procedure simply compares simulated water levels (in the corresponding h-point) with the ground level in the MIKE SHE cells. If the water level is higher than the ground level, flooding occurs.

Three options are available:

- No flooding
- Manual
- Automatic

The selected option may differ for each coupled reach. These options are described below.

No flooding

If the 'No flooding' option is adopted, rivers are considered lines located between adjacent cells from the MIKE SHE model. No flooding can occur and overbank spilling is not possible, even if the water level in the river rises above the elevation of the adjacent cells in MIKE SHE.

River-Overland exchange is always one way with this option, namely overland to river. If this option is used, one of the three river-aquifer exchange formulations will be adopted.

Manual

The manual option allows the user to delineate the potentially flooded areas, using a 2D file (.dfs2) containing code values - the flood code file specified in MIKE SHE's user interface. If the river system considered is a very complex system with a looped or meandering network, generating a complex geometry, the best result will be obtained by creating a flood code file manually by digitizing the floodplain / lake, and using this option.

Each (potentially flooded) MIKE SHE cell is linked to the nearest MIKE+'s river h-point on a coupling reach with the same flood code value. The flood code specified for each coupling reach must match the grid code in the flood code file. The flood code must be an integer.

Automatic

The automatic option is often useful if the geometry of rivers, lakes etc. is not too complex. This option may, for instance, be applied for wide rivers without too much meandering.

When the automatic option is chosen, MIKE SHE will automatically generate the potentially flooded areas (flood grid code map) depending on the location of the river branches and on the width and location of their cross-sections. The specified flood code is used as grid code, and the flood-mapping procedure described above is applied. Thus it is important to use unique flood codes to ensure correct mapping between MIKE+ and MIKE SHE.

Flood code

As described above the flood code is used for mapping MIKE SHE cells to MIKE+ rivers h-points, and for the automatic option also for generating the flood grid codes of the coupling reach. It is important to use unique flood codes to ensure correct flood-mapping.

Bed topography

This option controls the source of the topography used in the flooded area. It should be emphasized that the flood mapping requires a good consistency between the river cross-sections and the ground levels in MIKE SHE in order to provide relevant results.

The options are:

- **Use cross sections**

  When this option is used, the ground levels of the flooded cells are substituted with values directly interpolated from the river cross sections. Notes on this option:

  – This interpolation uses an inverse-distance-weighted method, using points (elevations) from the cross sections as discrete input points. When the distance between cross sections is higher than half the cell size, extra cross sections are interpolated before the cells' ground levels are interpolated.
  – Please note that the ground levels are only interpolated inside the area delineated by the cross sections.
  – When the Manual option is used, the user-defined flood area does not have to be identical to the area covered by the cross sections. If the Automatic option is used, the area covered by the cross sections and the flood area will always be consistent, as the flood-area is generated (automatically) based on the cross sections.

- – In principle the 'Use cross sections' option ensures a good consistency between MIKE SHE ground levels and river cross sections. There will, however, often be interpolation problems related to river meandering, tributary connections, etc. where wide cross sections of different coupling reaches overlap. Thus, it is recommended to make the initial model set up using the 'Use cross sections' option and then subsequently retrieve and check the resulting ground levels (using the MIKE SHE Input Retrieval tool). If needed the retrieved ground levels (.dfs2 file) can be modified (MIKE SHE Graphical Editor) and then used as input for a new set up, now using the 'Use grid data' option described below.

- **Use grid data**

  MIKE SHE grid data is used instead of river cross sections. This option checks whether the optional bed elevation file has been specified in MIKE SHE's user interface:

  - – If the bed elevation file has been specified, the ground levels of the flooded cells are substituted with values from the specified .dfs2 file. The option is useful when the surface elevation data of the flood areas is more detailed than the regional terrain model.
  - – If the bed elevation file has not been specified, then the regional MIKE SHE surface topography is also used in flood areas. As described above, the specified .dfs2 file will often be a retrieved and modified surface topography from a previous set-up with the 'Use cross sections' option.

Bed leakage

As described in the technical documentation of MIKE SHE, the infiltration/seepage of MIKE SHE is calculated as ordinary exchange from the overland domain to the saturated or unsaturated zone, either using full contact or reduced contact with a specified leakage coefficient.

The bed leakage option tells whether the overland-groundwater exchange option and leakage coefficient specified in MIKE SHE's user interface should also be used in the actual flood area, or substituted by the corresponding river-aquifer conductance type and Leakage coefficient specified for the coupling reach.

Options are:

- **Use grid data**

  The overland-groundwater exchange option and leakage coefficient specified in MIKE SHE's user interface is used. Both can be single value or distributed (from a .dfs2 file).

- **Use river data data**

The MIKE SHE overland-groundwater exchange option and leakage coefficient in flood areas are substituted with the corresponding river-aquifer conductance type and leakage coefficient specified for the coupling reach. Please note that the two reduced contact options (exchange types B and C) lead to the same overland-groundwater exchange option.

The substitution is made in all cells from the flood area of the coupling reach.

6 Water Quality

MIKE+ provides several modules for the simulation of water quality for both urban catchment surfaces and sewer systems.

MIKE+ can model complex water quality and pollutant transport mechanisms using Stormwater Runoff Quality (SWQ), Advection-Dispersion (AD), and the MIKE ECO Lab engine to solve Biological Processes equations. The MIKE ECO Lab framework offers versatile and flexible options for WQ-modelling. Water Quality simulations can be performed using either standard, predefined MIKE ECO Lab templates or user-defined templates with tailored Water Quality models.

This user guide focuses on models involving 2D overland features. Further details on water quality modelling for 1D models may be obtained from the respective user guides (i.e. MIKE+ Collection System, MIKE+ River Network Modelling or MIKE+ SWMM Modelling User Guides).

Furthermore, note that at present, couplings of water quality components between 1D and 2D models is only available for models of type 'Rivers, collection system and overland flows' in MIKE+. Thus, this chapter presents information relevant only for projects with models of this type.

6.1 Advection-Dispersion

6.1.1 WQ Components

The Advection-Dispersion model can be used for calculation of the transport of dissolved or suspended substances, and for modelling of water temperature variation within the domain. The model is based on the 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 main assumptions of the model are:

- The considered substance is completely mixed within an element / cell of the 2D domain. This implies that a source term is considered to mix instantaneously within the domain's element;

- The substance is conservative or subject to a first order reaction (linear decay).

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 modelled, e.g. initial concentrations, dispersion coefficients and decay rates.

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 specified WQ components can be declared as 'Pollutant', 'Microorganisms', 'Temperature', 'pH', 'Salinity', 'Water Age', 'Water Blend', 'Other' or 'Fixed matter'. This categorization is used for handling of units: each component type has its own list of possible units.

When working with a water quality model using MIKE ECO Lab, each MIKE ECO Lab state variable is associated to a WQ component from this 'WQ component' editor, and the biological processes are controlled by properties defined in the MIKE ECO Lab editors. The 'Fixed matter' type of WQ component is a special type which is relevant for this MIKE ECO Lab models, and is used to describe e.g. components deposited at the bottom and not moving with water.

Decay of the WQ components is defined in the '2D decay' editor, and therefore the 'Decay' parameter in the 'WQ components' editor is ignored for 2D overland modelling.

Initial conditions for the WQ components can be specified in the '2D AD Initial Conditions' editor. Boundary conditions for the WQ components can be specified in the '2D WQ boundaries' editor. Refer to the corresponding chapters for more description.

6.1.2 2D AD Decay

Many processes can be approximated by a simple first-order decay, such as die-off of E. Coli due to exposure to light, decay of the activity of radioactive substances or estimating the age of water bodies.

First order decay of a component is generally described by

$$\frac{dc}{dt} = kc \tag{6.1}$$

or

$$c = c_0 \cdot e^{-kt} \tag{6.2}$$

where c is the specific concentration, c_0 is the initial concentration and k is the decay factor. In the model the decay term is added to the general transport equation.

The decay is specified individually for each component.

The format of the decay factor can be specified as

- None
- Constant (in time)
- Varying in time

For the case with time varying decay factors, a time series file (dfs0) containing the decay factors must be supplied. 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.

Note: The decay may affect the stability of the numerical solution, in a way similar to the advection or diffusion terms. If the decay represents a very rapid process such that the product kΔt>1 the decay term may be the source of instability or at least occurrence of negative concentrations. A solution is then to reduce the time step.

Note: The decay is only relevant for Advection-Dispersion components. For state variables from a MIKE ECO Lab model, decay is simulated as part of the biological processes. As a consequence, no decay information has to be specified for WQ components associated to a MIKE ECO Lab state variable.

6.1.3 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:

- Uniform (in both time and domain)
- Varying in domain

For the case with dispersion coefficient varying in domain, you can prepare a data file containing the dispersion coefficient before you set up the Advection-Dispersion simulation, and use it with the source type '2D file'. 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.

Alternatively, you can select the source type 'MIKE+ 2D AD dispersion layer' to assign constant values to a number of polygons drawn on the map, or select the source type 'Background layer' if a file (e.g. shape file) containing polygons and an attribute holding the dispersion coefficient, is available.

Scaled eddy viscosity

When selecting 'Eddy viscosity scale factor' option, the format of the scaling factor can be specified as:

- Uniform
- Varying in domain

For the case with values varying in domain, you can prepare a data file containing the scaling factor before you set up the Advection-Dispersion simulation, and use it with source type '2D file'. 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.

Alternatively, you can select the source type 'MIKE+ 2D AD dispersion layer' to assign constant values to a number of polygons drawn on the map, or

select the source type 'Background layer' if a file (e.g. shape file) containing polygons and an attribute holding the dispersion coefficient, is available.

Alternatively, you can select the source type 'MIKE+ 2D AD dispersion layer' to assign constant values to a number of polygons drawn on the map, or select the source type 'Background layer' if a file (e.g. shape file) containing polygons and an attribute holding the dispersion coefficient, is available.
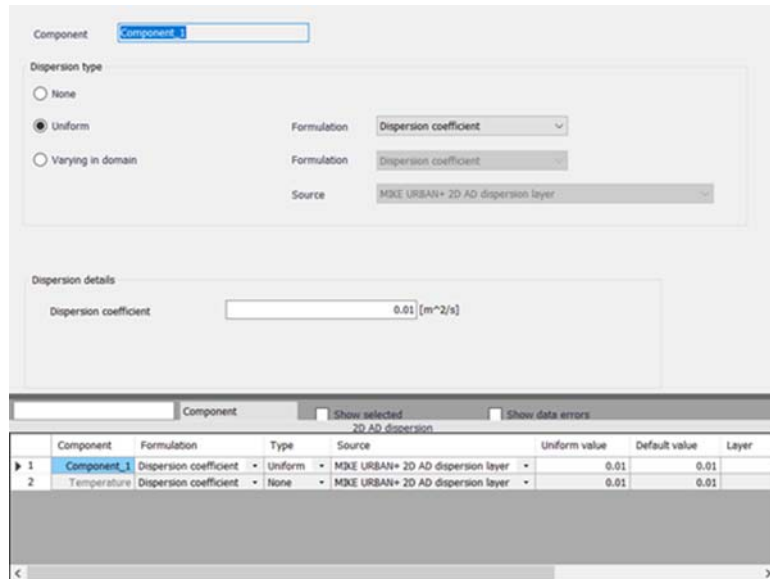

Figure 6.1   2D AD Dispersion editor

6.1.4   2D AD Initial Conditions

The format of the initial concentration (in component unit) for each component can be specified as:

- Uniform (in domain)
- Varying in domain

For the case with varying in domain, you can prepare a data file containing the concentration (in component unit) before you set up the Advection-Dispersion simulation, and use it with source type '2D file'.

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.

Alternatively, you can select the source type 'MIKE+ 2D AD initial condition layer' to assign constant values to a number of polygons drawn on the map, or select the source type 'Background layer' if a file (e.g. shape file) containing polygons and an attribute holding the dispersion coefficient, is available.

Figure 6.2   2D AD Initial Conditions editor

6.2   Water Quality with MIKE ECO Lab

Biological processes can be modelled through MIKE ECO Lab. It simulates the reaction processes in multi-compound systems, including degradation of organic matter, bacterial fate, exchange of oxygen with the atmosphere and oxygen demand from eroded sediments. This allows realistic analysis of complex phenomena related to water quality.

Biological processes can e.g. include diurnal variation of foul flow discharges and user-specified concentrations of foul flow components. The sediment types included in the interaction with biological processes are foul flow organic sediments, fine and course mineral in-pipe sediments originating from catchment runoff, potholes, and stilling basins. Biological processes can account for:

- Decay of BOD/COD in biofilm and water phase
- Hydrolysis of suspended matter
- Growth of suspended biomass

- Oxygen consumption from decay of BOD/COD, biofilm, and erosion of sediment

- Re-aeration

- Bacterial fate

- Interaction with sediments for nutrients and metals

Hydrodynamic and advection-dispersion (transport) simulations can be coupled with MIKE ECO Lab for efficient and versatile simulation of water quality processes. This option offers practically an unlimited range of processes to be simulated, as long as these can be described by a consistent model. MIKE ECO Lab in MIKE+ is supported by a set of MIKE ECO Lab editors:

- MIKE ECO Lab Templates

- MIKE ECO Lab State Variables

- MIKE ECO Lab Forcings

- MIKE ECO Lab Constants

MIKE ECO Lab and these four editors are used to model the biological processes. Each MIKE ECO Lab state variable must be associated to an Advection-Dispersion component (from the 'WQ components' editor), and therefore the initial conditions and boundary conditions for the state variables are defined by initial conditions and boundary conditions of these WQ components.

6.2.1 Notes on MIKE ECO Lab

MIKE ECO Lab is a numerical lab for Ecological Modelling. It is a highly flexible and open framework for the formulation of water quality models. It is an open and generic tool for customising Aquatic Ecosystem models to describe water quality, eutrophication, heavy metals, and ecology. The module can describe dissolved substances, particulate matter of dead or living material, living biological organisms and other components (all referred to as state variables in this context).

The module is mostly used for modelling water quality as part of an Environmental Impact Assessment (EIA) of different human activities, but the tool is also applied in aquaculture for e.g optimizing the production of fish, sea grasses, and mussels. Another use is in online forecasts of water quality.

Templates

MIKE ECO Lab uses template files (*.ecolab file) where water quality models are transparently defined. MIKE ECO Lab template files are customized collections of equations and parameters required for a specific type of Water Quality simulation.

The user may use a predefined MIKE ECO Lab Template installed with the software or may choose to develop their own. The MIKE+ installation includes some pre-defined water quality templates that can be applied "as is" or adjusted to conform to specific project requirements. A MIKE ECO Lab template is an ASCII file which can be accessed and edited through the MIKE ECO Lab editor.

Note: It is necessary to install MIKE Zero in order to edit a template file in the MIKE ECO Lab editor.

Please consult the MIKE Zero ECO Lab documentation on how to create or modify model templates, i.e. how to work with the MIKE ECO Lab editor.

A simulation job with the coupled HD/AD and MIKE ECO Lab model is set up and launched from within MIKE+ in the usual way. Also see Chapter 6.2.6 Running MIKE ECO Lab Simulations *(p. 205)*.

Integration Methods

Water quality calculations are based on predefined MIKE ECO Lab templates which contain models defined by a number of coupled differential equations solved through numerical integration and interactions between each equation.

Several Integration Methods are available for solving the coupled ordinary differential equations defined in the MIKE ECO Lab file. These options are offered in the Simulation Setup editor in MIKE+. See Chapter 6.2.6 Running MIKE ECO Lab Simulations *(p. 205)*.

Three integration routines (solution methods) are available (please consult the MIKE ECO Lab Reference Manual for details on the methods for solving the coupled linear differential equations in the MIKE ECO Lab framework):

- **Euler**: Euler or Linear Solution
  A very simple numerical solution method for solving ordinary differential equations.

- **RK4**: Fourth order Runge-Kutta.
  A classical numerical solution method for solving ordinary differential equations. It has normally higher accuracy than the Euler method but requires longer simulation times. The fourth order Runge Kutta method requires four evaluations of equations per time step.

- **RKQC**: Fifth order Runge-Kutta with Quality Control
  A numerical solution method for solving ordinary differential equations. The accuracy is evaluated and the time step is adjusted if results are not accurate enough. The method requires 6 evaluations at each time step to take a so-called Cash-Karp Runge Kutta step and the error is estimated as the difference between a Runge Kutta fourth order solution and the Runge Kutta fifth order solution.

The accuracy (and the computing time) varies for the three integration routines.

The most accurate result will be calculated when using RKQC. However, in some cases the same results can be obtained using less computational time with the less advanced options: RK4 or EULER.

In general, it is recommended to use the RKQC routine. RK4 ad EULER methods are generally only applied during the set-up and initial calibration phase of a project. If the RK4 or the EULER routines are used, it is strongly recommended to run an additional simulation with the RKQC routine and compare the two results (RKQC versus RK4/ EULER) before making any conclusions based on the model.

In the case of a very dynamic model system with steep concentration gradients in one or more of the components, integration may not be possible when using the RKQC routine, and an error message will appear. Reducing the time step will help in most cases, but sometimes the gradients are so steep, that they cannot be solved accurately. The Quality Control of RKQC ensures that all components are calculated within an accuracy of 1 µg/l. Using the second best routine (RK4), where no Quality Control is included, the steep gradients can be solved in a relatively accurate way and RK4 is therefore recommended when integration is impossible with the RKQC routine.

6.2.2 MIKE ECO Lab Templates

A MIKE ECO Lab template, describing the water quality processes, must be included in the project. Multiple templates can be used, if they apply to different spatial locations on the network. Templates are selected via the MIKE ECO Lab Templates editor (Figure 6.3).
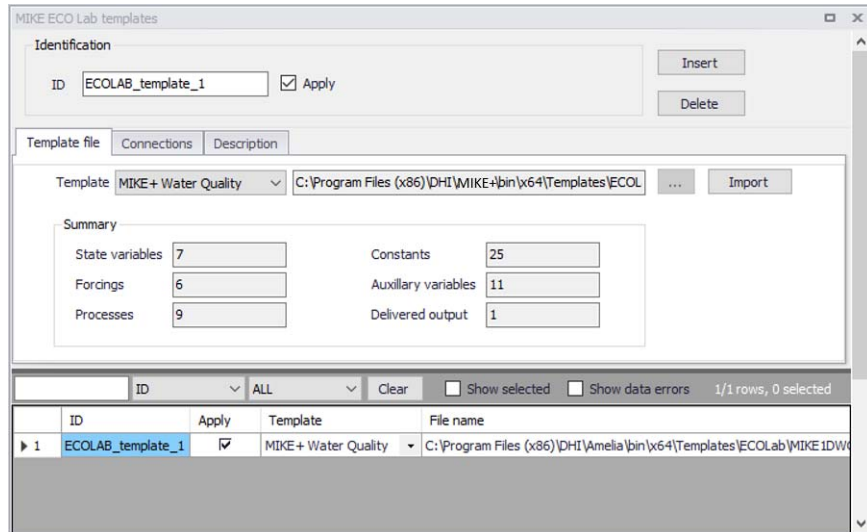
Figure 6.3   The MIKE ECO Lab Templates editor in MIKE+

For coupled simulations, coupling a 1D network to the 2D overland model, it is therefore possible to include one or more templates to model water quality in the network, and another template to describe water quality in the 2D domain.

A template is included in the simulation only if its 'Apply' check box is ticked, and is ignored otherwise.

Template File

Predefined MIKE ECO Lab templates are included in the MIKE+ installation located in the folder: 'C:\Program Files (x86)\DHI\MIKE+\<Version>\Templates\ECOLab'.

Point to the MIKE ECO Lab template file in the Template File tab page of the editor:

- **Template**: Define the MIKE ECO Lab template to which to connect using the drop-down list (or the ellipsis button for custom templates).

- **Import**: Use the Import button to read and import the contents of the template file (state variables, forcings and constants) into the MIKE+ project and editors. Pressing the Import button after the template has already been loaded (imported) will reload the template and reset all state variables, forcings and constants to default settings.

- **Summary**: Displays general information on the contents of the MIKE ECO Lab template file, such as the number of State variables, Forcings, Processes, Constants, Auxiliary variables, and Delivered output defined in the template.

Connections

Define for which model parts the loaded MIKE ECO Lab template is applied:

- **Global network**: The template is used over the whole network model (River and CS)

- **Nodes**: The template is used for CS network nodes. It may be in All nodes, a List of nodes, or Individual nodes selected from a list accessed via the ellipsis button, or interactively from the Map using the arrow button.

- **Links**: The template is used for links (i.e. River branches and/or CS pipes and canals). It may be in All links, a List of links, or Individual links selected from a list accessed via the ellipsis button, or interactively from the Map using the arrow button.

- **2D overland**: The template is used for the entire 2D domain as part of a 2D overland simulation. Only one template can apply at a time for a 2D overland simulation.
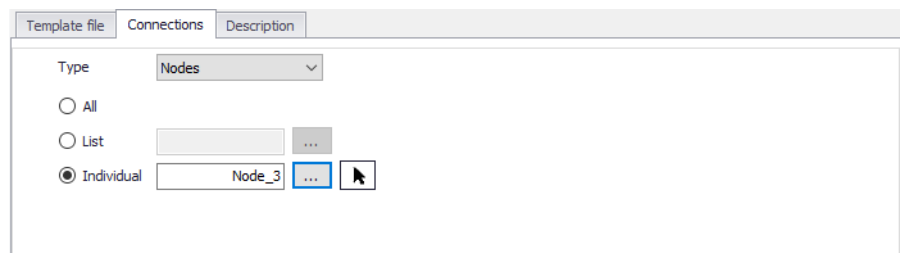

Figure 6.4   The Connections tab page in the MIKE ECO Lab Templates editor

Description

Add a free text description for a MIKE ECO Lab template on the Description tab page of the MIKE ECO Lab Templates editor.

6.2.3   MIKE ECO Lab State Variables

The MIKE ECO Lab State Variables editor is used for coupling MIKE+ WQ components to MIKE ECO Lab state variables and for possible reconciliation of differences in units used in MIKE+ and MIKE ECO Lab, respectively.

- **ID**: State variable identifier in MIKE+.

- **MIKE ECO Lab template**: MIKE+ ID for the template to which the state variable belongs.

- **State Variable**: State variable identifier in MIKE ECO Lab template file.

- **WQ Component**: MIKE+ WQ component coupled to the MIKE ECO Lab state variable.

- **Conversion Factor**: Conversion factor for possible units conversion between MIKE+ and MIKE ECO Lab

- **Description**: Add a free text description for a state variable via the Description tab page of the editor.

In the MIKE+ WQ Components editor, there must be one component defined for every MIKE ECO Lab state variable defined in the MIKE ECO Lab template. This means that all MIKE ECO Lab state variables are subject to both AD transport mechanisms and MIKE ECO Lab transformations.

For a **State Variable**, the associated **WQ Component** must be selected by the user. These WQ components must represent the same constituents as those in the MIKE ECO Lab template, but their names can be freely defined.

For coupled models, coupling a 1D network to the 2D overland model, different templates have to be used for the 1D network and for the 2D domain. When state variables from the different templates describe the same component, they must be associated to the same WQ component in order to be coupled during the simulation

The specified conditions and boundary conditions for the WQ components will therefore control the initial and boundary conditions of the state variables.

Figure 6.5   MIKE ECO Lab State Variables

The import operation described in the MIKE ECO Lab Templates editor will automatically create the required entries in the MIKE ECO Lab State Varia-

bles editor. The user must then couple each of these MIKE ECO Lab state variables to the relevant WQ component. Select the appropriate component from the WQ Component dropdown list, which is populated by components defined in the WQ Components editor.

The **Conversion Factor** (default value 1, i.e. no conversion) may be set different from 1.0 if the units for the AD components and MIKE ECO Lab state variables are different.

Note that the active fields in the editor are only those for coupling the MIKE+ WQ components with MIKE ECO Lab state variables, and for specifying a conversion factor.

6.2.4   MIKE ECO Lab Forcings

Forcings are external variables which affect some of the processes on the state variables. They are automatically listed in this editor when the MIKE ECO Lab template is imported, from the 'MIKE ECO Lab templates' editor.

Forcings are "Built-In" or "User-Defined", depending on the forcing definition in the MIKE ECO Lab template. "Built-in" means that forcings are provided by the hydraulic model through coupling with MIKE ECO Lab. "User Defined" forcings must be provided by the user, and can be either constant values (constant in time and domain), varying in time or varying in domain and time (the latter option is only enabled when the MIKE ECO Lab template is connected to the 2D overland model).
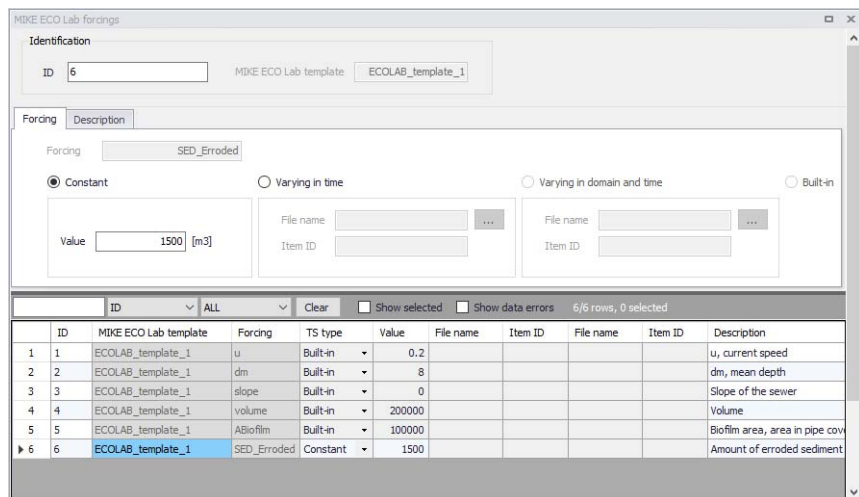

Figure 6.6   MIKE ECO Lab Forcings editor

The editor is used for setting the user-defined forcings associated with the loaded MIKE ECO Lab templates:

- **ID**: Forcing identifier in MIKE+.

- **MIKE ECO Lab template**: MIKE+ ID for the template to which the forcing belongs.

- **Forcing**: Forcing identifier in MIKE ECO Lab template.

- **Value**: Constant value for the forcing.

- **File Name**: Path/file name for the *.DFS0 time series file containing forcing time series data, or for the 2D data file when the forcing is varying in domain and time.

- **Item**: Data item selected in the file.

- **Description**: Free text description for the forcing defined via the Description tab page.

Built-In forcings are also imported into MIKE+, but don't need any configuration.

Note that the active fields in the editor are only those for specifying forcing constant values or input files. The remaining fields are read-only.

Note that for templates connected to the 1D network, some forcings may be defined as built-in in the template but appear as user-defined in MIKE+, in case the corresponding variable does not exist in the model setup. That is especially the case for Temperature and Salinity forcings. In order for them to actually work as a built-in forcing, it is necessary to include a WQ component with the corresponding type (temperature or salinity)..

6.2.5 MIKE ECO Lab Constants

Constants are values which are fixed in time and affect some of the processes on the state variables. They are automatically listed in this editor when the MIKE ECO Lab template is imported, from the 'MIKE ECO Lab templates' editor.

This editor is used for setting the values of the constants used in the MIKE ECO Lab model, typically during model calibration:

- **ID**: Unique MIKE ECO Lab constant identifier in MIKE+.

- **MIKE ECO Lab template**: MIKE+ ID for the template to which the constant belongs.

- **Constant ID**: Constant identifier in MIKE ECO Lab template.

- **Value**: Value for the constant, when it's defined as uniform

- **File name**: Path and file name for the 2D data file containing the constant values, when the constant is defined as varying in domain (only available when the MIKE ECO Lab template is connected to the 2D overland model).

- **Item**: Data item selected in the file.
- **Description**: Free text description for the constant defined in the Description tab page of the editor.

Figure 6.7   MIKE ECO Lab Constants editor in MIKE+

Note that the active fields in the editor are only those for specifying uniform values or input files. The remaining fields are read-only.

6.2.6   Running MIKE ECO Lab Simulations

To run a MIKE ECO Lab simulation, the wanted MIKE ECO Lab template(s) should first be imported to MIKE+. Then, WQ components must be created in the 'WQ components' editor as appropriate and each MIKE ECO Lab state variable must be associated to a WQ component. Make sure that the WQ component is set to use the unit which is expected for the state variable in the template file. Possible unit inconsistencies must be considered and resolved by providing proper conversion factors. Note that no other unit conversion / correction will be performed during the simulation.

Values of constants should be reviewed and modified as appropriate, as well as user-defined forcings.

Initial conditions and boundary conditions of the WQ components in the 1D network must also be specified when they should differ from 0. They must be

defined for all components for the 2D overland model, even when they equal 0.

Result items from the MIKE ECO Lab model to be saved to result files during the simulation, must be configured in the 'Result files' editor.

The simulation with MIKE ECO Lab is activated by activating the 'Transport (AD, SWQ)' and 'Water quality (MIKE ECO Lab)' modules for the simulation setup in the Simulation Setup editor (Figure 6.8).
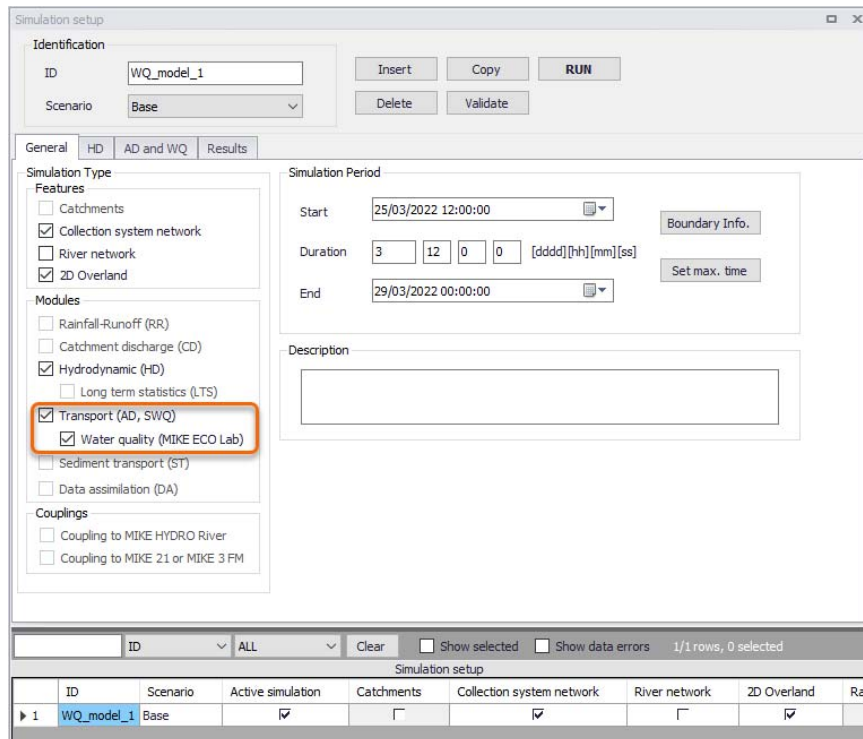

Figure 6.8   Setting up a MIKE ECO Lab simulation in the Simulation Setup editor

In the 'AD and WQ' tab page, define the MIKE ECO Lab update frequency and the MIKE ECO Lab Integration method to use for the simulation. The update frequency is a parameter that defines how often the water quality processes will be calculated during the simulation. It is defined as a multiple of the simulation time step used for the HD and AD simulation and therefore determines the frequency for simulating water quality processes on top of the standard Advection and Dispersion processes which are calculated at every time step. The Update Frequency must be a positive integer. Definition of the frequency should be based on careful considerations. The dynamics of the advection dispersion is comparable to the dynamics of the process descriptions. Thus, it is strongly recommended to use an update frequency of 1. The selection of the Update Frequency has to be based on considerations of the

time scales of the processes involved. Please notice that this selection can be rather decisive for the precision of the numerical solution as well as for the CPU time of the simulation. A large Update Frequency will decrease the precision as well as the CPU time. It is therefore advisable to perform a sensitivity analysis on the Update Frequency before making the final selection. Also see Chapter 6.2.1 Notes on MIKE ECO Lab *(p. 197)* for more information on the Integration Methods.

Figure 6.9   Define the MIKE ECO Lab Integration Method in the AD and WQ tab page of the Simulation Setup editor.

In the 'Results' tab, select the result files to be saved during the simulation.

7 Tables

The Tables section in MIKE+ holds editors for the following tabular data types:

- Curves and Relations
- Two-dimensional Tables
- Materials
- Outlet Head Loss
- On-Grade Captures

Some editors are available depending on the project model type.

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. Available tabular data types vary depending on the project model type.
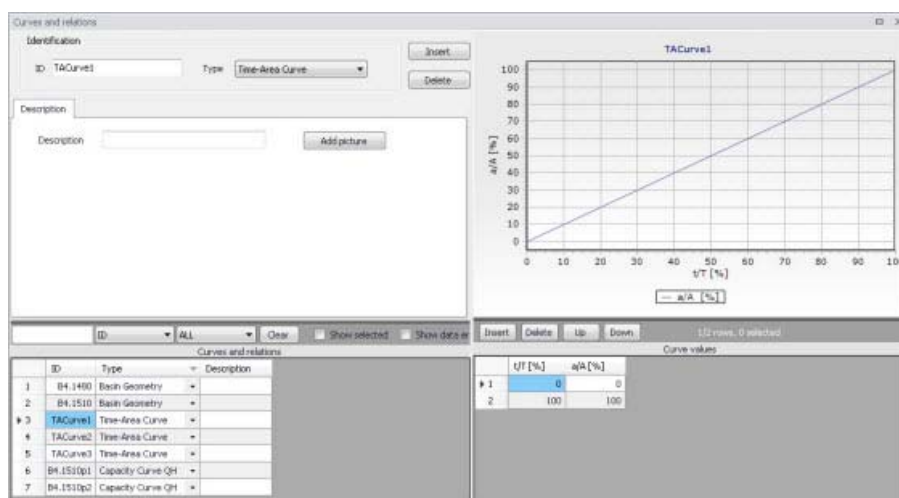

Figure 7.1   Curves and Relations editor

The Curves and Relations editor organizes the related input data into the following groups:

- Identification: General identification and curve type information
- Curve Values: Secondary table containing tabular data values
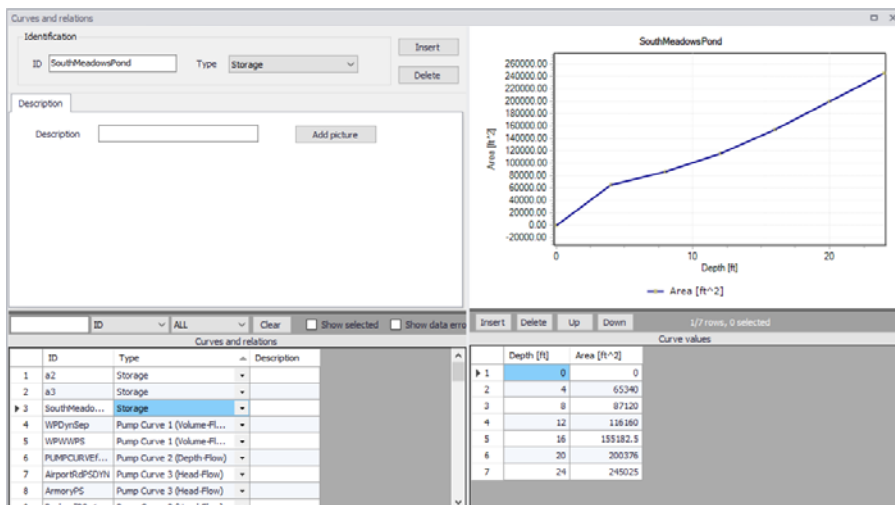
Figure 7.2   The Curves and Relations editor

Identification

The identification groupbox holds curve ID and Type information.

Figure 7.3   The Identification groupbox

Use the Insert or Delete buttons to add or remove records from the editor, respectively. Records are added to the primary table on the lower left corner of the editor.

Figure 7.4   Primary table with the Curves and Relations list on the lower left side of the Curves and Relations editor

Table 7.1   Edit fields in the Curves and Relations Identification and Description groupbox

Edit field	Description	Used or required by simulations	Field name in datastructure
ID	Curve ID	Yes	MUID
Type	Type of curve	Yes	TypeNo
Description	User's descriptive information on the curve	No	Description

Add optional descriptive information for a curve on the Description tab page. An option for adding an images is also available.

Figure 7.5   The Description tab page

Curve Values

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). Secondary table parameters/columns that should be filled vary depending on the curve and relation type.

A plot of the tabular data is also shown on the upper right corner of the editor.
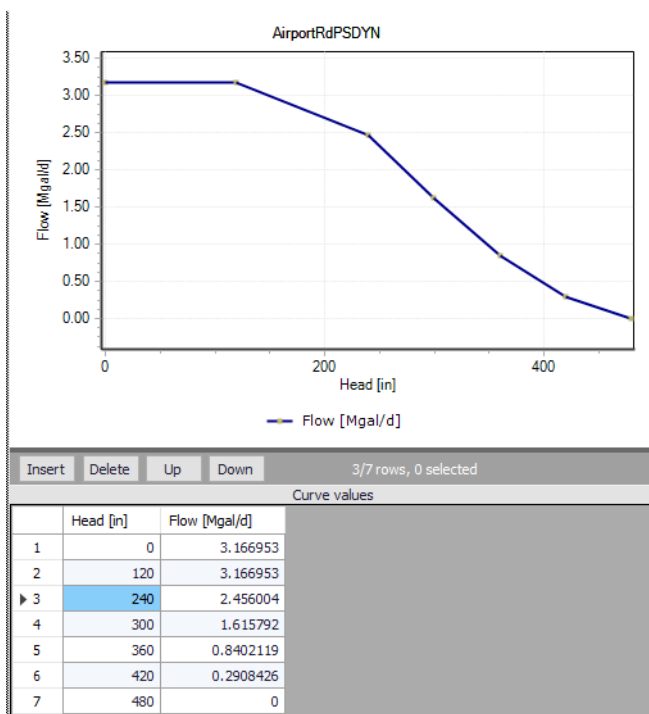
Figure 7.6   Secondary table containing Curves Values on the lower right side of the Curves and Relations editor. Also shown is the tabular data plot above the secondary Curve Values table.

Table 7.2   Edit fields in the Curve Values secondary table

Edit field	Description	Used or required by simulations	Field name in datastructure
'Value1'	Value1, dependent on Type of curve (Depth, Inflow, Hour, Volume, Head)	Yes	Value1
'Value2'	Value2, dependent on Type of curve (Area, Outflow, Stage, Flow)	Yes	Value2

7.1.1   Rivers, Collection System and Overland Flows

For models of type 'Rivers, collection system and overland flows,' available curve types are, among others:

- Capacity Curve QH or Capacity Curve QdH: Used to define pump operation. 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.

- Pump Acceleration Curve: 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.

- Regulation Qmax(H) or Regulation 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.

- 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.

- 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).

- 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+ 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+ maintains T-A curves in percent (%), and the last pair of values in the table must be (100,100).

  There are 3 pre-defined Time-Area Curves in the database (TACurve1, TACurve2 and TACurve3), which should not be deleted.

- 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).

- DQ Relation or QQ Relation (used for Curb Inlets and On Grade Captures): 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.

- 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. After the simulation with a 'Capacity Curve QdH & Power' the summary will contain information on the power consumption during the simulation period.

- Runoff Pollutants (used for SWQ): 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.

- 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+ 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.

- Depth dependent infiltration: Lookup table of Infiltration Rate and Depth values. Infiltration Curve input for 2D Infiltration input of type 'Varying in domain and flow dependent' defined in the 2D Infiltration editor.

- Depth dependent Manning (M) or Depth dependent Manning (n): Lookup table comprised or Depth-Manning M (or Depth-Manning n) value pairs. Used as Roughness Curve input for 2D Surface Roughness input of type 'Varying in domain and flow dependent' that is a function of Depth defined in the 2D Surface Roughness editor.

- Flux dependent Manning (M) or Flux dependent Manning (n): Tabular data comprised or Flux-Manning M (or Flux-Manning n) value pairs. Used as Roughness Curve input for 2D Surface Roughness input of type 'Varying in domain and flow dependent' that is a function on Flux defined in the 2D Surface Roughness editor.

- Generic control rule (used for actions' set point in control rules): Generic control rule 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.

- Control rule time series (used for action's set point in control rules): Control rule time series 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.

- Crest level profile. Tabular Chainage-Level values describing embankment elevation profiles used in defining lateral river bank and natural channel couplings. See 1D-2D Couplings editor Attributes tab page.

- Undefined (general placeholder): The Undefined table type is an extra generic type of table used as a placeholder for potential future functionality.

7.1.2 SWMM5 Collection System and Overland Flows

The following curve types are available for 'SWMM5 collection system and overland flows' models:

- Storage: Defines the geometry of a Storage Unit node by describing the variation of surface area as a function of water depth

- Divider: Defines how the diverted outflow from a Flow Divider node is related to the total inflow.

- Tidal: Defines how the stage at an Outfall node varies by the hour of the day.

- Rating: Relates flow through an Outlet link to the head difference across the outlet

- Control: Sets how the control setting of a pump or flow regulator varies as a function of some control variable (such as water level at a particular node) as specified in control rule. Is also used to adjust the flow from an LID unit's underdrain based on head.

- Shape: Describes how the width of a customized cross-sectional shape varies with height for a Conduit link.

- Weir: Defines how a weir discharge coefficient varies with head.

- Pump curve: Relates flow through a pump to the depth or volume at the upstream node or to the head delivered by the pump. The following types of pump curves are available:

  - Pump Curve 1 (Volume-Flow). An off-line pump with a wet well where flow increases incrementally with wet well volume
  - Pump Curve 2 (Depth-Flow). An in-line pump where flow increases incrementally with inlet node depth
  - Pump Curve 3 (Head-Flow). An in-line pump where flow varies continuously with head difference between the inlet and outlet nodes
  - Pump Curve 4 (Depth-Flow). A variable speed in-line pump where flow varies continuously with node depth

- Depth dependent infiltration: Lookup table of Infiltration Rate and Depth values. Infiltration Curve input for 2D model infiltration input of type 'Varying in domain and flow dependent' defined in the 2D Infiltration editor.

- Depth dependent Manning (M) or Depth dependent Manning (n): Lookup table comprised or Depth-Manning M (or Depth-Manning n) value pairs. Used as Roughness Curve input for 2D model surface roughness input of type 'Varying in domain and flow dependent' that is a function of Depth defined in the 2D Surface Roughness editor.

- Flux dependent Manning (M) or Flux dependent Manning (n): Tabular data comprised or Flux-Manning M (or Flux-Manning n) value pairs. Used as Roughness Curve input for 2D model surface roughness input of type 'Varying in domain and flow dependent' that is a function on Flux defined in the 2D Surface Roughness editor.

7.2 Two-Dimensional Tables

The Two-Dimensional Tables editor is used for tables defining an output value as a function of two input parameters.

The two-dimensional tables with type 'Time varying control rule' may be used in control rules where a relationship between an input value and a return value (output) changes over time.

Each row represents a specific date and time, for which the return values (output values) are specified in the table in the cells with white background, as a function of the input values specified in the top row with dark background.
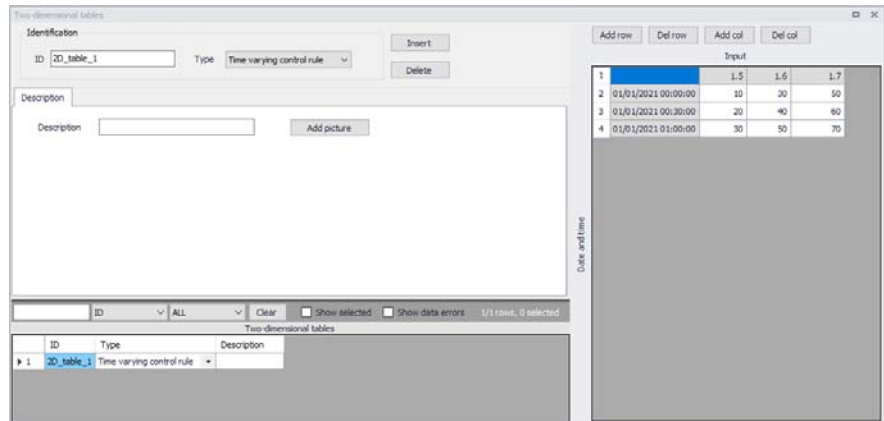

Figure 7.7   Two-Dimensional Tables editor

The two-dimensional tables with type 'Tabulated Q' or 'Tabulated h' are used by tabulated structures on river networks to define the relationships between the discharge in the structure and the upstream and downstream water levels.

The multiple two-dimensional tables with type 'Bridge' are used by bridge structures on river networks, to define the various coefficients that may be required for these structures.

The type of a table can only be selected when it is created, but cannot be changed afterwards.

7.3   Materials

In MIKE+, 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 7.8   Materials Editor

MIKE+ has the following pre-defined Material types with friction loss proper-
ties:

- Cement Mortar
- Ceramics
- Concrete (Normal)
- Concrete (Rough)
- Concrete (Smooth)
- Iron (cast)
- Iron (wrought)
- Plastic
- Stone

Table 7.3 Overview of the Materials Editor attributes

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 roughness	Yes If 'Colebrook-White' formulation is used	EQRough
H-W Coefficient	Hazen-Williams roughness coefficient	Yes If 'Hazen-Williams' is used	HWCoef
Description	User's descriptive information on the material	Optional	Description

7.4 Outlet Head Loss

MIKE+ models outlet head losses at nodes. The general flow equations are valid only for continuous conduits, where the only resistance to flow is bottom and sidewall friction. Hydraulic conditions in nodes, i.e. at manholes and structures, serve as boundary conditions in the computation of flows in conduits. In turn, hydraulic conditions in a node depend on the flows in the inlet and outlet conduits.

These hydraulic conditions, expressed in terms of the energy conservation principle, are calculated as water levels and velocity heads. The calculation is based on the mass continuity and formulation of more or less advanced energy relation between the node and the neighbouring links, including energy losses caused by local flow disturbances at different locations in the node.

The following parameters constitute a definition of head loss calculation option in MIKE+:

Computation Method

Three different methods are available:

- Classic. This is a simplified computational model for energy losses in junctions based on F.A. Engelund's energy loss formulae. The total head loss comprises of node inlet loss and node outlet loss, including losses due to change in flow direction, due to change in elevation, and due to contraction, if relevant.

- Mean Energy Approach. This is an alternative solution which fully ignores the energy loss at the inlet. For a flow-through manhole, this means that the energy level in the manhole is set equal as at the downstream end of the inlet pipe. For manholes with multiple inlets, the energy level is calculated as the weighted average of the inlet flows (i.e. large flows contribute most to the energy level). In this formulation, the total loss at the manhole is concentrated computationally at the outlet, and can be fully controlled by the user.

- No Head Losses. This option ignores all local losses. Regardless of the shape of the outlets, geometrical set-up of the junction and distribution of flows among inlet and outlet conduits, water levels in the junction and the outlet conduit are set equal as if there is no change of geometry and flow conditions between the junction and outlet conduit. This means that this option should be applied only where there is no change in cross section. Inconsistent results may be generated if inappropriately applied. This option is recommended for an artificial node/junction along a straight section of conduit, where no losses actually occur.

Further details on the various head loss computation methods mentioned above are found in the MIKE 1D Reference Manual.

Effective Node Area

This parameter is only relevant for the Classic head loss computational method. In all other cases, the default total wetted node area is applied. The following choices are available:

- Full Node Area. Calculated as product of diameter and water depth for manholes and read from the basin geometry table (Ac) for basins. Typically results in overestimate of local loss in a node.

- Calculated Effective Area. The effective area in a manhole is calculated based on an empirical formula (see Section 4.6.2 'Headloss calculation for inflowing water' section in the MIKE 1D Reference Manual). This results in a significantly smaller area than full wetted area and, consequently, with a more realistic flow calculation.

- Reduced Calculated Effective Area. The effective area in a manhole is further reduced to 50% of the calculated effective area.

Loss Coefficient

The available loss coefficient types distinguish three different interpretations of the specified head loss coefficient.

- Km. Interprets the specified value as the outlet 'shape' coefficient Km (see Equation (4.17) in MIKE 1D Reference Manual).

- Contraction HLC. Interprets the specified value as the outlet 'contraction' coefficient ζcontr(j) (see Equation (4.17) in MIKE 1D Reference Manual). This means that the model ignores the geometrical relations between the node and the outlet links (outlet shape), and applies the specified value directly as the ζcontr. The contraction losses in the outlet links are then computed by multiplying the velocity head in the respective link by the ζcontr. The total head loss for an outlet link is computed as a sum of the contraction, direction and elevation loss.

- Total HLC. Interprets the specified value as the total outlet head loss. This means that the model completely ignores the geometry of the node/links, and applies the specified value (Total HLC) directly as the ζout, the same for all outlet links at the node. The total head losses in the outlet links are then computed by multiplying the velocity head in the respective link by the specified ζout.

Changes to outlet head loss parameter sets used for Nodes are made via the Outlet Head Loss editor (Tables| Outlet Head Loss). Figure 7.9 also lists five pre-defined head loss parameter sets in MIKE+.

Figure 7.9   Outlet Head Loss editor

Table 7.4 Outlet head loss parameter set Editor attributes (Table msm_LossPar)

Edit field	Description	Used or required by simulations	Field name in datastructure
Head Loss ID	Unique head loss parameter set identifier	Yes	MUID
Method	Dropdown menu to select head loss calculation method for parameter set: - Classic - No Head Losses - Mean Energy Approach	Yes	OutletShapeNo
Effective Node Area	Choice of method for the calculation of wetted area: - Full Node Area - Calculated Eff. Area - Reduced Calculated Eff. Area	Yes If Method = Classic	EffAreaNo
Loss Coefficient [dropdown menu]	Definition of the interpretation of the head loss coefficient: -Km - Contraction HLC - Total HLC	Yes If Method = Classic or Mean Energy Approach	CoeffNo
[Field next to Loss Coefficient dropdown menu]	Value for the loss coefficient	Yes If Method = Classic or Mean Energy Approach	Coeff

7.5 On-Grade Captures

On Grade Captures tabular data are used as hydraulic properties for On Grade Curb Inlets describing the transfer capacity of connections. The On Grade Captures editor allows the user to group together QQ relations (tabular Curves and Relations data) that comprise a single On Grade Curb Inlet geometry (collective of QQ relations defining the capture rate as a proportion of approach flow). The transfer capacity for an On Grade Curb Inlet is dependent on the approach slope in the overland flow network, and a number of QQ relations can apply for a single curb inlet.
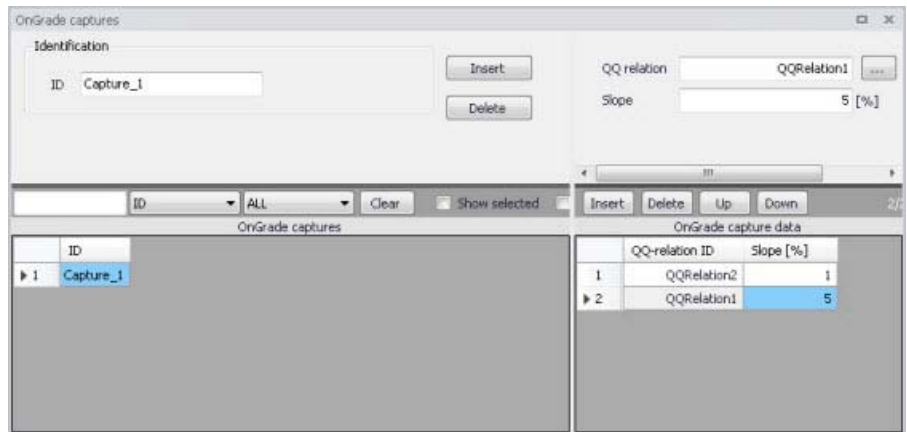
Figure 7.10   On Grade Captures editor

For calculated or user-defined slopes in the Curb Inlet dialog that are outside the range of slopes specified in the On Grade Captures table, the closest slope curve will be used. For intermediate calculated or user-defined slopes (lying between slope curves in the On Grade Captures table), linear interpolation is applied.

In the case of an On Grade Curb Inlet capacity that is not dependent on slope of the overland flow network, the user needs to define the On Grade Capture with a single QQ relation. *Note: In this case, the calculated or user-defined slope in the Curb Inlet dialog for On Grade Type will be ignored.*

8 Result Specifications

MIKE+ 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.

Note that this user guide focuses on 2D Overland model components, and aims to describe functionalities more specific to 2D overland model features. Details on parameters and options more related to catchment and network modelling may be obtained from the MIKE+ Collection System, MIKE+ River Network Modelling, and MIKE+ SWMM Modelling User Guides.

8.1 Result Files

The Result Files editor (Figure 8.1) provides a facility for viewing and specifying result file setups in a project. The types or results available depend on the active features and modules for the project (General Settings | Model type).

Result setups are a mix of Default and User-specified results.

Note that for models of type 'SWMM5 collection system and overland flows,' only results related to 2D overland features may be configured via the Result Files editor. Catchment and network model-related results for these types of models are configured in the Reporting tab page of the Simulation Setup editor. Consult the MIKE+ SWMM Modelling for more details on catchment and network modelling with SWMM in MIKE+.

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+.

Table 8.1   Overview of possible Default result files in MIKE+

Default ID	Model Type	Format	Content Type
Default_Surface_runoff	Catchments	.RES1D	Surface runoff
Default_RDII	Catchments	.RES1D	RDII

Table 8.1   Overview of possible Default result files in MIKE+

Default ID	Model Type	Format	Content Type
Default_Storm_wa-ter_quality	Catchments	.RES1D	Storm water quality
Default_Storm_wa-ter_sediments	Catchments	.RES1D	Storm water sedi-ments
Default_LIDs	Catchments	.DFS0 (currently hard-coded)	LIDs
Default_Catch-ment_discharge	Catchments	.RES1D	Catchment dis-charge
Default_Catch-ment_dis-charge_quality	Catchments	.RES1D	Catchment dis-charge quality
Default_Net-work_HD	Network	.RES1D	Hydrodynamic
Default_Net-work_RTC	Network	.RES1D	Real time control
Default_Net-work_AD	Network	.RES1D	Pollution transport
Default_Network_-MIKE_ECOLab	Network	.RES1D	MIKE ECO Lab
Default_LTS_ex-treme_statistics	Network	.RES1D	LTS extreme statis-tics
Default_LTS_chron-ological_statistics	Network	.RES1D	LTS chronological statistics
Default_2D_over-land	2D Overland	DFSU / DFS2	2D area
Default_2D_over-land_AD	2D Overland	DFSU / DFS2	2D area, AD
Default_2D_-Flood_statistics	2D Overland	DFSU / DFS2	2D flood statistics
Default_2D_Vol-ume_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.
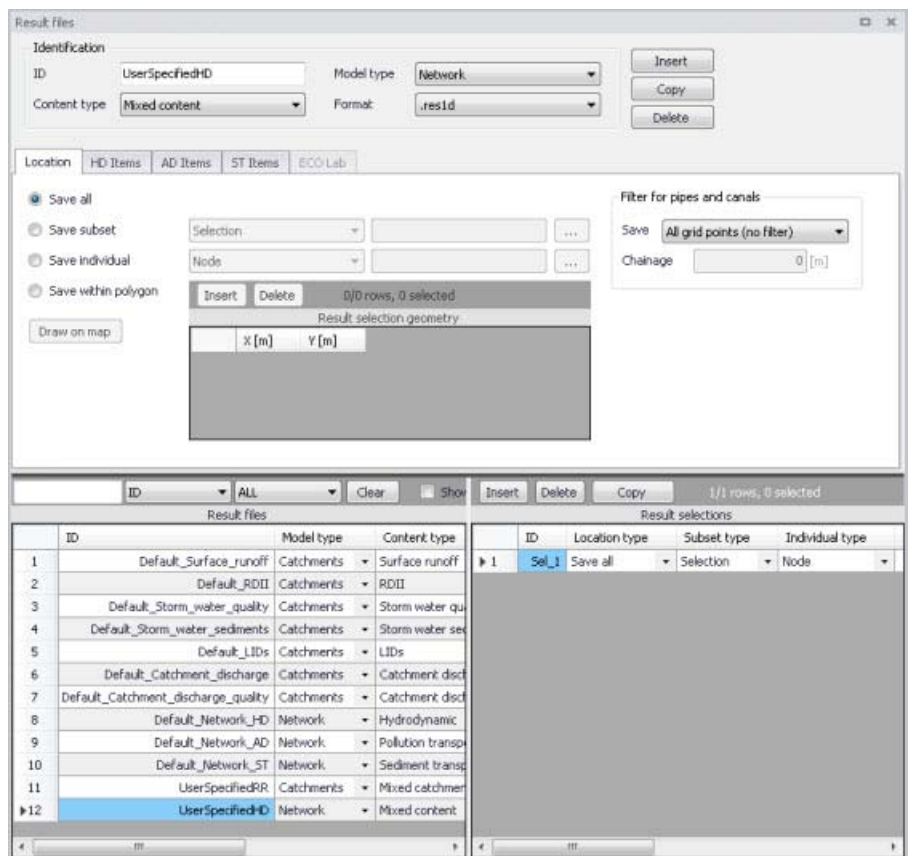
Figure 8.1   The Result Files editor

The various data tabs and components of the Result Files editor are described in succeeding sections.

8.1.1 Identification

The Identification group box on the Result Files editor contains general information on a result file item.
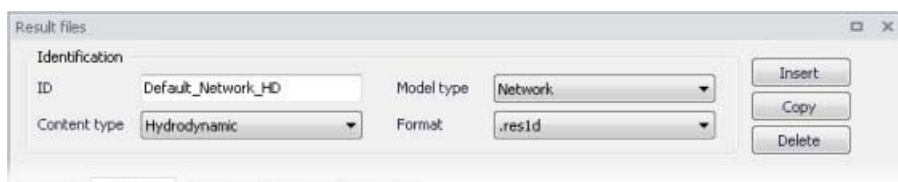

Figure 8.2   The Identification group box on the Result Files editor

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 culverts results: time series of results in selected culverts. The time series will save for each culvert the target discharge (discharge calculated using the empirical formulas), the discharge (effective discharge, which can be less than the target discharge if the upstream water depth is too low), accumulated discharge, and water level on both sides of the culvert (for a short culvert the left and right water level is the mean water level in the real wet elements to the left and right of the section of faces; for a long culvert the start and end water levels are the mean water level in the real wet elements to the right of the two end sections).
  - 2D weirs results: time series of results in selected weirs. The time series will save for each weir the target discharge (discharge calculated using the empirical formulas), the discharge (effective discharge, which can be less than the target discharge if the upstream water depth is too low), accumulated discharge, and water level on both sides of the weir (the left and right water levels are the mean water level in the real wet elements to the left and right of the section of faces).
  - 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
  - Section statistics: a *.dfs0 file with a relative item axis, showing the maximum water depth and/or maximum water level in each element face from the mesh, along a defined cross section. The axis of this result file is the index number of the element faces along the cross section.

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 8.2 below.

Table 8.2   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, Network model or 2D overland model	Yes	ModelTypeNo
Content Type	The type of result set under which the result file item falls under.	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

Adds a new result item in the Result Files table.

Copy

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

Deletes the current selected rows from the left overview table in the editor.

8.1.2   Location

The flexibility in results-saving in MIKE+ 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).

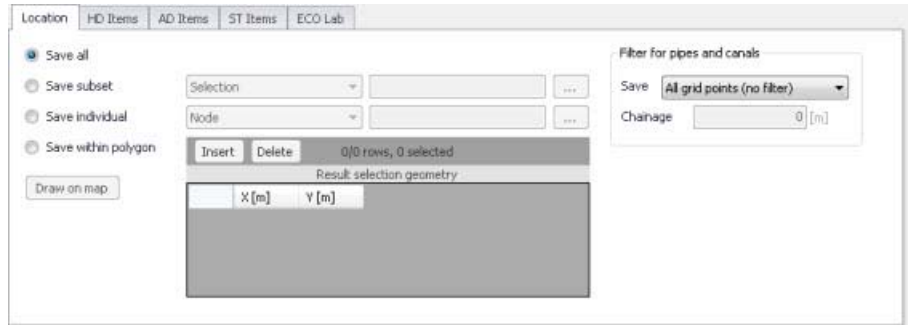Result saving locations are specified in the Location tab in the Result Files editor (Figure 8.3).

Figure 8.3   The Location Tab in the Result Files editor

Table 8.3   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

Table 8.3   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
[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 225) 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 8.4) when the subset is a 'Selection'.
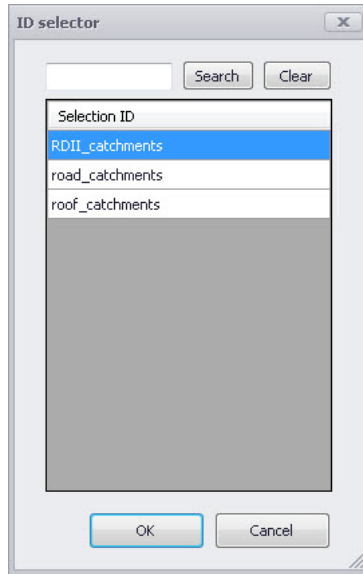
Figure 8.4   Selection lists in the ID Selector window

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 8.5).
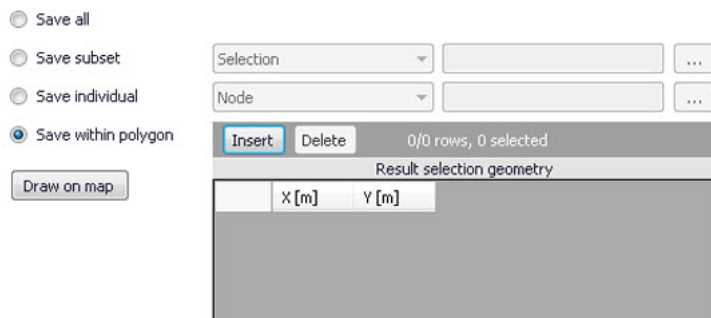
Figure 8.5   The Result Selection Geometry secondary grid in the Location Tab

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.
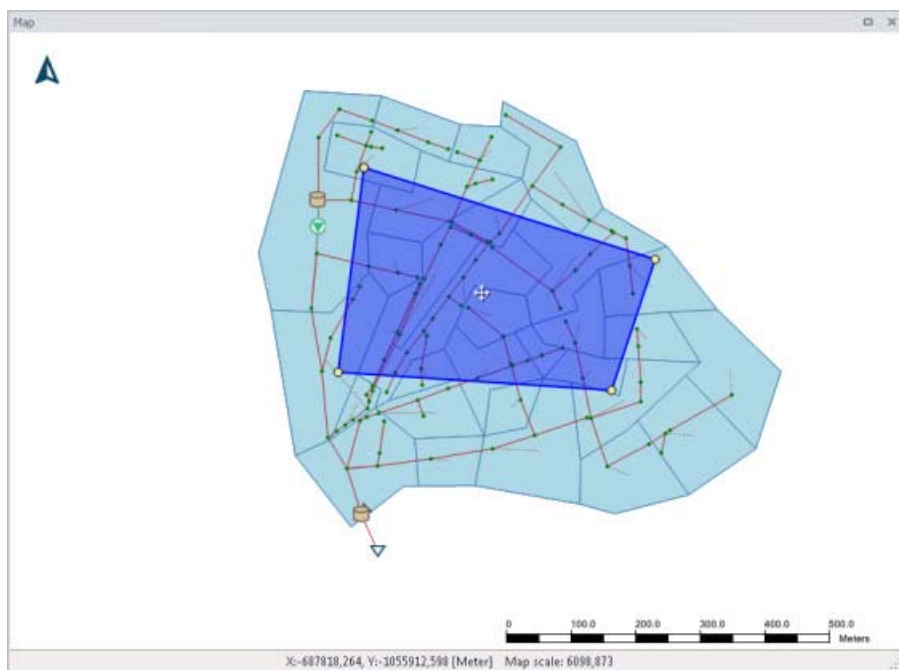

Figure 8.6   Defining a result location polygon on the Map

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.

For a 2D section statistics result file, coordinates must be specified for the cross section polyline, along which the maximum water depths and water levels will be reported.

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.

2D Culverts and Weirs List

For '2D culverts results' and '2D weirs results', the location is defined with a list of structures in a table.

Figure 8.7   The list of 2D structures

Use the buttons above the table to add or remove structures from the list.

Insert
Opens the list of valid structures, to select one to add to the table.

Delete
Deletes the current selected rows from the list.

Include all
Updates the list to save results for all 2D weirs or 2D culverts.

Exclude all
Removes all items from the list.

8.1.3   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 MIKE+ interface.
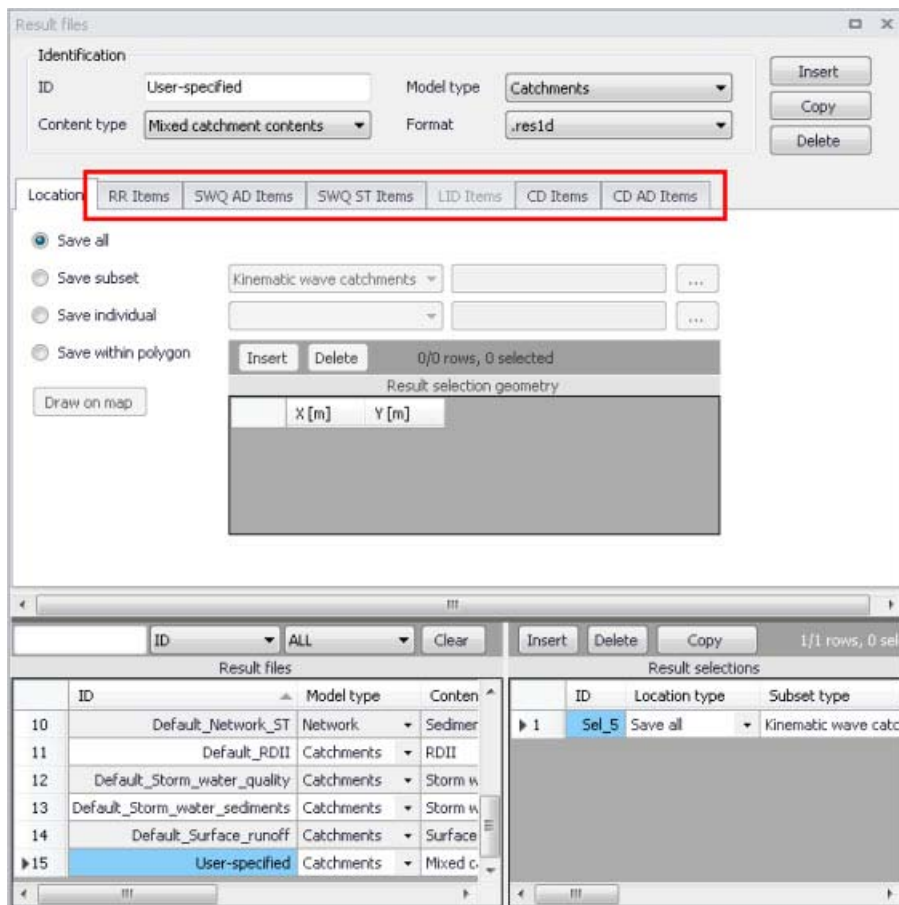
Figure 8.8   Item Tabs in the Result Files editor. This figure shows available tabs for
            results from Catchment models.

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.

8.1.4   Interpolation
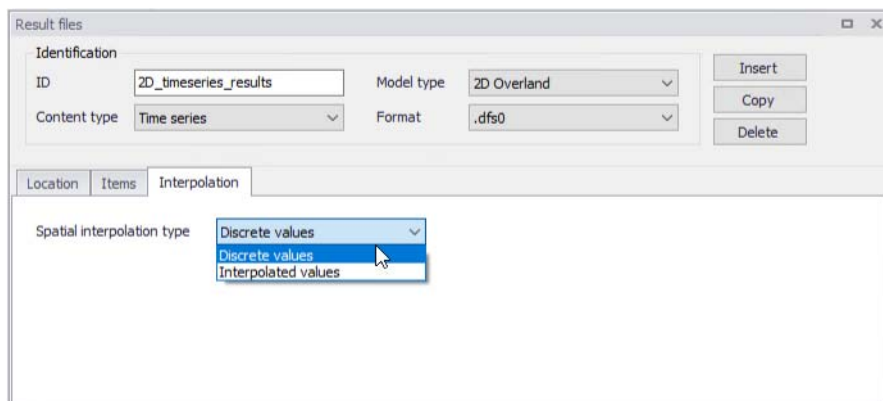
For 2D time series results, the spatial interpolation type must be specified.

Figure 8.9   The Interpolation tab page on the Result Files editor

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.

8.2   Simulation Summaries

Running an integrated model in MIKE+ produces separate simulation log and summary files for the network and 2D overland components in the project directory.

8.2.1   Integrated Model

An HTML file is generated from the model coupling engine at the end of an integrated model simulation. The file is named with the suffix '_mf.html', and prefixed with the MIKE+ project name.

For example, 'IntegratedFloodModelFinal_mf.html' is generated from a coupled model simulation from the MIKE+ model setup 'IntegratedModelFinal.mupp.'

Figure 8.10   Example of integrated model simulation summary

The file presents links to simulation log and summary files for the various components of the integrated model:

- 2D model simulation log
- Network model summary html
- Network model simulation log html

8.2.2   2D Overland Model

For the 2D overland model, a simulation volume balance summary is presented in a file suffixed '_m21fmVolumeBalance.html' and prefixed with the MIKE+ project name.

The file is overwritten for every simulation. An example of a 2D overland simulation volume balance file is shown below. Note that the values are only with respect to the 2D model component.

MIKE 21 Flow Model FM

C:\Users\mikeadmin\Documents\MIKE+ Examples\CS, Rivers, 2D\2D Blue Beach\100y_combined_m21fm.m21fm

Volume Balance Summary

A: Initial volume in model area					498752.91	m^3
Final volume in wet area					503795.58	m^3
Final volume in dry area					24.83	m^3
B: Final volume in model area					503820.42	m^3
Source inflow	0.00	m^3				
MIKE 11 inflow target	106000.23	m^3				
MIKE 11 inflow correction	-0.00	m^3				
MIKE Urban inflow target	18664.85	m^3				
MIKE Urban inflow correction	0.00	m^3				
Source outflow			0.00	m^3		
MIKE 11 outflow target			115059.06	m^3		
MIKE 11 outflow correction			-8.03	m^3		
MIKE Urban outflow target			3411.89	m^3		
MIKE Urban outflow correction			-0.00	m^3		
C: Total volume from source					6202.16	m^3
D: Total volume from precipitation/evaporation					0.00	m^3
E: Total volume from boundaries					-1134.66	m^3
F: Continuity balance (B-A-C-D-E)					-0.00	m^3

Figure 8.11   Example of a 2D model volume balance summary from a MIKE+ coupled model simulation with a 2D model component

The volume balance is broken down into the following items:

- Initial volume in model area (open boundary cells are not taken into account)

- Final volume in model area

- Total Volume from Boundaries: Inflow through open boundaries

- Total Volume from Source: Balance of inflows and outflows to/from external model sources, e.g. coupled models. At present, the items are grouped and categorized separately for the river model and collection system model components:

- Inflow Target: Expected inflow from the coupled network model source to the 2D model through the coupling
- Inflow Correction: Correction of inflow through the coupling to the 2D model as determined by the 2D model
- Outflow Target: Estimated outflow from the 2D model to the coupled network model source through the coupling
- Outflow Correction: Correction of outflow through the coupling from the 2D model to the coupled network model as determined by the 2D model

- Total Volume from Precipitation/Evaporation: The contribution from precipitation and evaporation. This amount considers losses through Infiltration, if active.

Deviations in the continuity balance (Relative deficit) may be caused by the numerical scheme and should remain as low as possible.

Additionally, deviations in volume balance may also be caused by corrections of water volumes during the simulation (e.g. Inflow and Outflow Corrections). These corrections represent artificial volumes of water added in the model in different situations:

- A large time step may give rise to water level below the bed level in a cell. If this happens the 2D model will correct the water level to just above the bed (0.0002 m default), by adding an artificial volume of water.

- Sinks/ links (River or MIKE+ Collection System) at dry spots draining more water than present.

- Extraction of water from a dry section of the model area through the river model, MIKE+ Collection System, or evaporation

- Steep slopes in the model area drying out uphill cells

The total correction of water volume in the model is reported in the volume balance as 'Inflow / outflow correction' (Flexible mesh model). These corrections should be checked after the simulation, as high corrections may affect the results and may be due to issues in the model setup.

8.2.3   Network Model

SWMM Collection System and Overland Flows

For models of type 'SWMM5 collection system and overland flows,' network simulation summaries are configured via the Reporting tab page on the Simulation Setup editor. Please refer to the MIKE+ SWMM Modelling User Guide for more details on simulation summary specifications for SWMM collection system models.

Rivers, Collection System and Overland Flows

For 'Rivers, collection system and overland flows' models, the basic contents of HTML summaries generated at the end of network model simulations may be customised in MIKE+.

The HTML file is named with the suffix '_Summary.html', and prefixed with the Simulation and Scenario IDs in MIKE+.

Various Network Summary configurations may be defined through the Network Summary editor (Result Specifications | Network Summary) (see). These configurations may then be invoked in the Simulations Setup editor when running Network (HD) simulations.

The basic HTML summary shows information on the simulation, overall results summary, and boundary inputs. Adding a network summary configuration will extend the basic summary with the 'HD Results Summary' section.

Additional items may be activated for each network summary setup in the editor. The Network Summary editor is organized into the following groups:

- Identification
- Tables with min, max and accumulated values for
- Summary of input data
- Use selection to reduce summary tables

Activate the various options on the editor to customise additional contents for the network simulation HTML summary.

Use the following button functions on the editor to add or remove network summary setups for the project:

Insert
Adds a new Network Summary setup in the editor.

Delete
Removes the active/selected Network Summary setup from the editor.

Please consult the MIKE+ Collection System User Guide for more details on the Network Summary items for network model simulations.

9 Simulation Specifications

MIKE+ 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.

9.1 Simulation Setup

The Simulation Setup editor has several tabs, which are shown depending on the active features and 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.

- **Catchments**. Includes parameters specific for Runoff simulation.

- **HD**. Includes parameters specific for HD simulation.

- **AD and WQ**. Includes parameters specific for network AD simulation and MIKE ECO Lab.

- **LTS**. Includes parameters specific for Network LTS simulation.

- **Results**. Includes specification of results (output) to be generated by the simulation.
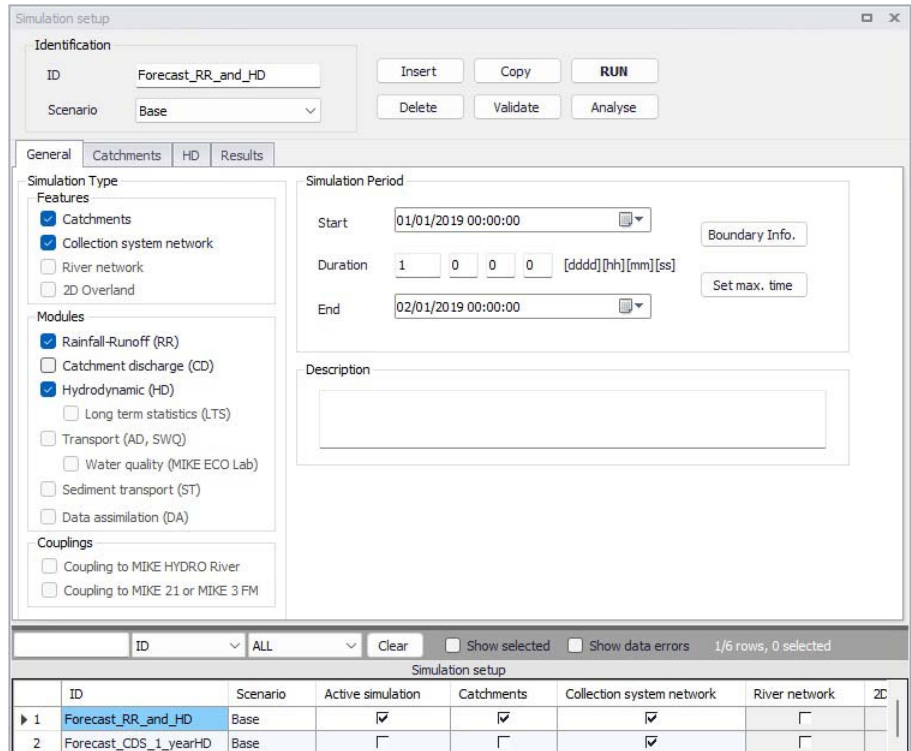
Figure 9.1   The Simulation Setup Editor

The Identification group at the top and the scrollable grid table at the bottom of the editor are common across all tabs.

Table 9.1   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 9.1   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 simulation check box	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 ID.

'Copy' button
Duplicates an existing (currently active) simulation setup record.

'Delete' button
Deletes a currently active simulation record.

'RUN' button
Triggers export of the current simulation job and execution of the simulation.

'Validate' button
MIKE+ employs automatic validation, checking for data errors as one navigates through the various editors. However, one may also collectively check the consistency of the MIKE+ project data through the validation of the simulation. Press the 'Validate' button to open the validation window:
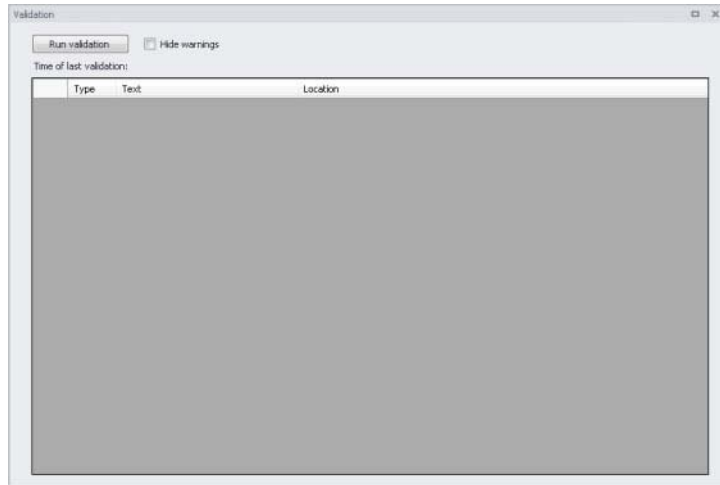
Figure 9.2   The Simulation Validation window

The validation applies to the simulation set as 'Active simulation' in the 'Simulation setup' editor. From this window, press the 'Run validation' button to start the data validation and see the list of warnings and errors in the table. The validation time depends on the database size, number of active modules and number of errors found.

A description of the warning or error is included in the 'Text' column in the list. The description usually indicates the data type or editor in which the error should be corrected.

Figure 9.3   The validation list indicates whether an item is an Error (highlighted in red) or a Warning (highlighted in yellow).

It is possible to activate the 'Hide warnings' option to only show errors without warnings in the list.

'Analyse' button

This button opens a tool that can be used to find elements of a river or CS network that may be contributing to instabilities during an HD simulation. The tool produces a list of h-points ranked according to one of the three following criteria, all ranked from smallest to largest:

- Time step required to achieve a specified stability criterion

- Surface area

- Length

Figure 9.4   The Analyse network tool.

More information about the three options is provided below.

Time step required to achieve a specified stability criterion

When using this option, the tool finds the time step needed to achieve a specified Courant number for each h-point. This option can be used to find an appropriate time step for a stable simulation or identify parts of the model net-

work that should be streamlined to enable a stable simulation at a longer time step.

Because the Courant number evaluation depends on the depth of water, a depth definition must be provided. Four options are available:

- User-defined global value: single value applied throughout network.
- Full-running pipe or river (min. of markers 1 and 3): for rivers, this option uses the minimum of markers 1 and 3 as water level in each cross section. A full-running pipe is used for collection system modelling.
- Full-running pipe or river (min. of markers 1 and 3): for rivers, this option uses the minimum of markers 4 and 5. If markers 4 and 5 are not defined, then the minimum of markers 1 and 3 is used. A full-running pipe is used for collection system modelling.
- Initial conditions: Water depth definitions are taken from the initial conditions specification.

Surface area

Under this option, the tool computes the surface area associated with each h-point, in other words the surface area from the upstream Q-point to the downstream Q-point (or from the calculation point to the nearest upstream or downstream Q-point, if the calculation point is located at the start or end of a river branch).

A depth is needed to compute a surface area. The same four options are available as for 'Time step required to achieve a specified stability criterion'.

For rivers, the area is calculated assuming a constant width equal to the width of the cross-section at the specified depth. For collection system modelling, the area is calculated using a very small width corresponding to a pipe that is almost full. The flow area criterion is not particularly useful for analysis of pipes.

Length

Under this option, the tool computes the length associated with each h-point, in other words the length from the upstream Q-point to the downstream Q-point (or from the calculation point to the nearest upstream or downstream Q-point, if the calculation point is located at the start or end of a river).

9.1.1 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 9.5   The Simulation Setup Editor General tab

An overview of the editor fields and corresponding database attributes is pro-
vided in Table 9.2 below.

Table 9.2   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	Enable_Catchment
Collection system network	Activates CS related simulation type	Yes	Enable_CS
River network	Activates river related simulation type	Yes	Enable_River
2D Overland	Activates 2D related simulation types	Yes	Enable_2DOverland
Rainfall-Runoff (RR)	Activates runoff simulation	Yes	Enable_RR
Catchment Discharge (CD)	Activates catchment discharge computations	Yes	Enable_CD
Hydrodynamic (HD)	Activates HD simulations for CS, Rivers and/or 2D overland	Yes	Enable_HD

Table 9.2 Overview of the Simulation Setup General tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Long Term Statistics (LTS)	Activates LTS simulation	Yes	ENable_LTS
Transport (AD, SWQ)	Activates AD simulations for CS, Rivers and/or 2D overland and/or SWQ simulations for catchments	Yes	Enable_AD
Water quality (MIKE ECO Lab	Activates MIKE ECO Lab simulations for CS, Rivers and/or 2D overland	Yes	Enable_ECOLAB
Sediment Transport (ST)	Activates CS network sediment transport simulation	Yes	Enable_ST
Data Assimilation (DA)	Activates DA network update simulation	Yes	Enable_DA
Coupling to MIKE HYDRO River	Activates coupling to MIKE HYDRO River. HD coupling is always activated. AD coupling is activated if Transport (AD,SWQ) module is also activated.	Yes	Enable_MHRiver_Coupling
Coupling to MIKE 21 or MIKE 3 FM	Activates coupling to MIKE 21 FM or MIKE 3 FM.	Yes	Enable_M21FM_Coupling
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 9.2   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.
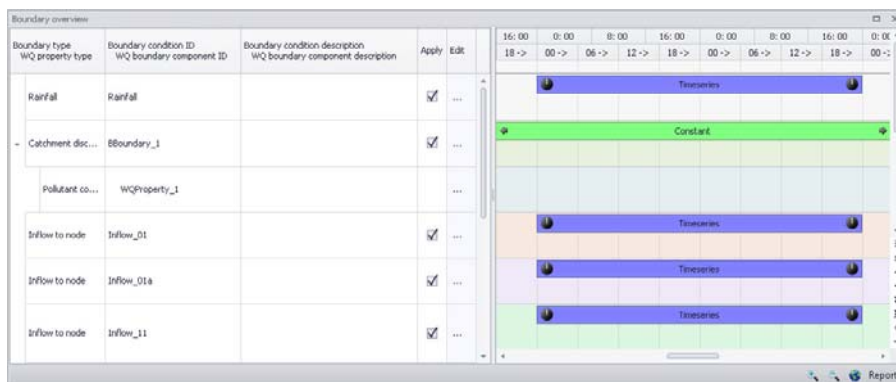

Figure 9.6   The Boundary Overview appears when pressing the 'Boundary Info.' button

'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.

9.1.2 Catchments

The following parameters can be specified on the Catchments tab:

- Surface runoff model simulation time step
- RDI simulation time step
- Catchment discharge simulation time step
- Spatial distribution of rainfall
- RDI hotstart specification
- Additional parameters specification

Figure 9.7   The Simulation Setup Catchments Tab

Table 9.3　　The Simulation Setup Catchments tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Time-Area	Fixed time step for Time-Area runoff model	Yes if at least one catchment is set for simulation with Time-Area model	TSDt
Kin. Wave	Fixed time step for Kinematic Wave runoff model	Yes if at least one catchment is set for simulation with Kinematic Wave model	KWDt
UHM	Time step for UHM model	Yes if at least one catchment is set for simulation with UHM runoff model	UHMDt
Wet weather	Time step during wet periods	Yes if at least one catchment is set for simulation with Linear Reservoir runoff model	DtWetPeriod
Dry weather	Time step during dry periods	Yes if at least one catchment is set for simulation with Linear Reservoir runoff model	DtDryPeriod
RDI	RDI slow runoff component time step. The fast runoff component time step is the corresponding surface runoff model time step.	Yes if at least one catchment is set for simulation with RDI runoff model	SRCDt
Catchment Discharge	Time step for catchment discharge	Yes if Catchment Discharge simulation active	CDDt

Table 9.3   The Simulation Setup Catchments tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Distribution type	When rainfall boundary conditions are defined with the geographical location of the rain station (boundary type 'Data source location'), it is possible to control how the rainfall intensity from the various rain gauges (boundary conditions) are applied to the different catchments. Three methods are available: assign the closest rainfall time series to each catchment, apply weighting using Thiessen polygons, or apply weighting using an inverse distance interpolation. Weights can be reviewed in the 'Boundary conditions' editor.	Yes, if rainfall boundary conditions are of spatial extent type 'Data source location'	DistribTypeNo
Power parameter	The power coefficient, in the inverse distance interpolation equation.	Yes, when the distribution type is set to 'Inverse distance weighting'.	Power
Maximum search radius	The maximum distance for searching rainfall stations, in the inverse distance interpolation equation.	Yes, when the distribution type is set to 'Inverse distance weighting'.	SearchRadius
RDI Hotstart Apply checkbox	If this checkbox is ticked, a hotstart file for RDImust be specified.	Optional	RDIHotStartNo

Table 9.3   The Simulation Setup Catchments tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Use simulation start time	If 'Use simulation start time' option is enabled, the initial condition is automatically extracted from the hotstart file using the simulation start date and time as specified in the 'Simulation Setup' editor.	Optional	RDIHotStart-TimeUseNo
RDI hotstart time	The date and time at which the initial conditions are loaded from the hotstart file. Only active if 'Use simulation start time' option is inactive.	Optional	RDIHotStartTime
Additional Parameters Apply checkbox	If this checkbox is ticked, an *.ADP file for the runoff simulation can be specified	Optional	ADPRunoffFileNo

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.

9.1.3   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 9.8   HD Tab of the Simulation Setup Editor

For network simulations, the initial condition definition may be of the following types:

- Empty network: with this option, the network will be empty at the first time step of the simulation, except at outlet nodes where water levels from possible boundary conditions will propagate into the network.

- User-specified: with this option, a set of initial conditions must be selected. Sets of initial conditions contain definition of default values, local values, and hotstart files. They are defined in the 'Initial conditions' editor.

- State file: with this option, initial conditions for all modules included in the simulation (Hydrodynamic, Rainfall-runoff, Transport, etc.) are obtained from a state file, which is a detailed result file from a previous simulation. It is therefore required that state files have been saved during a previous simulation (see 13.1.1 Identification *(p. 418)*). For this type of initial condition, it is required to provide a path to the folder containing the state files created from the previous simulation, along with a date and time controlling which time step (i.e. which state file) will be used as initial condition. The appropriate state file from the selected folder, with the date and time the closest to the specified instant, will be selected automatically during the simulation. If 'Select state file using simulation start time' is ticked, the state file with the date and time the closest to the simulation start date will automatically be selected, otherwise the state file with the date and time the closest to the user-defined date and time will be used. A set of 'Initial conditions ID' must also be selected: initial conditions from the state file have the highest priority, but if the network has e.g. been extended and some calculation points do not exist in the state file, the initial conditions in these points will be controlled by the user-defined initial conditions.

- Steady state: with this option, initial conditions will be based on a steady state calculation for all rivers in the model, except for locations where local user-defined values are defined in the optional user-defined 'Initial conditions ID'. Default values and hotstart files defined in the selected 'Initial conditions ID' are ignored. This option is not supported for collection system networks.

Table 9.4   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

Table 9.4 The Simulation Setup HD tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Max. Increase Factor	Specifies maximum increase factor for adaptive time step for network model	Yes if Adaptive time step type	HDDtIncreaseFactor
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
Network initial conditions type	Specifies the type of initial condition for the network (CS and/or river)	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 except if defined as 'Empty network'	HDInitCondID
State files folder	The path to the folder containing the state files saved during a previous simulation	Yes if 'State file' is chosen	StateFilesFolder
Select state file using simulation start time	If selected, the simulation will use the state file with date and time matching the start time of the simulation.	Yes if 'State file' is chosen	UseSimulationTypeNo
Date and time	The user-defined date and time to select the state file from all state files saved during a previous simulation	Yes if 'Select state file using simulation start time' is not chosen	StateStartTime
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.

9.1.4 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 Transport (AD, SWQ) module is activated in the 'General' tab, otherwise it is hidden.

Figure 9.9 The Simulation Setup AD and WQ Tab

Table 9.5 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

Table 9.5   The Simulation Setup AD and WQ tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
Type	Controls whether the simulation start with an empty network (no component) or with pre-defined initial conditions	Yes	ADInitContTypeNo
AD initial condition ID	The ID of the selected set of Network AD initial conditions	Yes if Type = User-specified	ADInitCondID
Decoupling of HD and AD simulations	Decoupling a transport simulation (AD or WQ) from the HD simulation speeds up the simulation by getting the hydrodynamic conditions from a decoupled result file, instead of running the HD simulation at the same time. This is especially relevant when running multiple transport simulations with identical HD conditions.	Yes	DecouplingADHD-TypeNo
Decoupling file	The decoupled result file to be used as input for the decoupled transport simulation. It is a result file with content type 'Decoupling' which must be saved during a previous simulation.	Yes, if 'Run transport simulation using decoupling file' is chosen	DecouplingPath
MIKE ECO Lab Integration	Specified ECOLab integration method	Yes If Simulation Type = MIKE ECO Lab (WQ)	ELIntegrationNo
Time step multiplier	Multiplier used to scale the MIKE ECO Lab time step as a function of the HD time step	Yes	ELUpdateFre-quency

9.1.5   LTS

The LTS tab includes parameters specific for Network LTS simulations. The tab is shown if the Long Term Statistics (LTS) module is activated in the project and if at least 1 Job List Criterion is specified.
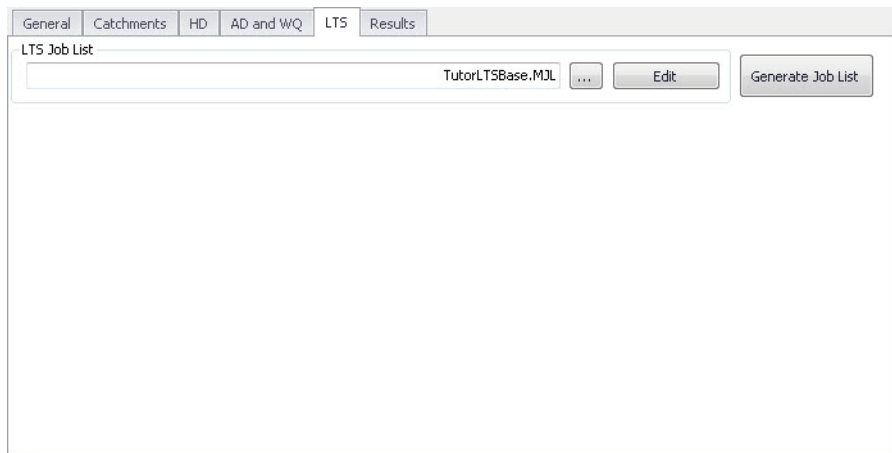

Figure 9.10   The Simulation Setup Editor LTS Tab

Table 9.6   Simulation Setup LTS tab attributes (Table msm_Project)

Edit field	Description	Used or required by simulations	Field name in datastructure
LTS Job List	Defines the Job List *.MJL file name and path to be used in the LTS simulation	Yes If Simulation Type = Long Term Simulation (LTS)	MJLFileName

The LTS tab also has buttons for the following functions:

'Edit' button
Opens the specified Job List file (*.MJL) in text editor. If the Job List File dialog is empty, or the specified file does not exist or is empty, the 'Edit' button opens an empty ASCII file.

'Generate Job List' button
Starts a Job List generation process. This is a special form of simulation where the output is a Job List file (*.MJL).

More details on Lob List generation and editing are found in the Long Term Statistics (LTS) User Manual.

9.1.6   Results

The Results tab includes parameters for defining output from a simulation setup.

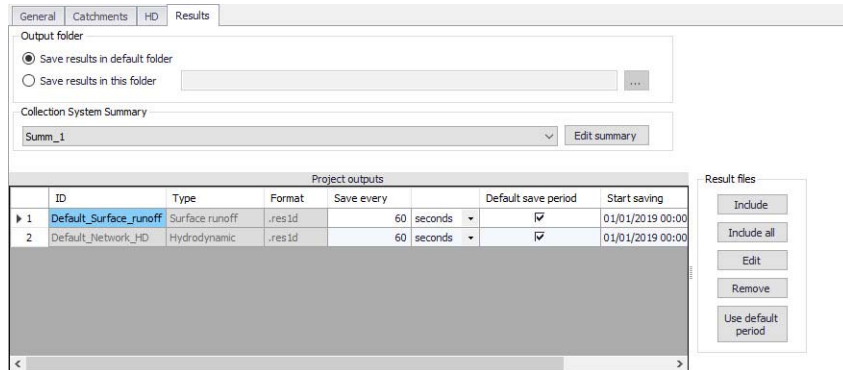Multiple result files may be specified for each simulation setup.

Figure 9.11   Results Tab of the Simulation Setup Editor showing multiple result items

Table 9.7   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 9.7   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 9.12). The grid retrieves the information from the Result Files editor (Result Specifications| Result Files).

Figure 9.12   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 9.8 below.

Table 9.8   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.

9.2 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.
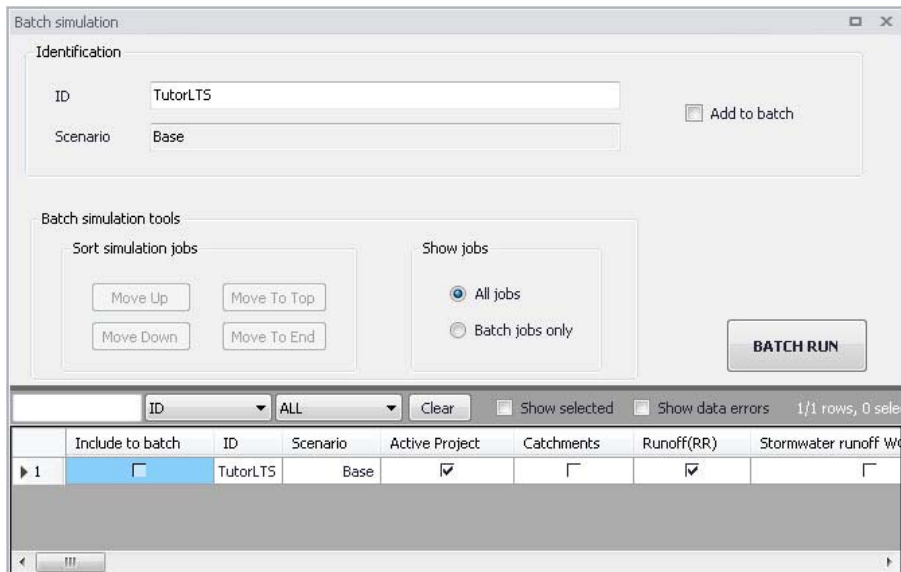
Figure 9.13   The Batch Simulation Editor

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 9.9   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.

Powering **Water Decisions**